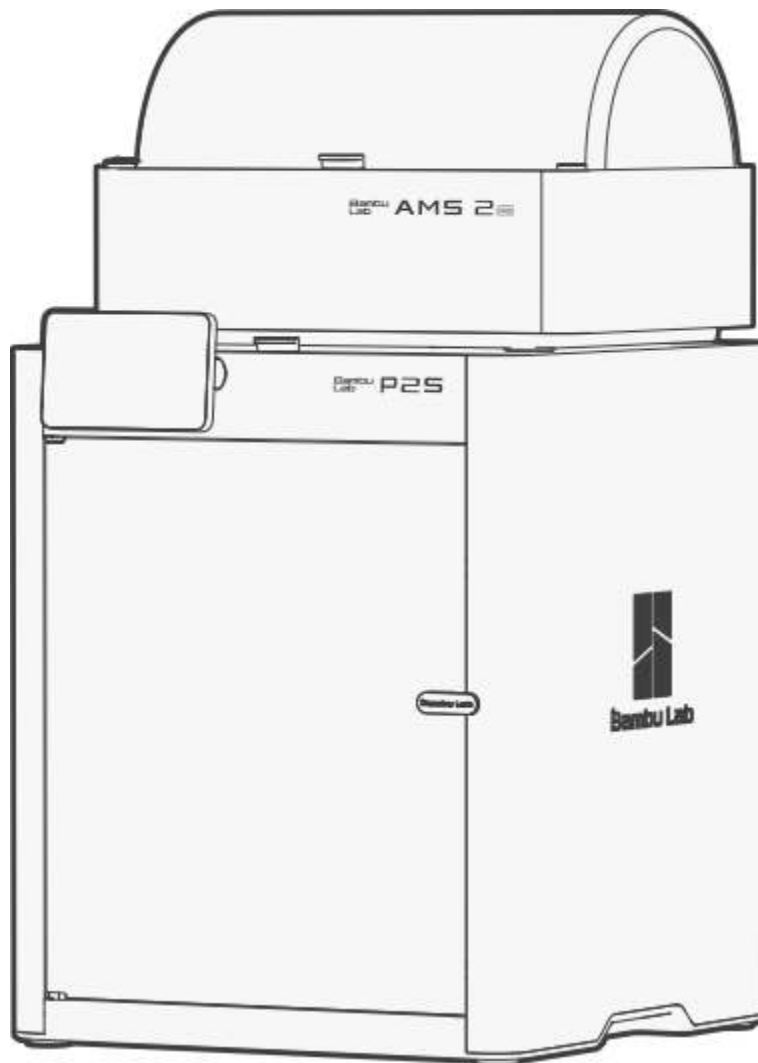


Bambu Lab P2S

3D Printing User Manual

V1.0 2026.05



Copyright Notice

This manual and its contents are intended solely to guide users in operating and using this product. The intellectual property rights belong to Shenzhen Bambu Lab Technology Co., Ltd. (hereinafter "Bambu Lab"). Without written permission from Bambu Lab, no part of this manual may be copied, distributed, modified, or provided to third parties in any form.

Manual Scope

This manual mainly introduces the P2S 3D printer. Some accessories or features mentioned in certain chapters are optional or upgrades and may not be included in your product package. Please refer to your specific model and configuration when studying the content and operating the printer.

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

Version and Updates

After a firmware update, the printer touchscreen will display information about new features. You can click the link below or visit Bambu Lab Wiki and navigate to **P Series > P2S > Printer Features > Firmware release history for the P2S** for details. If anything in this manual conflicts with the firmware release notes, the release notes take precedence.

wiki.bambulab.com/p2s/manual/p2s-firmware-release-history

Thank you for choosing the Bambu Lab P2S 3D Printer!

This manual provides essential information for using the 3D printing functions, including device setup, operation, and routine maintenance. To ensure safe, correct, and efficient use, read this manual carefully and make sure you understand it before use.

How to Use This Manual

Use the table of contents and in-document links for quick navigation.

Search this PDF for keywords.

Visit Bambu Lab Wiki (wiki.bambulab.com) and search for keywords within the chapter for more detailed steps and videos.

Quick Start

1. Go through the [Read Before Use](#).
2. Start your [first print](#).
3. [Access 3D model resources](#).
4. [Adjust slicing parameters](#).
5. Review [Print Quality Issues and Solutions](#) and [Printer Issues and Solutions](#).

Additional Resources

- **Unboxing and installation:** See the unboxing guide (wiki.bambulab.com/general/unboxing-guide) for printer unboxing and installation instructions.
- **Customized Courses:** Visit Bambu Lab Academy (bambulab.com/support/academy) for structured courses on the printer and software.

Feedback

If you have any questions or suggestions about this user manual, use the following link or scan the QR code to send feedback.



bambulab.com/support/documentation/feedback

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 printer performance. Proceed carefully to avoid potential issues.

DANGER

Potential hazards. Ignoring this may cause equipment damage or personal injury. Always follow the related instructions.

Technical Support

If you encounter issues during use, check this manual first. You can also visit Bambu Lab Wiki and search for troubleshooting steps, or use Bambu AI (support.bambulab.com) for maintenance tips, diagnostics, 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	9
Chapter 1 Introduction	10
1.1 Product Appearance.....	11
1.1.1 Printer.....	11
1.1.2 Toolhead.....	14
1.2 Screen Icons.....	14
1.2.1 Homepage.....	15
1.2.2 Control.....	15
1.2.3 Filament.....	16
1.2.4 Settings.....	16
1.2.5 Message.....	17
1.3 HMS Messages.....	17
Chapter 2 First Print	18
2.1 Preparation Before Printing.....	18
2.1.1 Printer Placement.....	18
2.1.2 Initial Installation.....	18
2.1.3 Initial Calibration.....	19
2.2 Load filament.....	20
2.2.1 From AMS.....	20
2.2.2 From the External Spool.....	21
2.3 Start a Print from Printer Screen.....	22
2.4 Remove the model.....	22
2.5 Unload Filament.....	24
2.5.1 From AMS.....	24
2.5.2 From the External Spool.....	24
2.6 Waste Disposal.....	24
Chapter 3 Access 3D Model Resources	26

3.1 MakerWorld Model Community.....	26
3.2 MakerLab Creative Tools.....	26
Chapter 4 Initiate a Print from Multiple Devices.....	29
4.1 Initiate a Print from Bambu Handy.....	29
4.1.1 Install Bambu Handy.....	29
4.1.2 Initiate a Print.....	30
4.1.3 Device Page Introduction.....	32
4.2 Initiate a Print from Bambu Studio.....	32
4.2.1 Install and Bind Bambu Studio.....	33
4.2.2 Download and Import Model.....	34
4.2.3 Initiate a Print.....	35
4.3 Initiate a Print from Printer Screen.....	36
Chapter 5 Adjust Slicing Parameters in Bambu Studio.....	37
5.1 Introduction to Bambu Studio Interface.....	37
5.1.1 Homepage.....	37
5.1.2 Prepare Page.....	37
5.1.3 Preview Page.....	39
5.1.4 Device Page.....	39
5.2 Model Size.....	40
5.3 Quality Settings.....	41
5.4 Strength Settings.....	44
5.5 Support Settings.....	48
5.6 Speed Settings.....	55
Chapter 6 Key Features Introduction.....	57
6.1 Intelligent Vision Detection.....	57
6.2 Adaptive Airflow System.....	59
6.2.1 Cooling Mode.....	59
6.2.2 Heating Mode.....	59
6.3 Auto Leveling and Calibration.....	60
6.3.1 Auto Bed Leveling.....	60

6.3.2 Flow Dynamics Calibration.....	60
Chapter 7 Basic Controls and Functions.....	61
7.1 Common Screen Operations.....	61
7.1.1 Print Speed.....	61
7.1.2 XYZ Axes Movement.....	62
7.1.3 Nozzle and Extruder.....	62
7.1.4 Heatbed and Chamber Temperature.....	62
7.1.5 Chamber Light Mode.....	63
7.1.6 Sound.....	64
7.2 Photo and Video.....	64
7.2.1 Video Recording.....	65
7.2.2 Photo.....	65
7.2.3 Timelapse.....	66
7.3 Connect to Network.....	67
7.3.1 LAN Only Mode.....	68
7.3.2 Developer Mode.....	68
7.4 Connect USB Flash Drive.....	69
7.4.1 Specifications.....	69
7.4.2 Connect and Format.....	69
7.4.3 Eject.....	70
7.5 Update and Restore.....	70
7.5.1 Update Firmware.....	70
7.5.2 Initialization.....	70
Chapter 8 Filament Introduction and Usage.....	71
8.1 Common Filament Types.....	71
8.2 Printing Conditions.....	74
8.3 Filament Preparation.....	76
8.3.1 Filament Drying.....	76
8.3.2 Filament Calibration.....	78
8.4 Special Filament Printing Requirements.....	80

8.4.1 TPU.....	80
8.4.2 High-Temperature Filament.....	82
Chapter 9 Print Quality Issues and Solutions.....	84
9.1 Common Print Quality Issue Overview.....	84
9.2 First Layer Not Sticking.....	85
9.3 First Layer Too High/Too Low.....	85
9.4 Poor Overhang Quality.....	86
9.5 Model Warping, Falling Off, or Collapse.....	87
9.6 Filament Sticking to the Nozzle.....	87
9.7 Under-Extrusion.....	88
9.8 Stringing and Oozing.....	89
9.9 Gloss Difference.....	90
9.10 Interlayer Cracking.....	91
9.11 Seam.....	91
9.12 Top Layer Gaps.....	92
Chapter 10 Printer Issues and Troubleshooting.....	94
10.1 Toolhead Clog Troubleshooting.....	94
10.2 Hotend Unclogging.....	95
10.2.1 Operation Overview.....	95
10.2.2 Unclog by Manual Extrusion.....	96
10.2.3 Unclog with Unclogging Pin.....	97
10.2.4 Unclog by Cold Pull.....	98
10.2.5 Unclog with Allen Key.....	99
10.3 Loose Belt.....	101
10.3.1 XY Belt Tension.....	101
10.3.2 Z Belt Tension.....	102
Chapter 11 Regular Care and Maintenance.....	105
11.1 Regular Maintenance.....	105
11.1.1 Maintenance Frequency.....	105
11.1.2 Print Calibration.....	106

11.1.3 Air Filter.....	106
11.1.4 XYZ Axes.....	107
11.1.5 Heatbed.....	109
11.1.6 Build Plate.....	109
11.1.7 Toolhead.....	110
11.1.8 Live View Camera.....	111
11.2 Replace Accessories.....	111
11.2.1 Hotend Silicone Sock.....	111
11.2.2 Hotend.....	112
11.2.3 Filament Cutter Blade.....	113
11.2.4 Nozzle Wiping Pad.....	114

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

- Verify that the printer's operating voltage matches the specified requirements to avoid damage or safety hazards. Check the label next to the power socket for specific voltage requirements.
- The printer operates at high temperatures and speeds. Do not allow children or unauthorized personnel to operate the printer to prevent burns, pinching, or other accidents.
- Do not touch the toolhead, heatbed, or motion parts during printing to prevent burns or mechanical injuries.

Operation and Maintenance Recommendations

- To ensure proper functioning of the printer's internal precision mechanisms, regular maintenance is recommended (see [Regular Care and Maintenance](#)).
- For the best printing results, we recommend using Bambu Lab filaments. These filaments have undergone rigorous compatibility, safety, and stability testing based on the product's characteristics.
- Do not load TPU filaments (except TPU for AMS) through the printer's filament buffer. See [TPU Printing Requirements](#) for recommended loading methods.
- Unless otherwise specified, always unplug the power cord before any operation, maintenance, or modification to prevent electric shock or equipment damage.
- Unless otherwise specified, allow the printer to fully cool down before any operation, maintenance, or modification.

Safety Instructions for AMS 2 Pro

- **To prevent filament feeding jams, do not load flexible materials such as TPU with a hardness of 95A or lower, or moisture-exposed 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.
- Connect the printer and a single AMS 2 Pro using the Bambu Bus cable 6-pin to use the AMS 2 Pro drying function. To dry filaments in multiple AMS 2 Pro units at the same time, purchase official Bambu Lab power adapters to supply power to the other AMS 2 Pro units.
- During filament drying, the AMS 2 Pro expels moisture through external air circulation. Ensure the air intake and air vent are not blocked by other objects for the best drying performance.
- For other AMS models, see the included quick start guides for usage instructions.

Chapter 1 Introduction

The P2S is an intelligent 3D printer based on Fused Deposition Modeling (FDM) technology. It features comprehensive hardware and software upgrades that greatly improve printing stability and ease of use.

- **Smooth user experience:** The 5-inch touchscreen and second-generation UI provide smoother interaction, while the quick-release hotend design makes routine maintenance and replacement easier.
- **Stable support for engineering materials:** With a nozzle temperature of up to 300°C, a heatbed temperature of up to 110°C, equipped with an adaptive airflow system, the passive chamber temperature can reach up to 50°C. The printer reliably supports a wide range of printing needs, from basic to engineering-grade filaments.
- **Intelligent Detection and Comprehensive Calibration:** The printer detects printing anomalies such as spaghetti and clumping to improve overall success rates. It supports multiple calibration functions to ensure stable long-term operation.

When used with the Automatic Material System (AMS), it further enhances printing intelligence and ease of use:

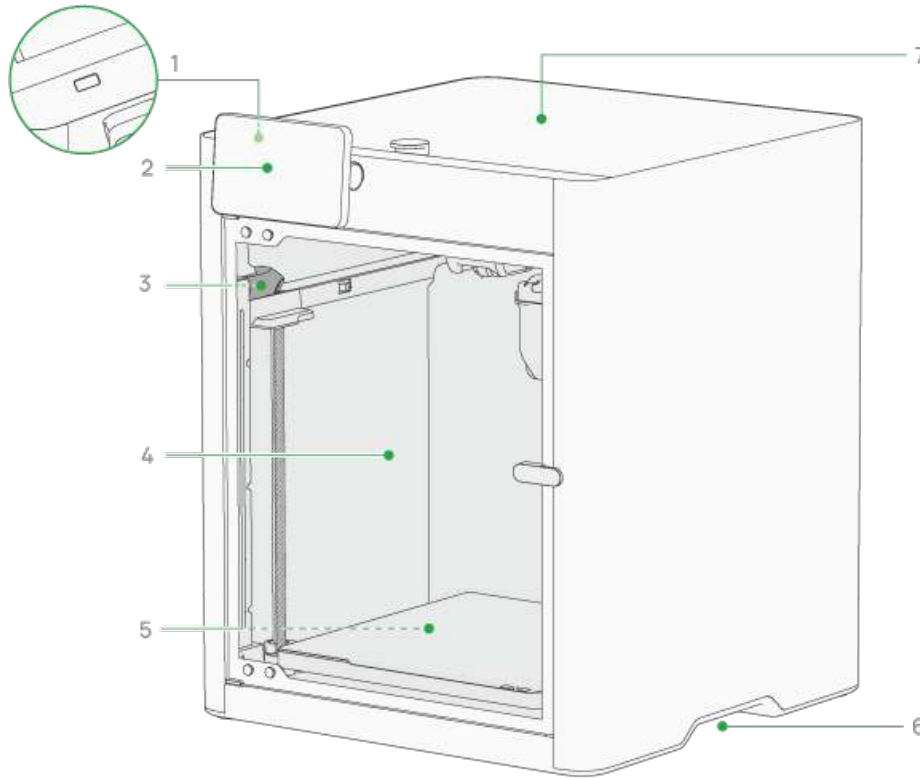
- **AMS 2 Pro:** Supports RFID filament sync, automatic multi-color/multi-material changing, and smart drying. Enhances multi-material printing and filament management for a smarter, more convenient, and improved printing experience.
- **AMS HT:** Specialized for high-temperature filaments, the AMS HT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

NOTE

Unless otherwise specified, **AMS** refers to AMS 2 Pro, AMS HT, and other AMS devices.

1.1 Product Appearance

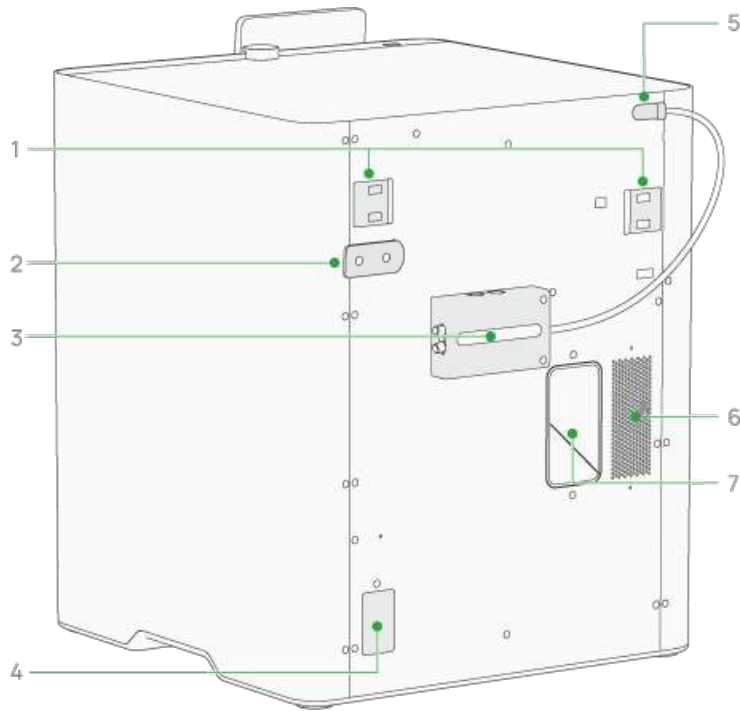
1.1.1 Printer



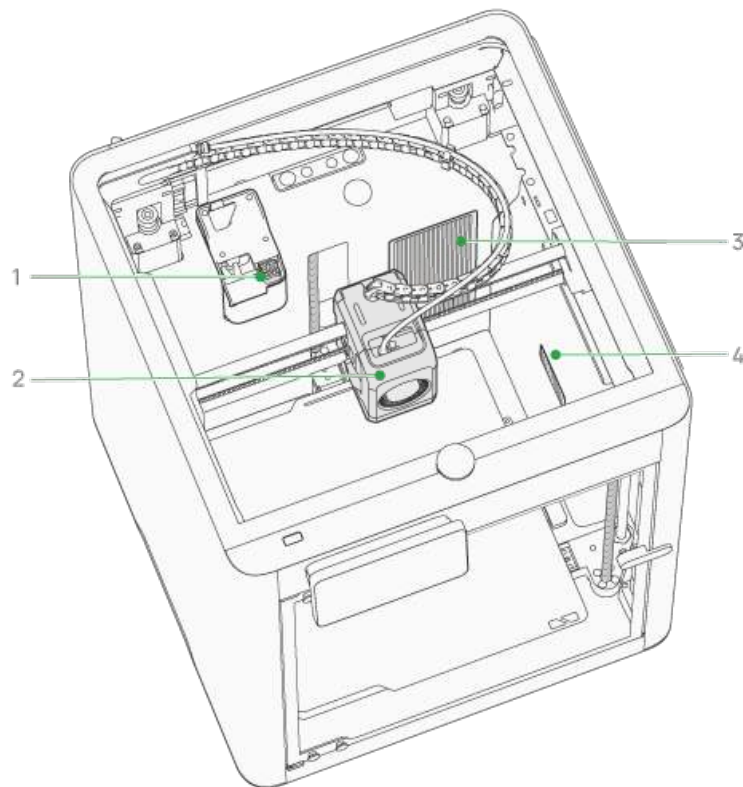
1. **USB port:** Insert a USB drive to print model files offline and store recordings and timelapse files.
2. **Touchscreen:** Displays printing parameters and controls the printer.
3. **Live view camera*:** Monitors print progress in real time. Also used for timelapse recording and AI vision detection.
4. **Front door:** Keeps the printer enclosed and helps maintain chamber temperature during high-temperature printing. It also allows observation of print progress.
5. **Heatbed:** Heats the print surface to ensure stable adhesion of printed layers to the build plate, preventing warping or detachment.
6. **External air intake (lift handle):** In **cooling mode**, it draws external air to lower chamber temperature. Use the handles to lift the printer safely during moving.
7. **Top glass cover:** Made of tempered glass, allows observation of the print progress and provides good sealing and safety. It can also be used to place the AMS.

i TIPS

- For privacy protection needs, you can visit MakerWorld (makerworld.com) and search for "P2S Privacy Cover" under the Bambu Lab official account. Print and install the live view camera cover.
- After the privacy cover is installed, camera-related functions (such as photo, video, and visual detection) will be unavailable.

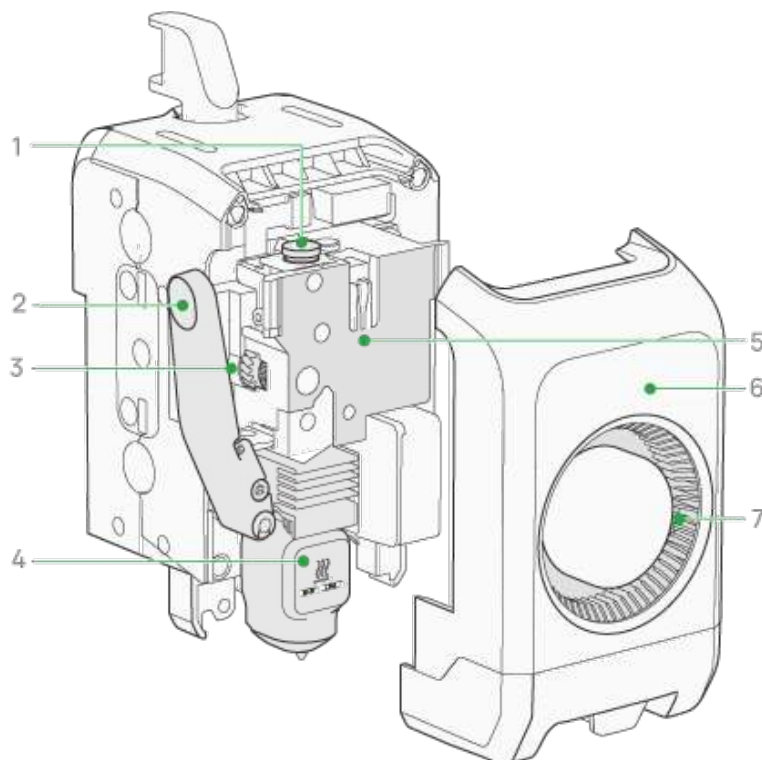


1. **Belt tensioner:** Adjusts and monitors belt tension. It sends data to the system to ensure printing accuracy and motion stability.
2. **Spool holder base:** Used to install the spool holder. The printer's back has 3 reserved installation positions. The pre-installed positions differ between the standalone and Combo versions.
3. **Filament buffer (P2S Combo only):** Reduces tension fluctuations during filament load to ensure smooth feeding.
 - The top includes two Bambu Bus ports 6-pin for data transmission to the AMS and external exhaust fan.
 - The left side includes two filament inlets for connecting to the AMS and external spool to load filament.
4. **Power socket and switch:** Connect the printer to an AC outlet using the provided power cord. Press the switch to turn on the power.
5. **Chamber filament inlet:** Connect the filament buffer or external spool to the printer filament inlet using the PTFE tube to load filament. To replace the tube, first remove the blue clip from the filament inlet, then press the black outer ring to release the tube.
6. **Ventilation holes:** In cooling mode, the printer expels waste air from the chamber through ventilation holes and the purge wiper.
7. **Purge chute:** Connects to the internal purge components, allowing waste filaments to exit the chamber.








1. **Purge wiper:** Consists of purge wiper part and a nozzle wiping pad. The toolhead automatically cleans the nozzle and expels waste through the purge wiper.
2. **Toolhead:** Precisely extrudes and heats filament, depositing melted filament layer by layer on the build plate to form the print.
3. **Air filter:** Air inside the chamber passes through an activated carbon filter before being vented, helping reduce particle and odor emissions.
4. **Adaptive airflow switching unit:** In **cooling mode**, it draws external cool air into the printer to lower chamber temperature and prevent hotend clogging caused by excessive heat.

1.1.2 Toolhead

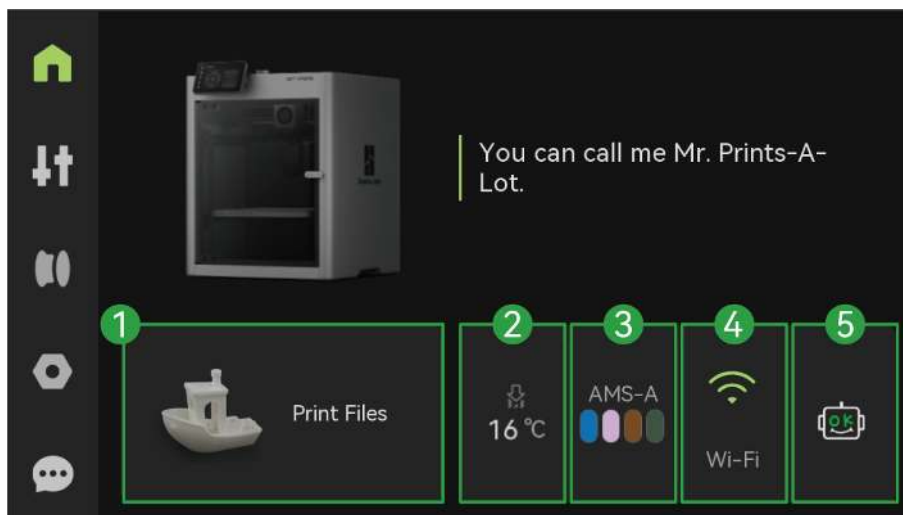


1. **Toolhead filament inlet:** filament is fed into the extruder through the filament inlet.
2. **Filament cutter lever:** during filament unloading or changing, the filament cutter lever is driven to press the filament cutter stopper to cut the filament.
3. **Extruder gear assembly:** uses hardened steel extruder gears, consisting of the drive gear and idler gear. It precisely pushes the filament driven by the extruder motor.
4. **Hotend:** heats filament. The melted filament is extruded through the nozzle and deposited onto the build plate for forming.
5. **Extruder:** grips and pushes filament into the hotend. It precisely controls filament length during extrusion and retraction to ensure print accuracy and appearance quality.
6. **Toolhead front cover:** protects the front part of the printer toolhead.
7. **Part cooling fan:** installed in the center of the toolhead front cover. It blows cooling air near the hotend nozzle to ensure rapid cooling of the printed parts.

1.2 Screen Icons

The main menu bar is on the left side of the printer screen. Tap each icon to navigate to the  **Homepage**,  **Control**,  **Filament**,  **Settings**, or  **Messages** pages to configure and operate the printer accordingly. Please see [Common Screen Operations](#) for detailed function descriptions and operation guides.

1.2.1 Homepage



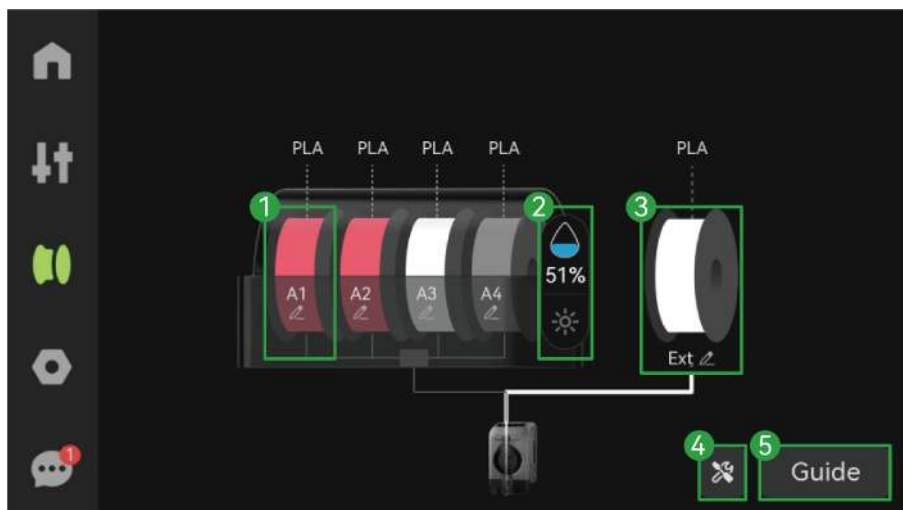
1. View model files on internal and USB storage.
2. Quick access to nozzle and extruder adjustments.
3. Quick access to the AMS filament page.
4. Quick access to network settings.
5. Quick access to the HMS (Health Management System) page.

1.2.2 Control



1. View and manually set the air management mode and fan speed.
2. Set print speed mode.
3. Control the motion of the toolhead and heatbed along the X, Y, and Z axes.
4. Set the nozzle type and temperature. Control the extruder to extrude or retract filament.
5. View chamber temperature.
6. Set heatbed temperature.
7. Control LED light inside the chamber.

1.2.3 Filament

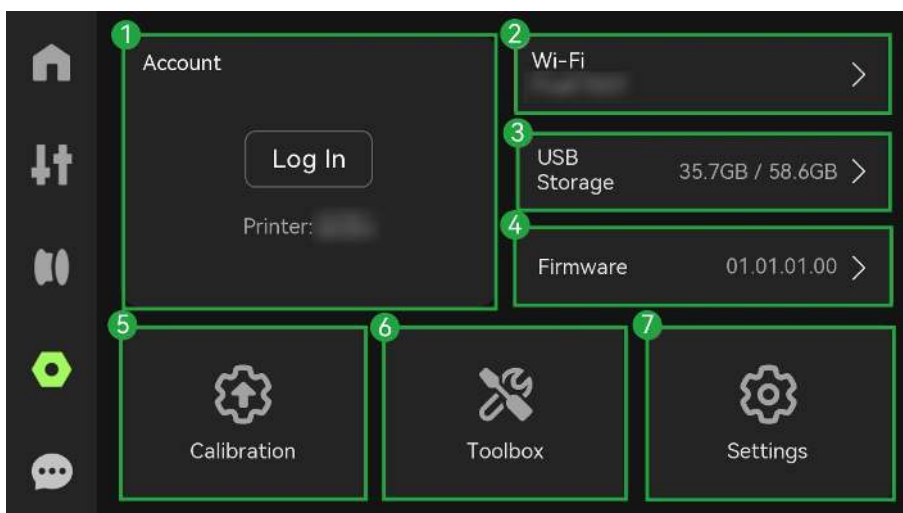


1. Tap any filament spool icon to edit filament, load or unload filament, and re-read filament RFID.
2. View humidity and temperature inside the AMS, and perform drying.
3. Tap the external spool icon to edit filament and load or unload filament.
4. Enable auto refill and AMS drying functions.
5. View filament loading guide.

NOTE

Some features shown require AMS connection to use. The P2S standalone version only displays the external spool icon.

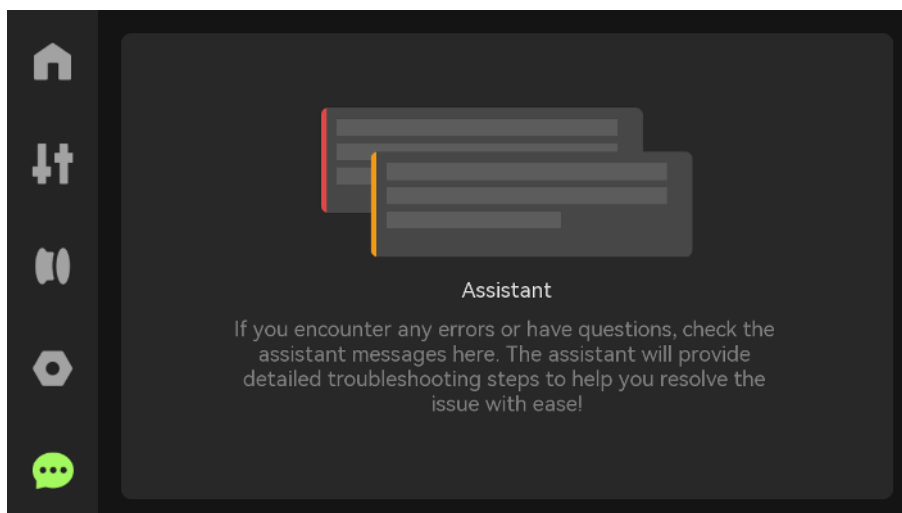
1.2.4 Settings



1. Log in and view account.
2. Configure printer network.
3. Configure USB storage.
4. View and update firmware.
5. Start print calibration.

6. Use functions such as printer maintenance and filament drying.
7. Other basic settings.



1.2.5 Message



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

1.3 HMS Messages

HMS (Health Management System) is a built-in diagnostic and status monitoring feature in Bambu Lab printers and AMS. When the device encounters hardware faults, print failures, or requires cleaning and maintenance, HMS alerts you through the printer screen or software notifications (such as Bambu Handy and Bambu Studio) and provides corresponding solutions.

You can click the icons  at the bottom right of the homepage or the left menu  to access the message page and view detailed information. Use any of the following methods to diagnose and resolve issues:

- 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 on the HMS homepage to view detailed causes and troubleshooting steps for the fault.

TIPS

The printer can assess its cleanliness based on task type and print duration, and provide maintenance reminders. After receiving the printer, update the firmware to the latest version to enable this feature.

Chapter 2 First Print

Before starting the first print, ensure the printer is properly placed, installed and auto-calibrated. This chapter guides you through these preparations to ensure the printer starts its first print in optimal condition.

2.1 Preparation Before Printing

2.1.1 Printer Placement

Keep the workspace clean and tidy to ensure print quality and equipment safety. Place the printer on a level, sturdy platform.

2.1.2 Initial Installation

Please follow the steps below to install the printer:

Step 1. Remove packaging materials

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

Step 2. Check the printer and accessories

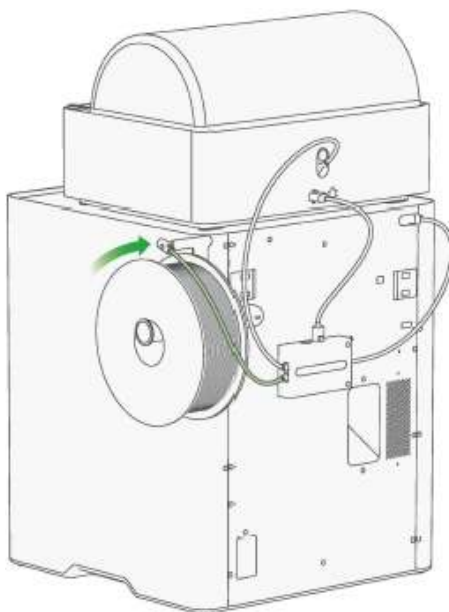
- Ensure that all included accessories are present (accessory box, power cord, PTFE tubes, spool holder, quick start guide, etc.).
- Ensure that the printer's standard **Bambu textured PEI plate** is correctly installed on the heatbed.

CAUTION

- # Do not touch the build plate surface with your hands. Oils can reduce adhesion, causing model collapse or spaghetti issues.
- # If the build plate surface has oil residue or dust, clean it with warm water and detergent. Dry completely before attaching it to the heatbed.

Step 3. Install filament components

Ensure the external spool and AMS 2 Pro (if any) are properly installed. If not, follow the steps below to complete the installation.



Scan the QR code or click the link (wiki.bambulab.com/p2s/manual/unboxing-p2s) to watch the installation video and jump to the relevant section as needed.



i TIPS

Included accessories may vary by product variant. Please refer to the actual items received.

Step 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 touchscreen should light up normally and display the initial setup interface.

2.1.3 Initial Calibration

⚠ CAUTION

Calibration and print jobs should be performed only after the printer has been properly positioned. Moving the device after calibration may affect measurement accuracy.

On first startup, the printer will automatically enter the initialization process. Follow the on-screen instructions to complete the steps:

Step 1. **Language, region, and network settings**

Select the appropriate language and time zone, then choose an available Wi-Fi network, with the option to skip network selection if it is not required.

Step 2. Account login

Open Bambu Handy and scan the QR code on the screen to bind the printer (see [Install Bambu Handy](#)). Skip binding if not needed.

Step 3. Automatic calibration

The printer will automatically perform a series of checks and calibrations, including motor noise cancellation, vibration compensation, and auto bed leveling.

Step 4. Remove the foam under the heatbed

The heatbed rises after the calibration. Remove the remaining foam from the bottom area.

Step 5. Calibration completes

When the touchscreen displays "Calibration completed," you can start your first print.

2.2 Load filament

2.2.1 From AMS



NOTE

This method applies to the P2S AMS Combo. If using other AMS devices (such as AMS or AMS HT), the operation is the same as for AMS 2 Pro.

Step 1. Place the spool into any AMS 2 Pro slot. Gently push the feeder tab and insert the filament. The AMS 2 Pro will detect the filament and load it automatically.



Step 2. After confirming the spool rotates smoothly, close the AMS 2 Pro cover and secure the locking tab.


Step 3. On the touchscreen, tap , select the loaded filament slot , then tap **Load**.

Step 4. The AMS automatically recognizes Bambu filament. For third-party filament, select the corresponding filament icon, then tap **Edit** to set filament details.

2.2.2 From the External Spool

NOTE

This method applies to the P2S standalone version. If you want to use AMS loading feature simultaneously, please visit the Bambu Lab Wiki (wiki.bambulab.com/home) and search for "P2S First Print" to find the corresponding P2S spool holder installation methods.

- Step 1. Place the spool into the external spool holder with the side matching the filament winding direction.
- Step 2. On the printer screen, tap  > **external spool (Ext)** > **edit**, select the filament type and color, then tap confirm.
- Step 3. Insert the filament into the PTFE tube and push forward until it stops. The line between the toolhead and spool on the printer screen will show the filament color, indicating the extruder has detected filament entry.



- Step 4. On the screen, tap **External Spool (Ext)** > **Load** and wait for the nozzle to heat up.
- Step 5. Follow the on-screen prompts to check if the filament extrudes smoothly from the nozzle. Then select the corresponding option on the touchscreen.

TIPS

- Manually push the filament again during loading to ensure it remains inside the extruder.
- If no filament extrusion is observed, select "Not extruded yet, retry" as prompted on the printer screen, then keep pushing the filament until it extrudes properly.

2.3 Start a Print from Printer Screen

i TIPS

Before printing, it is recommended to connect a USB drive to the printer's USB port. This allows you to import sliced files and start prints directly from the screen, and can also be used to store recordings, print history, and print cache. See [Connect a USB Drive](#) for more usage tips.

The printer includes pre-sliced files for common models such as the scraper handle, ready to print from the screen. You can also start prints from a mobile device using Bambu Handy or from a computer using Bambu Studio. See [Printing from Multiple Devices](#).

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

i TIPS

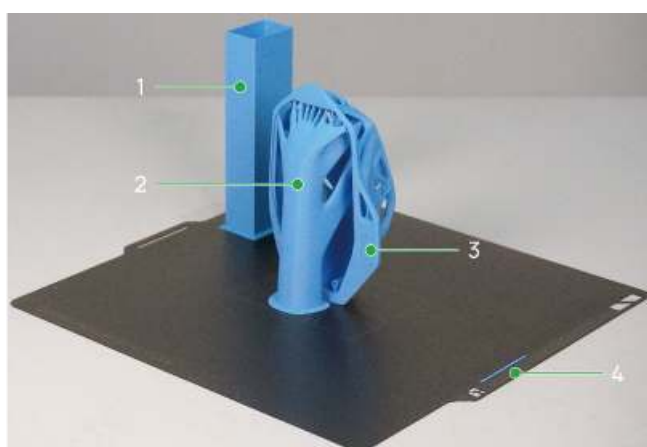
In advanced options, you can manually set bed leveling and flow calibration modes. We recommend using the default automatic mode.

Step 2. Tap the required filament on the AMS or external spool icon, then tap **Print**.

2.4 Remove the model

Step 1. **Wait for the heatbed and model to cool**

After printing is complete, wait for the heatbed to cool to room temperature before removing the build plate, model, and other print structures. This prevents model deformation, build plate damage, or hand burns.



1. Prime tower
2. Supports
3. Model body
4. Prime line

i TIPS

On the printer screen, go to the **🔧** page to view real-time temperatures of the nozzle, chamber, and heatbed.

Step 2. **Remove the model**

Pinch the edge of the build plate and lift up, then take out the detached model. If the model adheres firmly to the build plate, gently flex it to release the model or use a scraper to assist removal.



Step 3. **Clean the build plate**

Carefully remove the prime line with a scraper, then reinstall the build plate onto the heatbed for the next print. Ensure the build plate aligns properly with the heatbed stoppers when replacing it.



Step 4. **Model post-processing (if necessary)**

For models with support structures, remove the supports promptly after printing. Use your hands or needle-nose pliers to break them off from the model body.


NOTE

For appearance finishing of the prints, visit the Bambu Lab Wiki (wiki.bambulab.com/en/home) and search for "work after printing" to find related tutorials.

2.5 Unload Filament

2.5.1 From AMS


After the print job completes, the filament will automatically retract into the AMS 2 Pro.

If you need to unload filament after stopping or canceling a print job, go to  on the printer screen, select the corresponding filament icon, and tap **Unload** to return the filament to AMS 2 Pro.

NOTE

This method applies to the P2S AMS Combo. If using other AMS devices (such as AMS or AMS HT), the operation is the same as for AMS 2 Pro.

2.5.2 From the External Spool

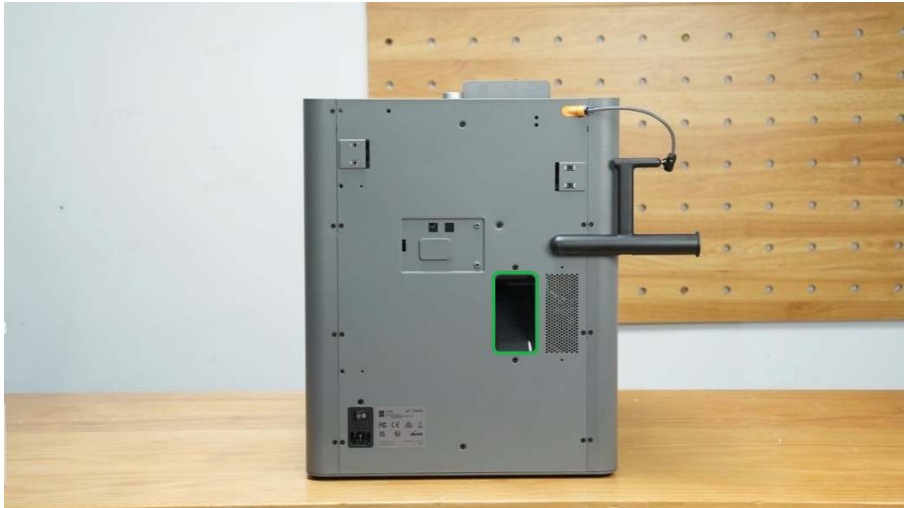
- Step 1. On the printer screen, go to the  page, select the external spool, and tap **Unload**.
- Step 2. When the filament exits the extruder, follow the printer prompts to rewind the filament, then insert the filament end into the spool's hole to secure it.
- Step 3. On the screen, tap **Resume (problem solved)** to complete unloading.

TIPS

Before switching from the external spool to AMS printing, ensure the filament is unloaded and the tip of the filament is removed from the filament buffer (or stored in the spool hole) to prevent loading errors.

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 expelled through the purge chute at 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 and clear it regularly.
- Before printing, make sure there is no waste left inside the purge chute and the outlet is unobstructed.
- 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, featuring a large collection of models uploaded by talented creators across a wide range of categories, from artistic designs to practical tools. Whether you are an artist seeking inspiration or an engineer solving real-world problems, you can find models that meet your needs here. It is also a place to exchange experience, share ideas, and learn from one another.

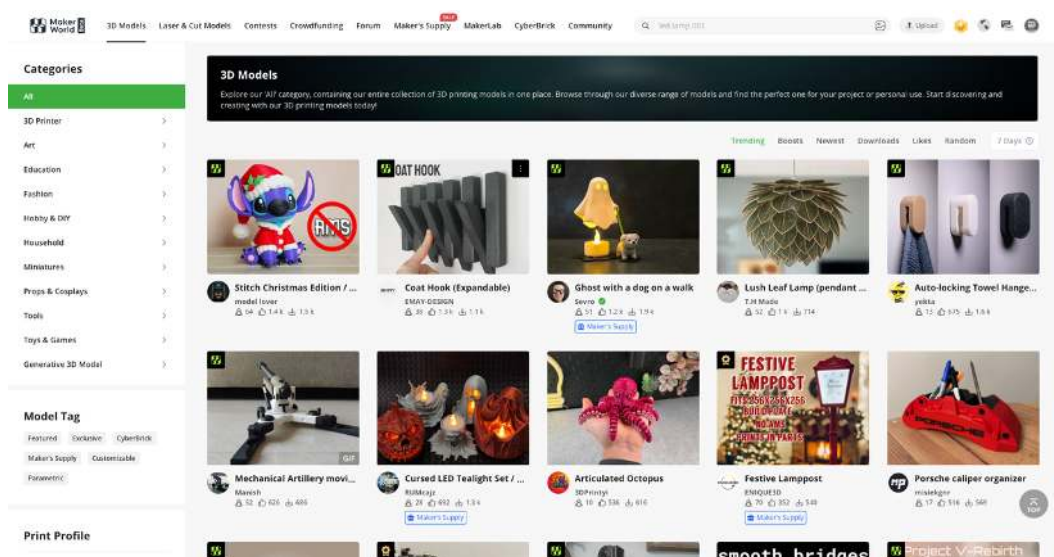
Through MakerWorld, you can:

- 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: Contain only the model's geometry and do not include print settings or color information, so parameters must be adjusted in the slicer or printer software.



3.2 MakerLab Creative Tools

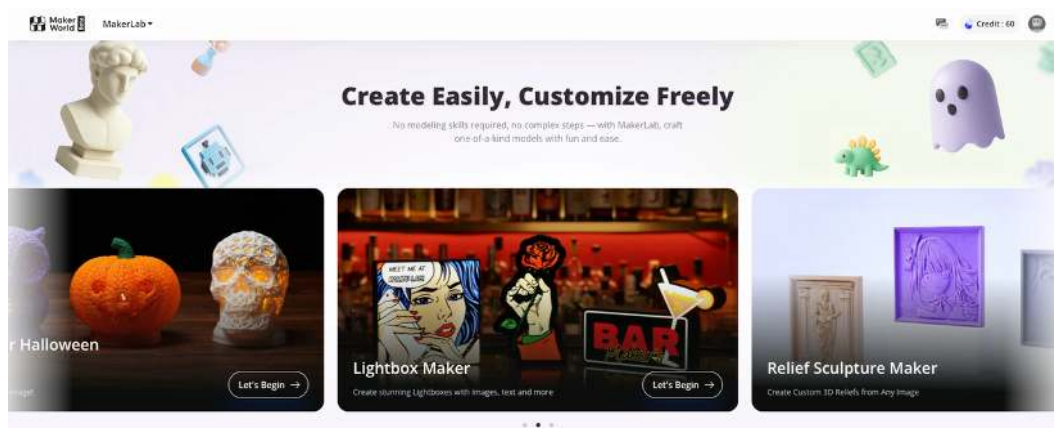
MakerLab offers a range of online creative tools to help turn your ideas into printable models. No modeling experience is needed to create personalized designs quickly.

Creative tools include generators for lightboxes, flexi toys, relief sculptures, and vases, as well as AI-powered and parametric modeling tools, all accessible via the web.

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

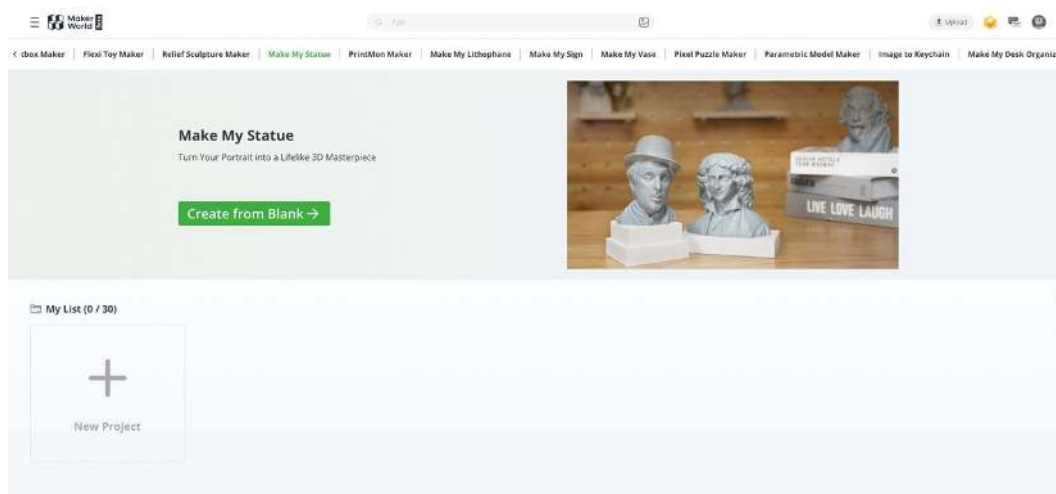
NOTE

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



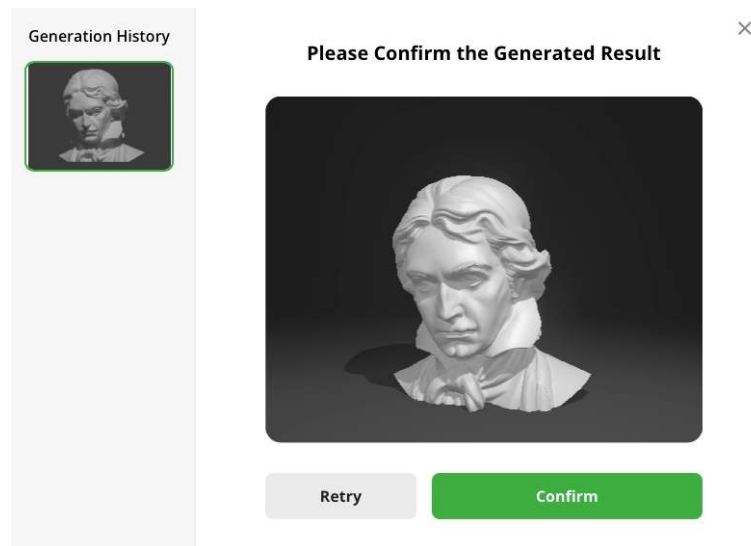
Each creative tool includes detailed instructions. The following uses the Statue Generator as an example to show the workflow on the web:

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 satisfied with the generated model, click **Confirm** to download it. If not, click **Retry** to generate again.



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

Chapter 4 Initiate a Print from Multiple Devices

4.1 Initiate a Print from Bambu Handy

Bambu Handy is an all-in-one mobile app designed for Bambu Lab 3D printers. It lets you search for and print models with one tap, remotely monitor and manage print jobs, quickly reuse past projects, and adjust settings during printing.

NOTE

The interface is continuously updated to improve the user experience. Please refer to the actual interface.

4.1.1 Install Bambu Handy

Before installing Bambu Handy, please ensure the following:

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



Step 1. Visit [bambulab.download](https://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 homepage.

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

NOTE

The registration method (phone number or email address) varies according to local laws and regulations. Please follow the actual display.

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

Step 5. On the Bambu Handy Devices page, tap **+ Bind Printer** (as shown below), then scan the QR code 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.

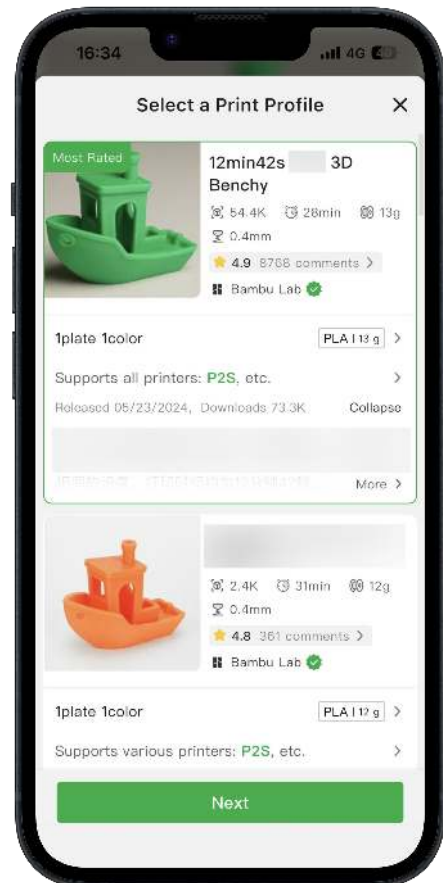
4.1.2 Initiate a Print

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

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

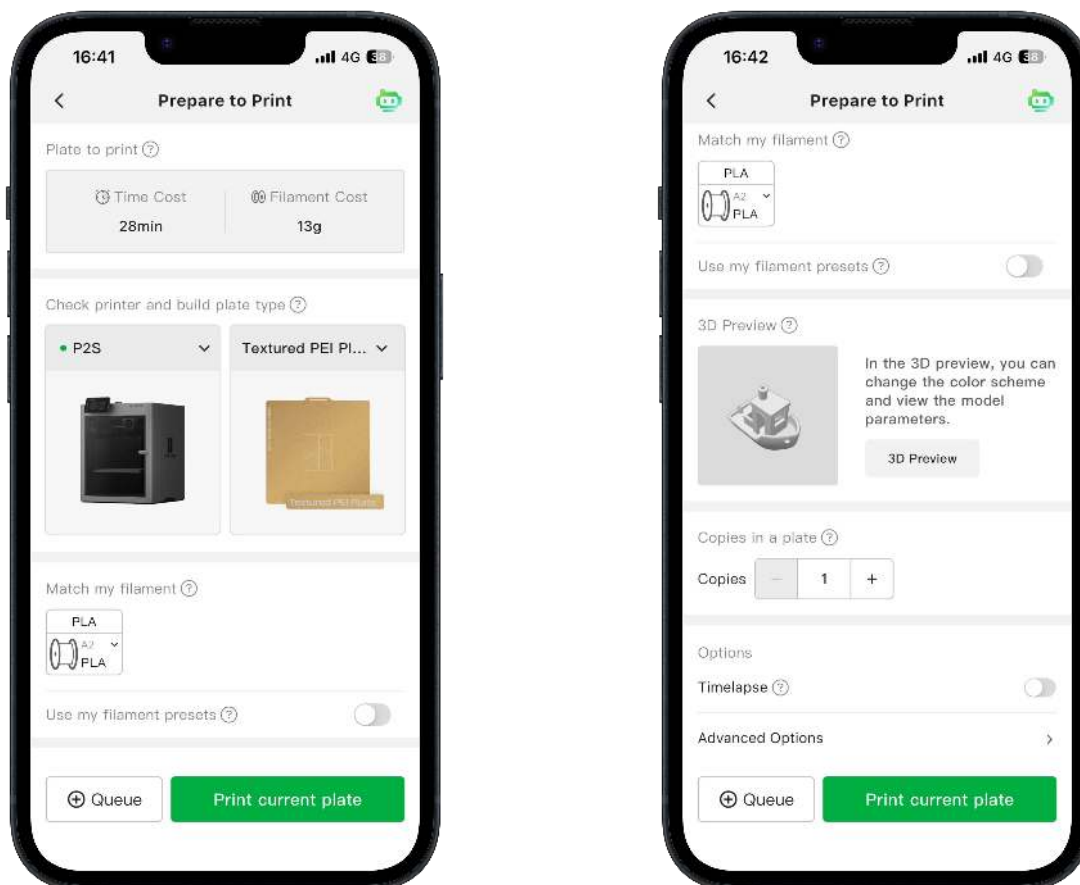


Step 1



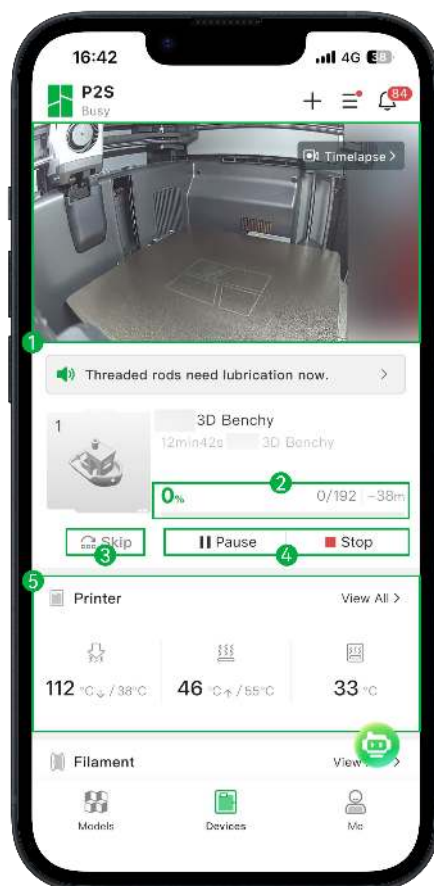
Step 2

Step 3. Check the printer model and the build plate type. Select the required filament, set the number of copies and print options, then tap **Start Print**.

**NOTE**

You can monitor the print job on the Bambu Handy device page or perform other operations (such as adjusting print speed).

4.1.3 Device Page Introduction



1. Camera View: Remotely view live feed of the printer to monitor print status.
2. Progress bar: Displays the current print layer and estimated remaining time to track print progress.
3. Parts skip: Skip the current object and continue printing the remaining objects when one of the objects collapses or fails to print correctly.
4. Task control: Pause or stop the print in real-time.
5. Printer settings: Adjust print speed, heatbed temperature, light, and more (see [printer screen controls](#)).

4.2 Initiate a Print from 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.

NOTE

The interface is continuously updated to improve the user experience. Please refer to the actual interface.

4.2.1 Install and Bind Bambu Studio

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.

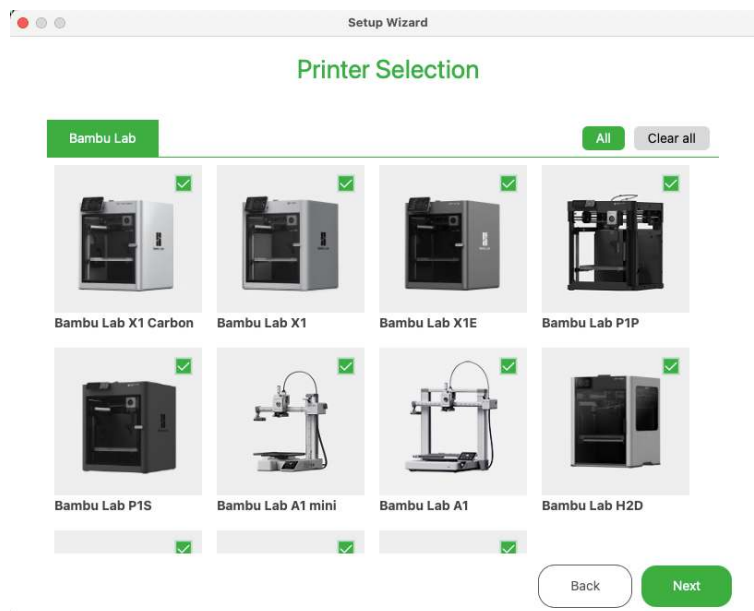
- Windows and MacOS versions: bambulab.com/download
- Linux version: github.com/bambulab/BambuStudio/releases

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**.

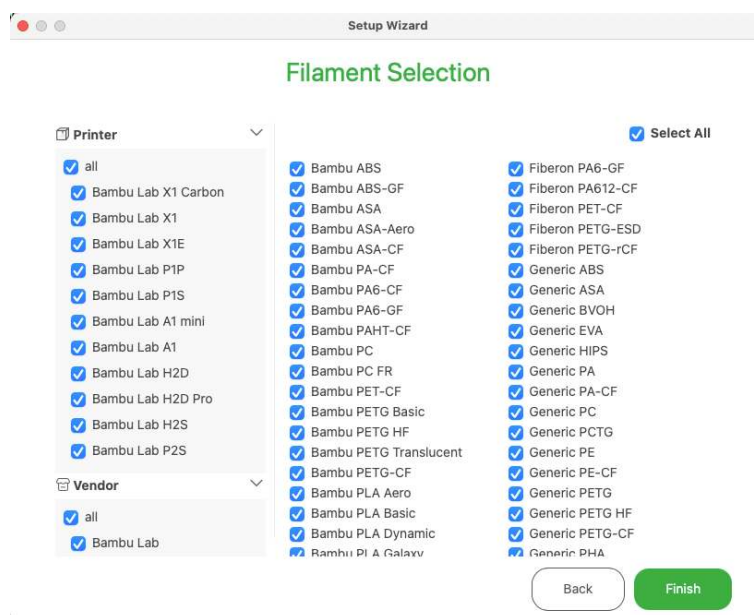
Step 4. Read and choose whether to join the Customer Experience Improvement Program.

Step 5. Select your printer model and nozzle size presets, then click **Next**. The chosen presets will be used for slicing to generate optimal print paths.



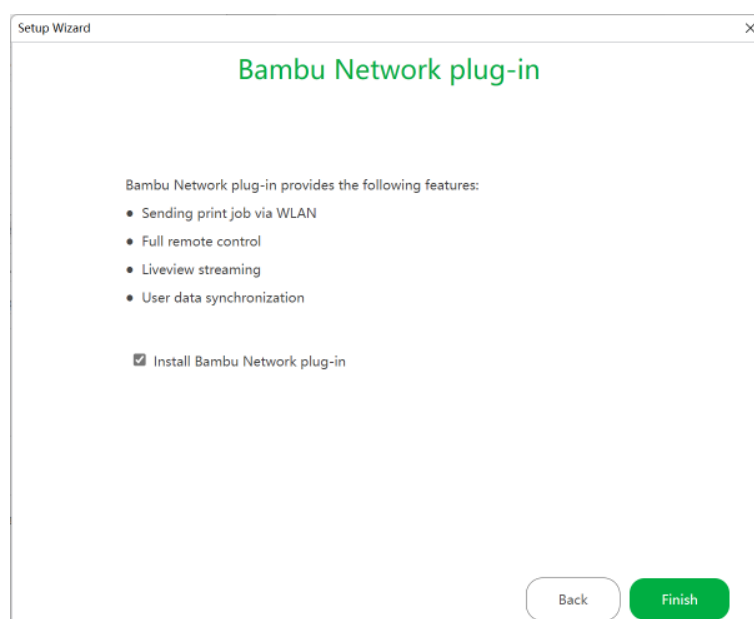
Step 6. Choose the filament preset, then click **Next**.

Filament presets include print temperature and other parameters. Once you select a filament, all printing parameters will be applied during slicing without manual adjustments.



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

This plug-in supports printing via local networks or the internet and provides remote control and user data synchronization features.



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

4.2.2 Download and Import Model

MakerWorld hosts a wide variety of high-quality models uploaded by creators, including model files and complete print profiles. The creators have predefined the print parameters, allowing you to download and send the model to the printer with one click, requiring no complex setup.

MakerWorld Models


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.
2. Choose a print profile that matches your printer model and printing requirements.
3. If you are using MakerWorld on a browser, click **Open in Bambu Studio**; if you are using Bambu Studio, click **Download and open**.
4. Bambu Studio will automatically download the model and open it in the Prepare interface.

Other Models

NOTE

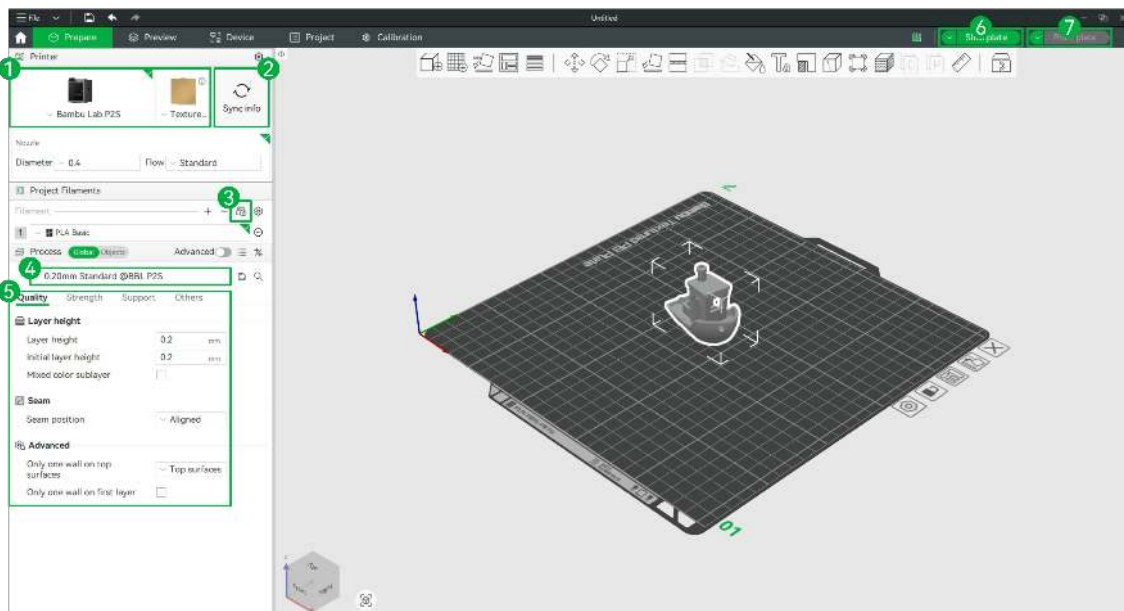
Bambu Studio supports the following file formats: .3mf, .stl, .stp, .step, .amf, and .obj.

Step 1. Import the model using your preferred method.

- In the top left corner, select **File > Import > Import File** (3MF/STL/STEP/SVG/OBJ/AMF..).
- On the Prepare page, click  in the top toolbar, select the model file, then click Open.
- Drag and drop the model file directly onto the plate in the Prepare page.

Step 2. After successful import, the model will automatically load onto the plate, ready for previewing, editing, or slicing.

4.2.3 Initiate a Print



Step 1. Select the correct printer model and build plate type.

Step 2. Click **Sync info** to automatically update the current printer's nozzle and AMS information.

Step 3. Click  in the Project Filament list to sync AMS filament information.

Step 4. Select the desired preset from the Process drop-down list. 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.

 **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. Adjust slicing parameters as needed.
- Step 6. After completing all settings, click **Slice plate** to slice the model and review information such as color schemes, filament usage, and print time.
- Step 7. Send print job. In the pop-up, select the appropriate filament from AMS or the external spool, then click **Send**.

 **NOTE**

You can enable Timelapse, Auto Bed Leveling, and Flow Dynamics Calibration features in the pop-up.

4.3 Initiate a Print from Printer Screen

On the printer touchscreen, tap  > **Print Files** > **Internal (or USB)**, select the model you want to print (see [Initiate a from Printer Screen](#)).

Chapter 5 Adjust Slicing Parameters in Bambu Studio

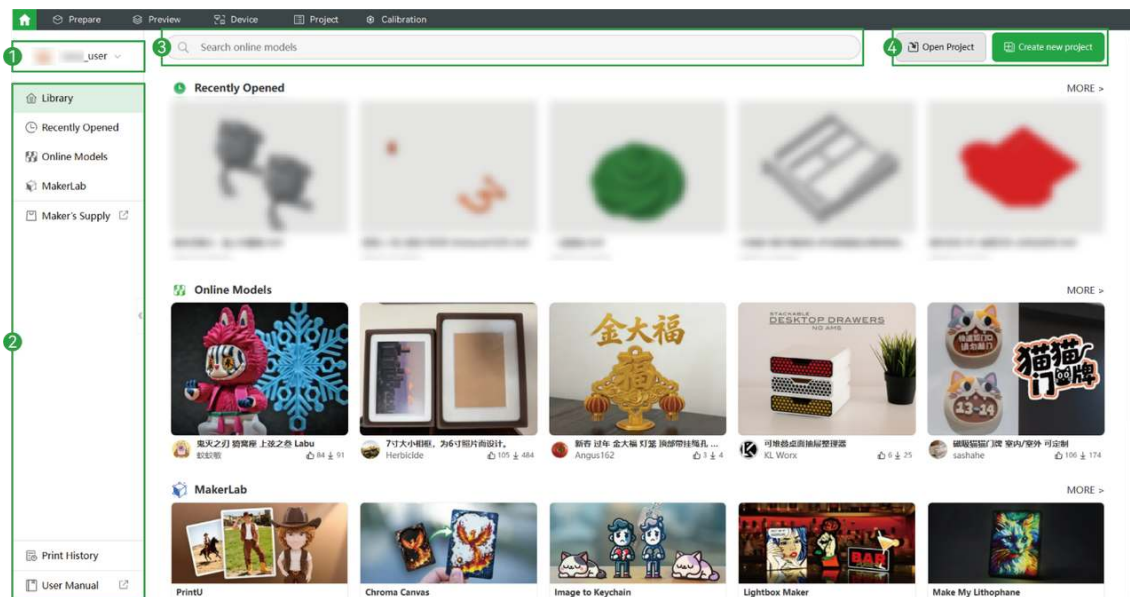
NOTE

- Some screenshots show other printer models for reference only and do not affect the actual workflow.
- The interface is continuously updated to improve the user experience. Please refer to the actual interface.

5.1 Introduction to Bambu Studio Interface

5.1.1 Homepage

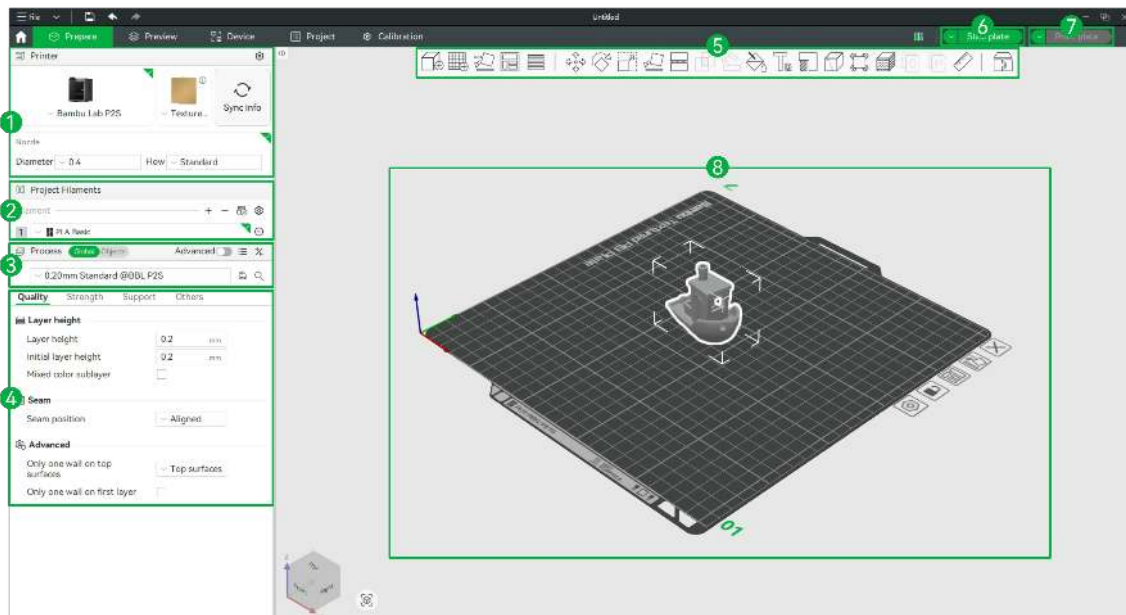
The homepage brings together model management, resource access, and learning guidance. It is the starting point when you open Bambu Studio.



1. **Account information:** View the current account or log out.
2. **Page navigation:** Quickly access recently opened files, online models on MakerWorld, MakerLab, Maker's Supply, print history, and tutorials.
3. **Search bar:** Enter keywords to search online models on MakerWorld.
4. **Project operations:** Open local projects or create new ones.

5.1.2 Prepare Page

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



1. **Printer settings:** Includes the printer model, nozzle type, AMS information, and more. Bambu Studio slices and generates suitable print parameters based on these settings.
2. **Filament settings:** Displays the filament types and colors used in the current file. Click the + icon to manually add or remove filament. The system automatically loads preset parameters for the selected filament type.
3. **Print presets:** Provides multiple quality-level presets (such as Standard, High Quality, etc.) ready to use.
4. **Print parameters:** Models downloaded from MakerWorld usually include all print parameters. Parameters can be manually adjusted to fulfill custom applications, such as lowering layer height to reduce layer lines.
5. **Top toolbar:** Import a model, select any object to perform basic editing and viewing operations using the toolbar.
6. **Slice buttons:** Slice the current plate or all plates and display the slicing results (see [Preview Page](#)).
7. **Print and export:** Initiate a print 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 or 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.

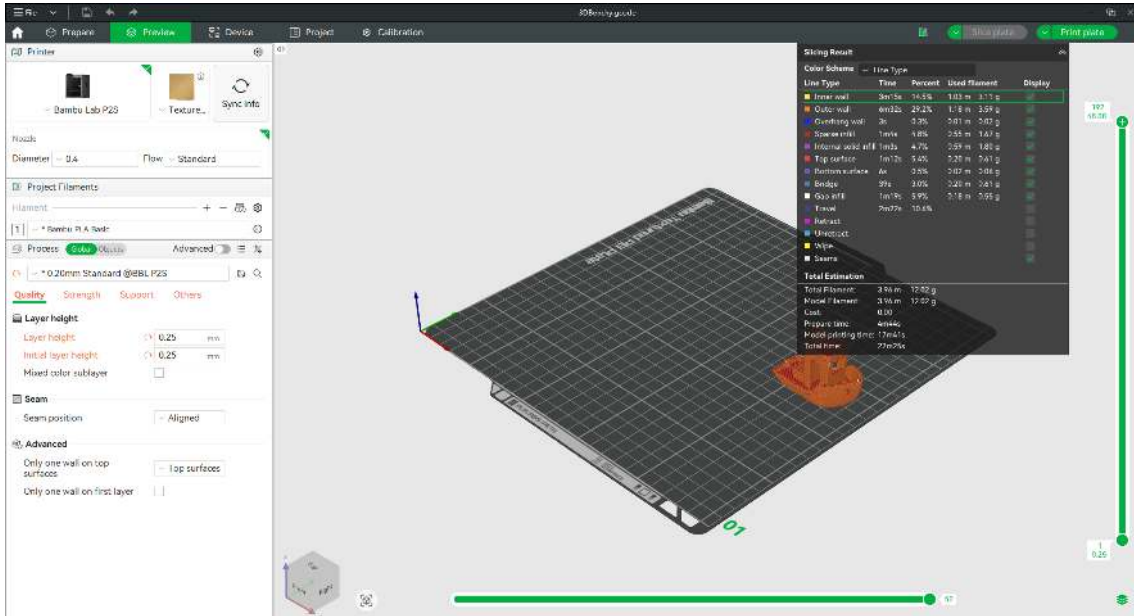
NOTE

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.

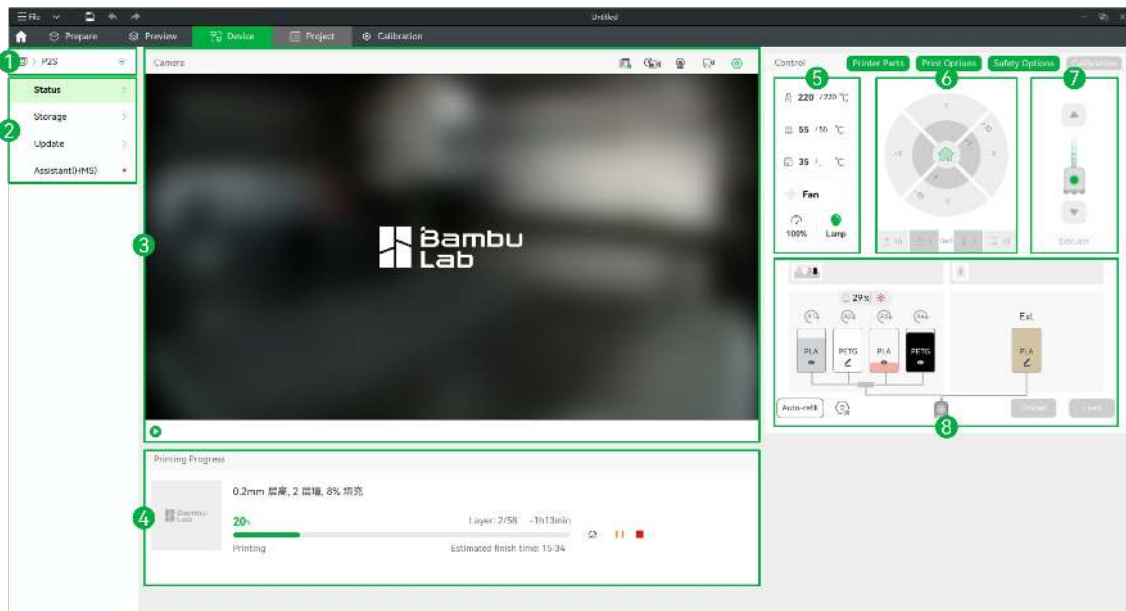
5.1.3 Preview Page

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.



5.1.4 Device Page

Displays the status of the connected printer and its operation options. You can also perform the same operations on the printer screen (see [Common Screen Operations](#))



1. Information about the connected printer. If you have multiple printers, click the printer name to switch between units.
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 or pause live video.

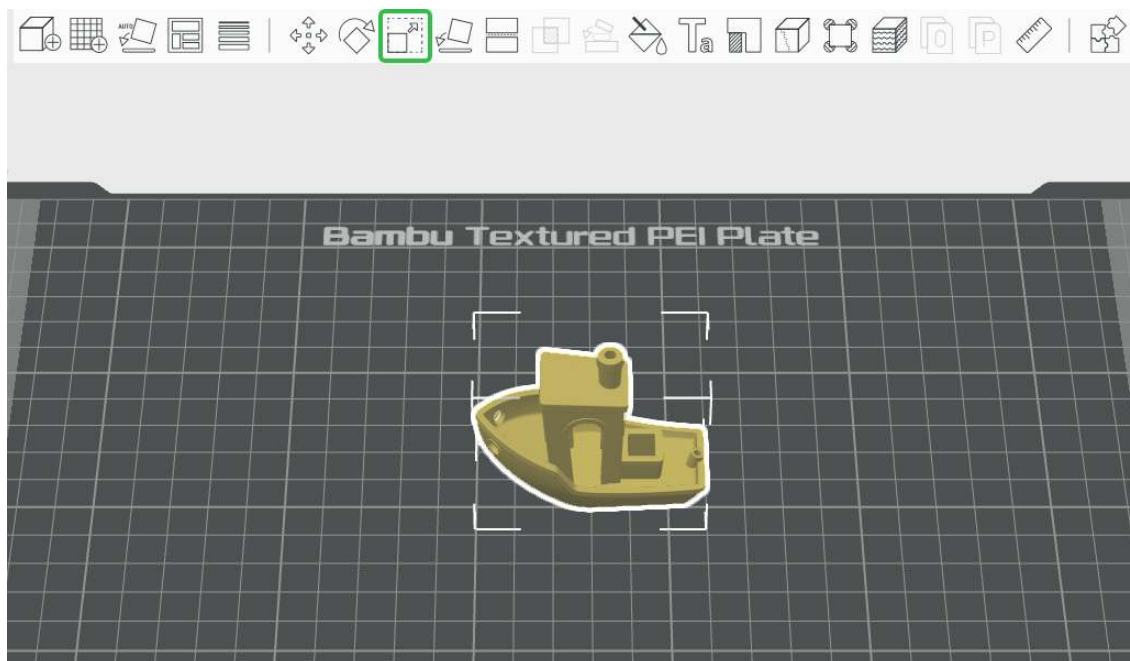
4. Display information about the current print job, including job thumbnail and print progress. You can skip objects, pause, start, or stop the job.
5. Adjust the air management mode, and set temperatures for the nozzle, heatbed, and chamber. The number on the left shows the current temperature, and the number on the right shows the target temperature.
6. Control toolhead and heatbed movement when the printer is idle.
7. Shows the extruder feed status and the color of the loaded filament. Use the up and down arrows to manually extrude or retract 1 cm of filament.
8. Displays and manages filament information, supporting filament editing, loading and unloading operations, and allows control of the Auto Refill feature. If connected to AMS, you can manage features such as insertion update, update on start up and AMS filament backup.

5.2 Model Size

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

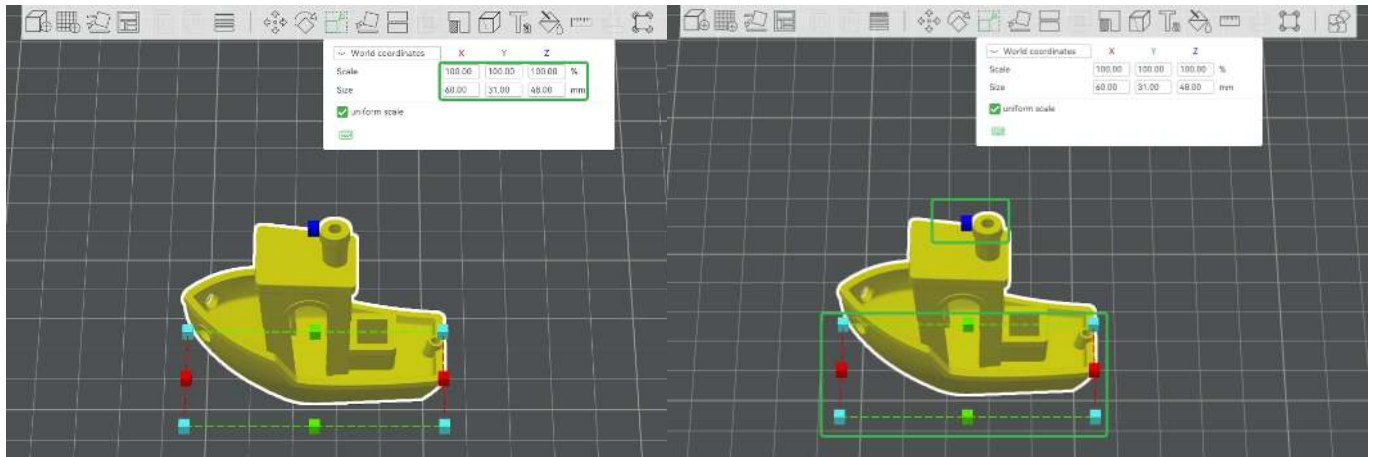
Step 1. Click the model you want to resize to activate the top toolbar.

Step 2. Click the Scale tool.



Step 3. Adjust the model size.

- Enter the 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.

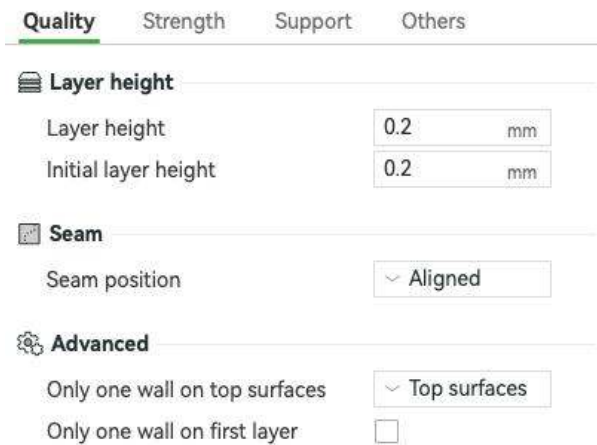


Modify values

Drag control points

5.3 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 (e.g., 0.12 mm) achieves finer layers, resulting in smoother surfaces and richer details, but significantly increases print time. In comparison, a larger layer height (e.g., 0.28 mm) decreases print time but may produce noticeable layer lines affecting surface quality.



Recommended settings:

- Layer height is usually **50%** of the nozzle diameter. The recommended range is **30% – 70% of the nozzle diameter**. For example, when using a **0.4 mm nozzle**, the recommended layer height range is **0.12 mm to 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 (first layer height)

The height of the model's first layer. 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

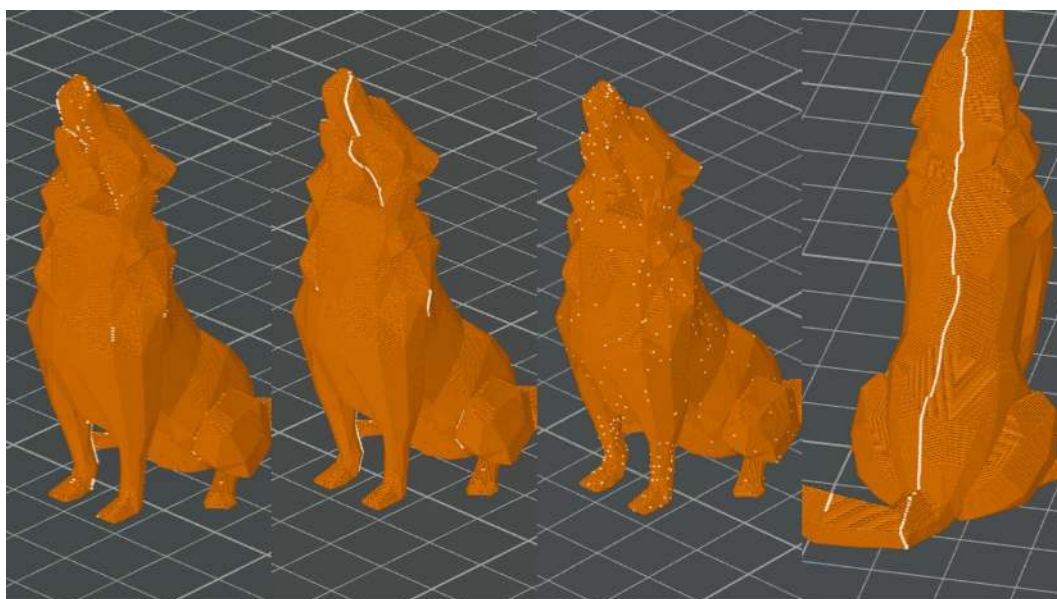
A seam is a small gap formed where the start and end points of wall paths meet on the model surface, usually appearing as a vertical line. This phenomenon is an inherent structural characteristic that cannot be completely avoided in FDM 3D printing.

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.



Nearest

Aligned

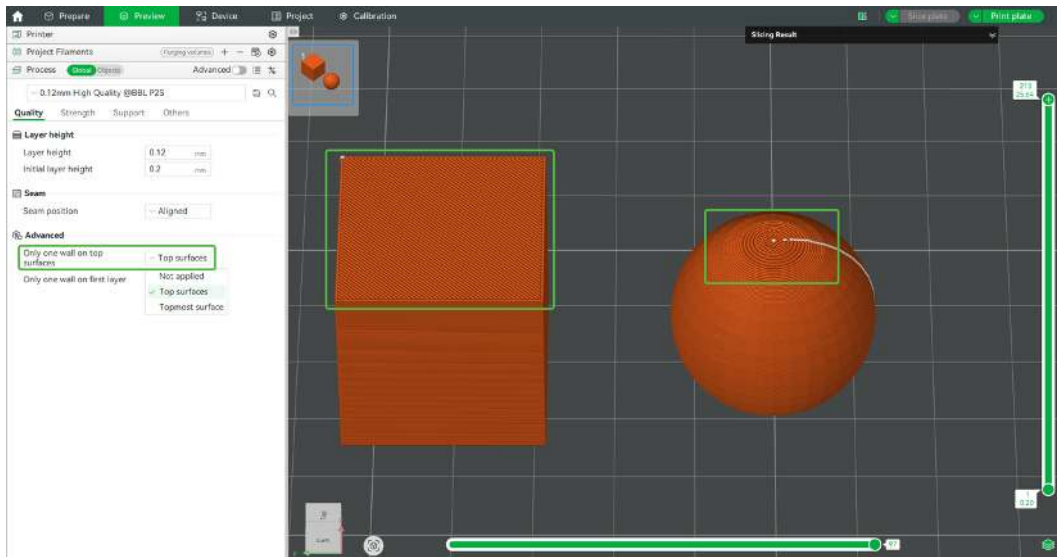
Random

Back

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

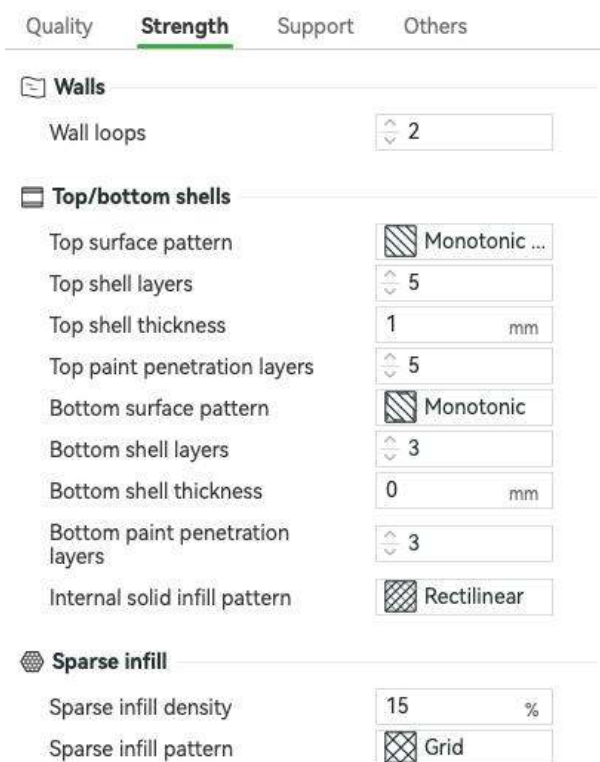
When enabled, the top surface of the model is printed as a single wall, helping achieve clean and neat surfaces. For models with flat top structures (such as cubes), this results in a smooth and neat finish. However, for models with curved top surfaces (such as spheres), it may cause noticeable layer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



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

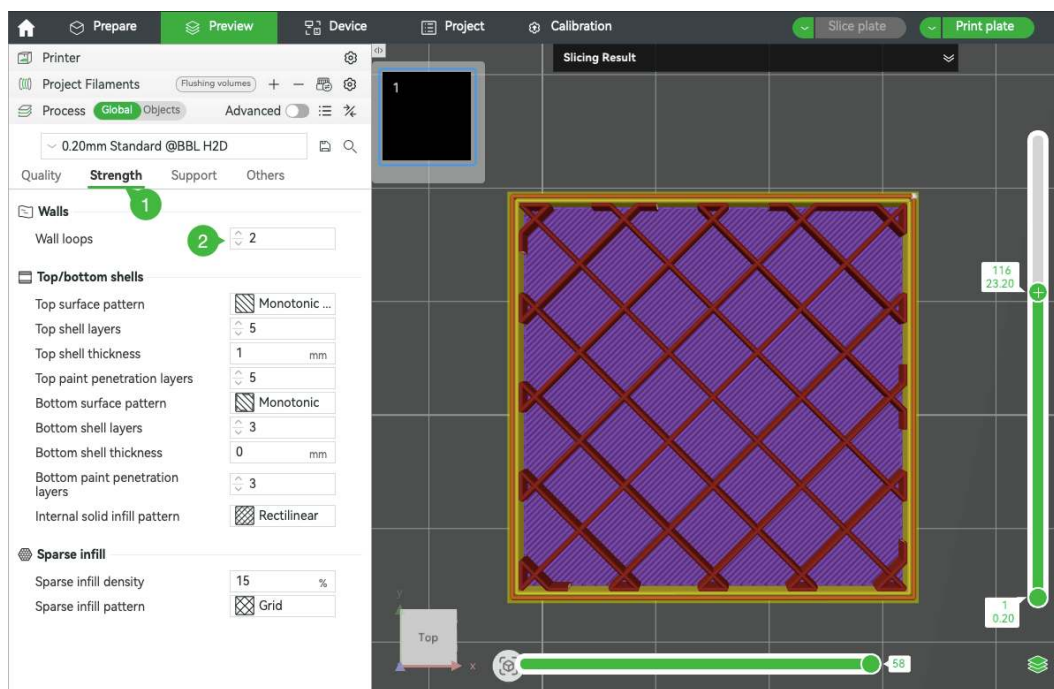
5.4 Strength Settings

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.

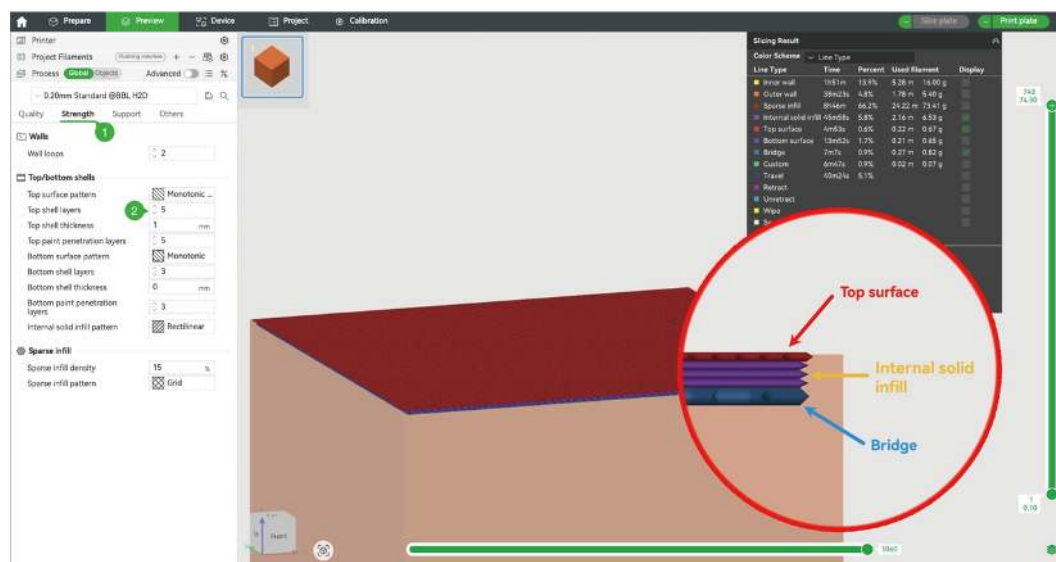
Setting the appropriate top shell thickness helps achieve a smooth, solid top surface and strengthens the model.

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

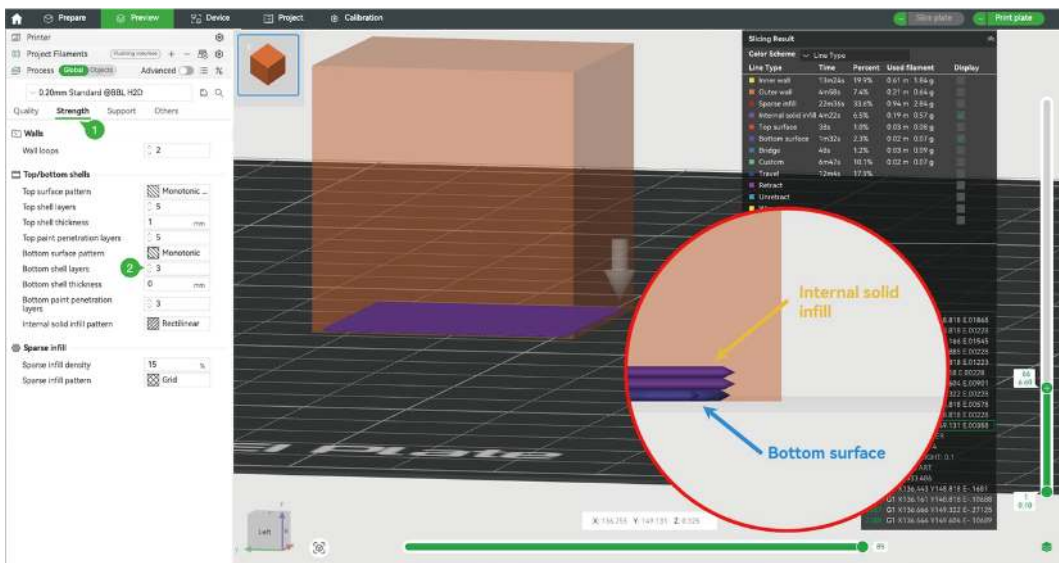
Increasing the bottom shell layers appropriately enhances bottom strength and stability, and improves adhesion to the build plate.

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 thickness is set to **0**, the system will **automatically calculate the thickness** based on bottom shell layers and layer height.

Recommended settings:

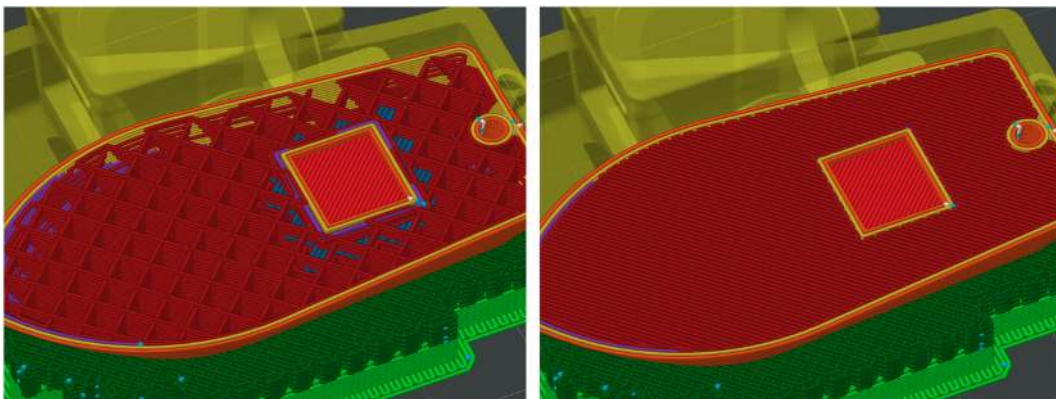
- Setting the bottom shell thickness to 0 means it is fully determined by layer height and bottom shell layers.
- In most cases, 3 or more bottom shell layers are recommended to ensure a solid and reliable base.



Sparse Infill

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

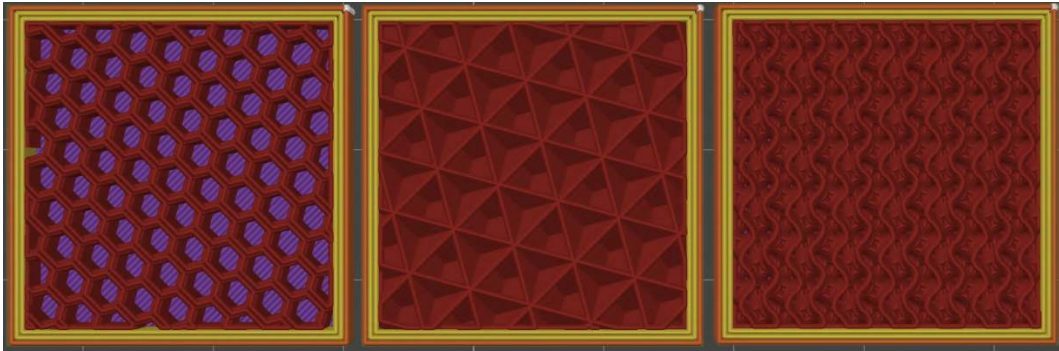
- Low density (10% – 20%): Suitable for non-load-bearing display models, significantly saving filament and print time.
- High density (above 30%): Enhances structural strength, suitable for functional parts.
- Recommended settings: 15%, suitable for most daily prints, balancing strength and efficiency.



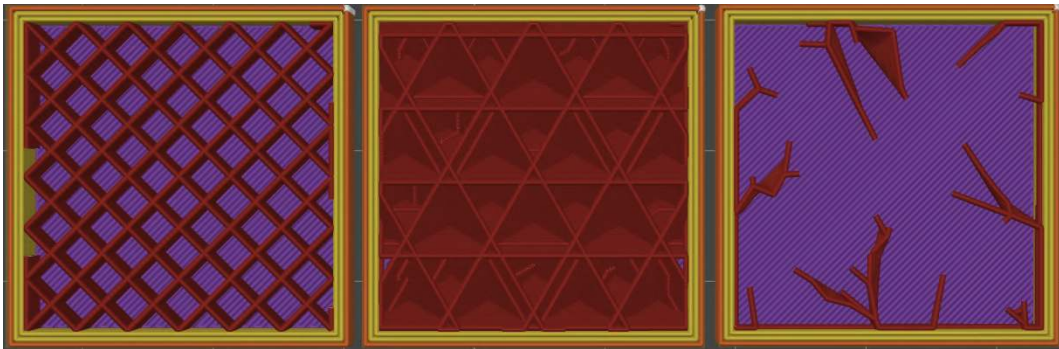
Sparse Infill Patterns

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.



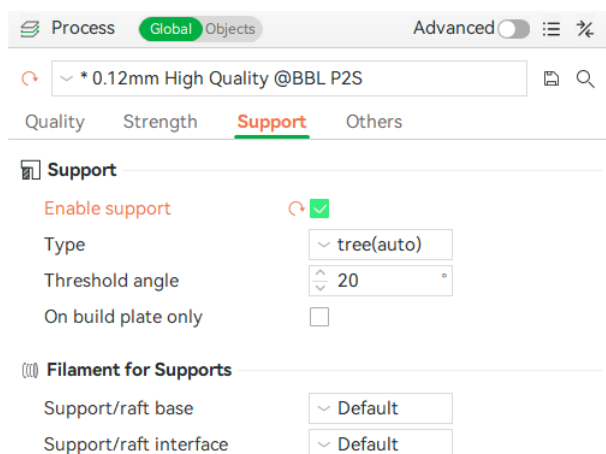
- **Aesthetic-focused patterns:** Such as Hilbert Curve, Octagram Spiral, Archimedean Chords. These patterns offer unique textures or smoother internal structures, improving the overall appearance of models with visible infill.



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 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.

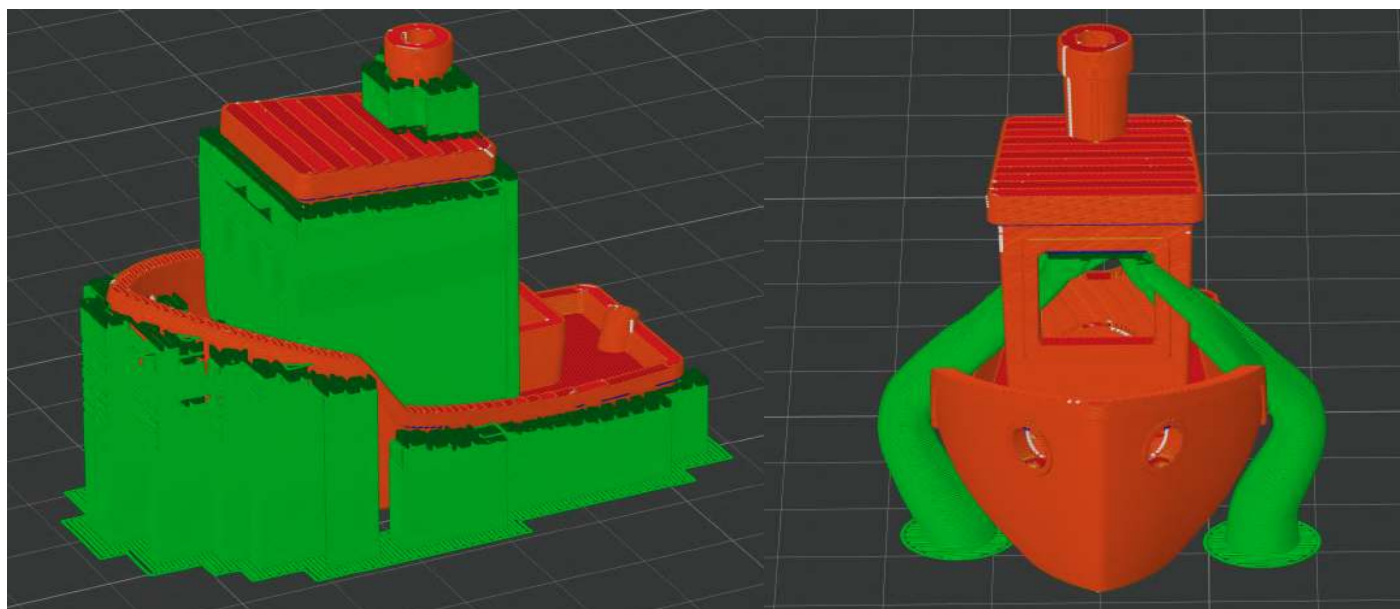


Introduction to Common Parameters

- Type

Normal supports are generated using a standard **linear structure**, providing a stable and evenly distributed layout, fast generation, and easy removal from the model surface. Suitable for mechanical parts with regular structures and geometric models dominated by flat or linear surfaces.

Tree supports employ a “**trunk and 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.



Normal support

Tree support

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

Manual means 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 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

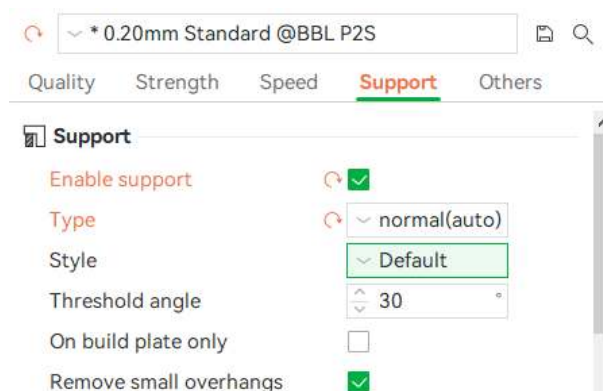
Enabling this option ensures that all supports originate only from the build plate and do not attach to the model itself. This helps reduce support marks on the model surface, improving appearance quality and simplifying post-processing. However, for structures far from the build plate or completely 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.

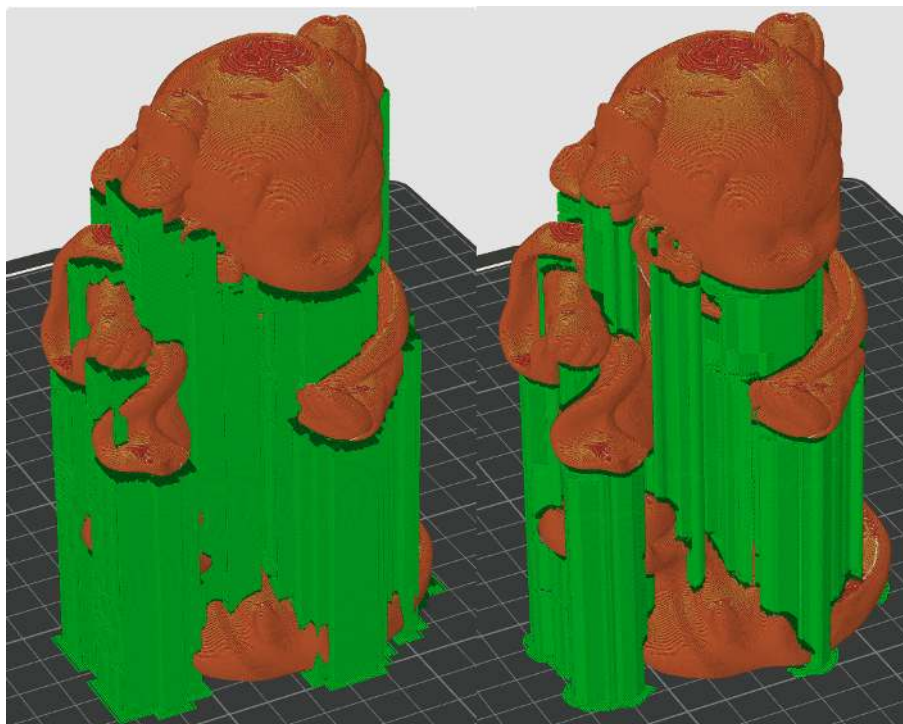
Introduction to Advanced Parameters

- Normal Support Style

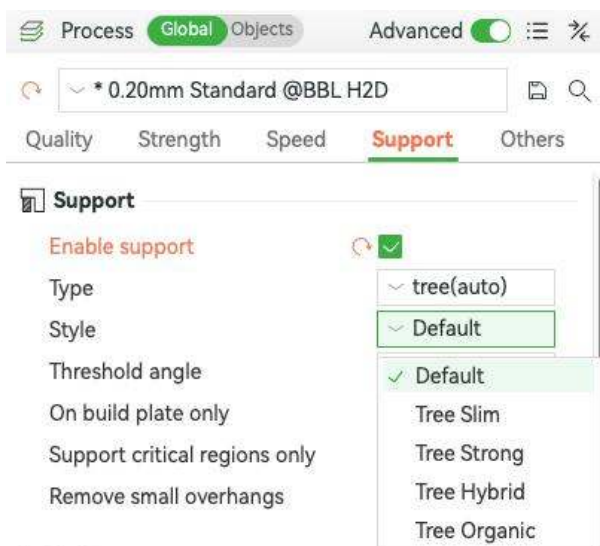


Grid: Projects overhang areas vertically onto the build plate and expand outward to create regular, grid-like support columns. The supports are stable and strong, suitable for **large horizontal overhangs**.

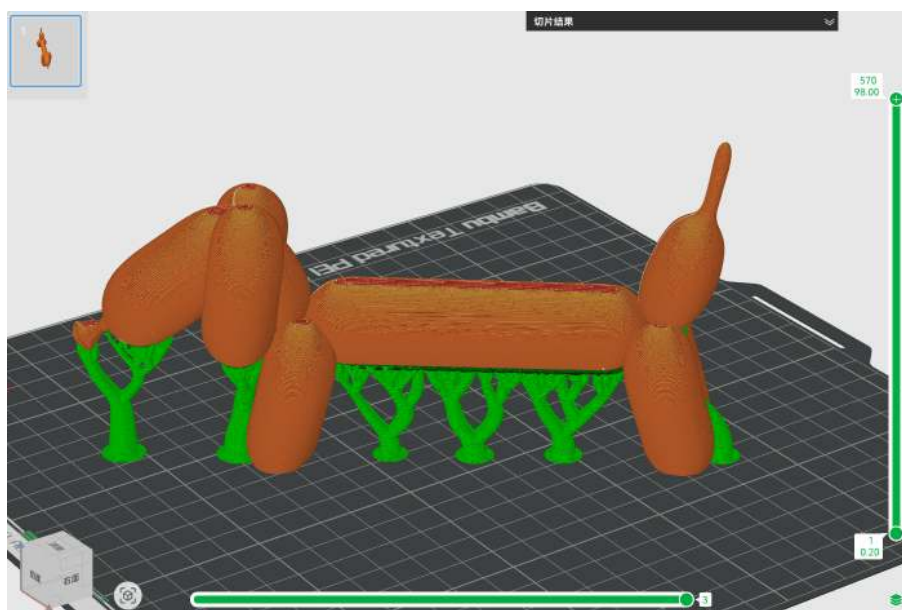
Snug: Grows precisely along the model contour, saving material and providing flexible support shapes. It is well suited for **complex or detailed areas**.



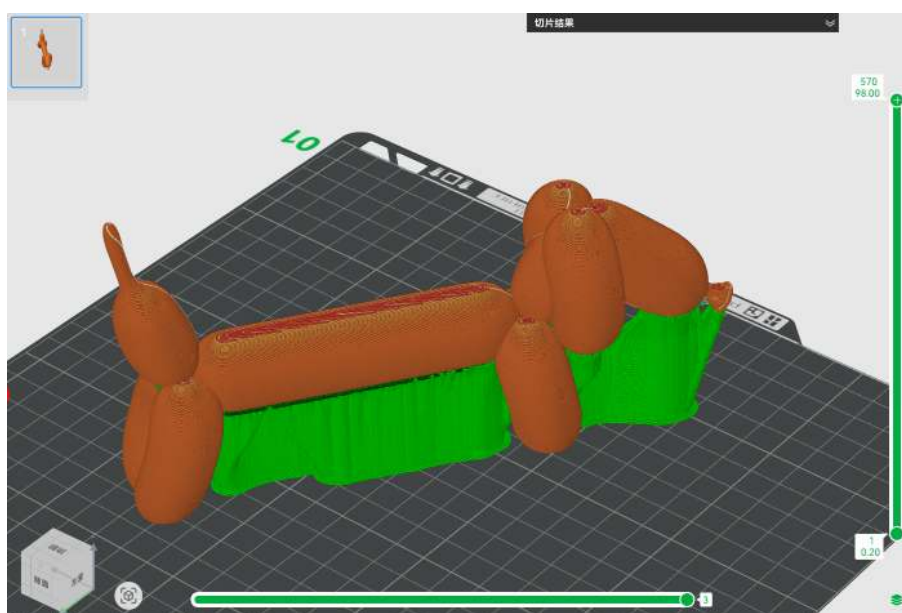
- Tree support style



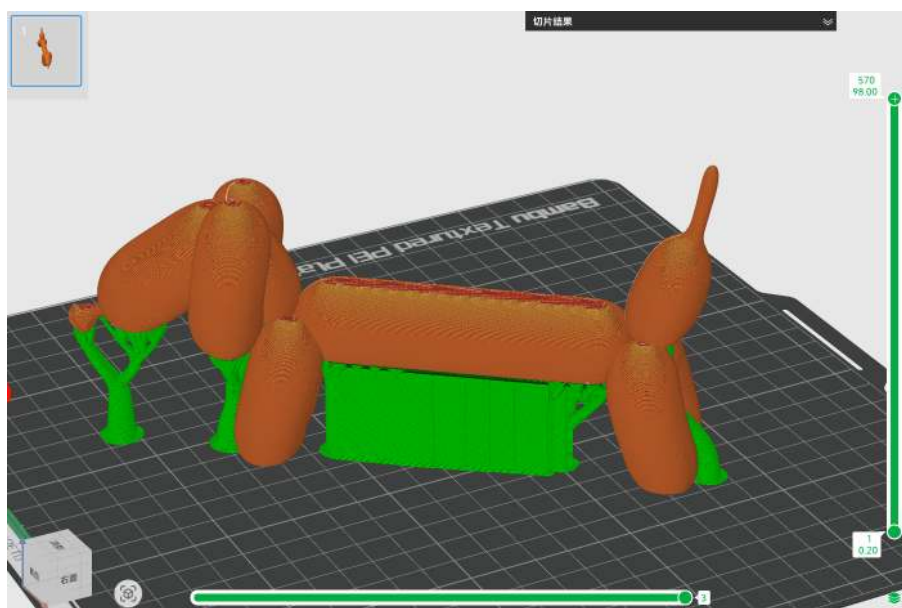
- # **Tree slim:** Features thin branches and an optimized overall structure. Suitable for small, lightweight overhangs. Its aggressive merging strategy reduces material consumption and minimizes removal marks.



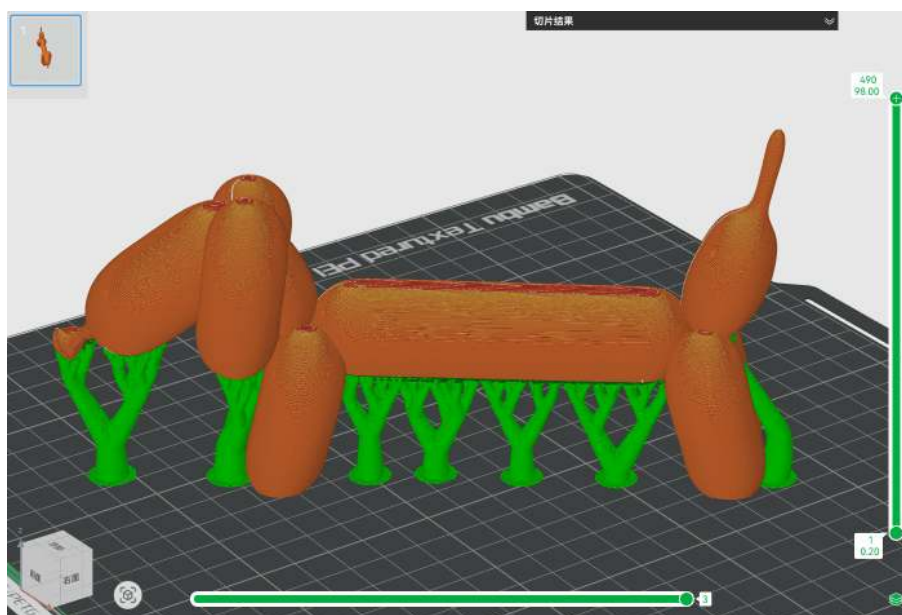
- # **Tree Strong:** Characterized by a thicker trunk and branches, offering a more robust structure with strong load-bearing capacity. Recommended for large or heavy overhangs where stronger support is required.



- # **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.

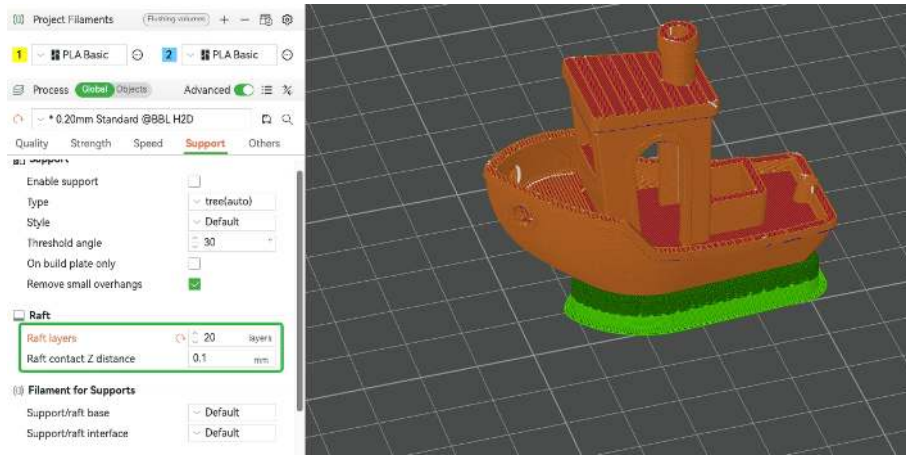


- # **Tree Organic:** Generated using biomimetic algorithms, producing natural, smooth, and curved branches. It intelligently routes support to reach overhangs while minimizing material usage and easing removal. Suitable for most models that require tree supports.



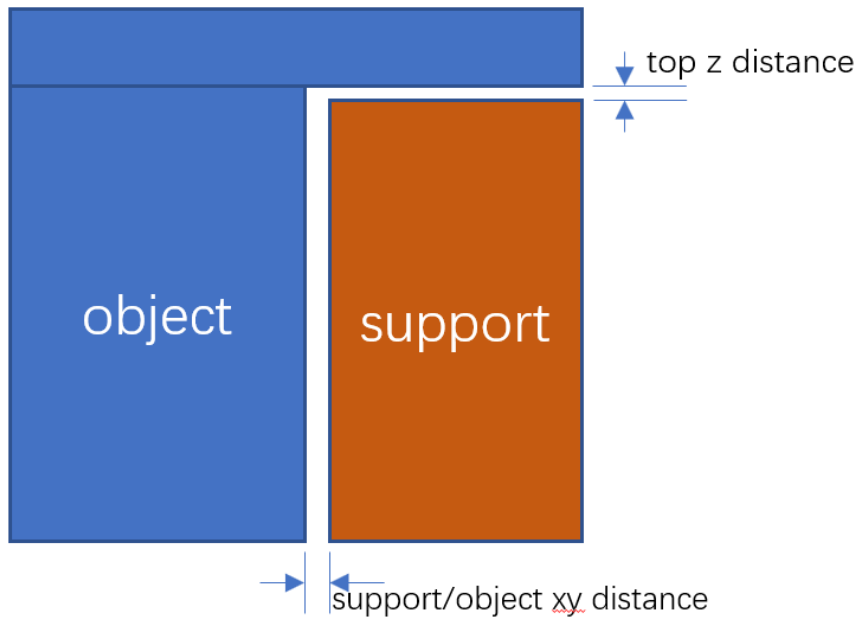
- **Raft**

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.

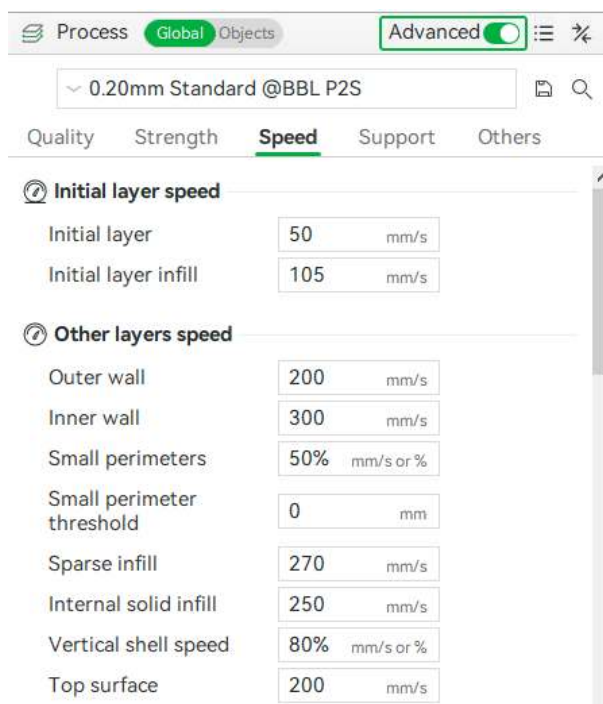
For more advanced support settings, please visit the Bambu Lab Wiki (wiki.bambulab.com/home) for relevant guidance.

5.6 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 (first 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

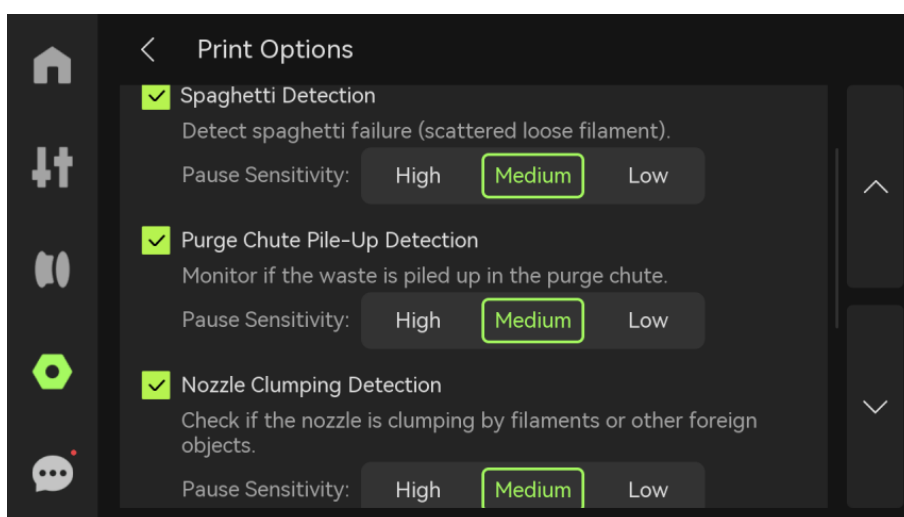
Print acceleration for travel, first layer, inner and outer walls, top surface, and sparse infill. This parameter can affect motion smoothness and speed transition during printing.

Chapter 6 Key Features Introduction

6.1 Intelligent Vision Detection

The printer includes a live view camera that identifies abnormalities during printing and alerts users, significantly improving print success rates, reducing filament waste, and minimizing device damage risks. The detection process operates fully automatically without user intervention, making printing more reliable and effortless.

- On the printer touchscreen, tap  > **Settings** > **Print Options**, then check the boxes to enable the corresponding AI detection features as needed, and adjust detection sensitivity accordingly.
- To configure these features in Bambu Studio or Bambu Handy, go to **Device Page** > **Print Options**.



NOTE

AI detection relies on good lighting conditions. Ensure the printer's chamber light is turned on.

TIPS

Do not turn off the chamber light during printing once this detection is enabled.

Spaghetti Detection

The live view camera captures 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.

CAUTION

Spaghetti detection cannot completely prevent print failures. False alerts may occur, and detection performance may be affected when using black or dark-colored filament.

Purge Chute Pile-Up Detection

The live view camera monitors the purge area to identify any accumulation of waste filament, preventing excessive scraps from causing toolhead collisions or step loss.

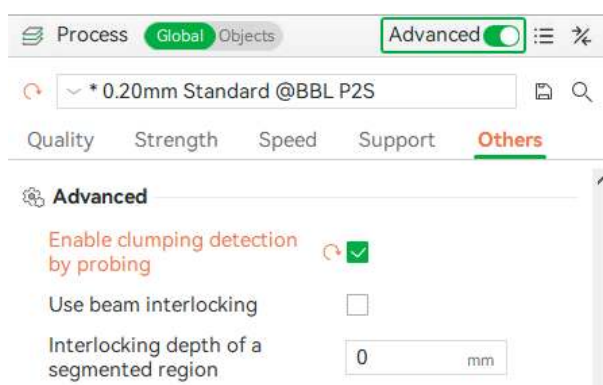
Nozzle Clumping Detection

The live view camera captures images at fixed intervals, analyzed by AI algorithms. Once a clump reaches a certain size, it can usually be detected within 2 minutes. A notification will pop up and the print job will be paused to prevent toolhead damage.

⚠ CAUTION

Clumping detection cannot completely prevent print failures. False alerts may occur and detection performance may be effected when using dark-colored filament or in Ludicrous mode.

When using dark or high-temperature filament, enabling **clumping detection by probing** in Bambu Studio will improve detection accuracy. The toolhead probes down to the heatbed on the 4th, 11th and 20th layers for contact checks. If the nozzle collides with the heatbed, an alert appears, and the print process pauses.



i TIPS

- After enabling clumping detection by probing feature, a prime tower of 20 layers (or matching the model's height) will be automatically generated to prevent oozing.
- The prime tower can be manually disabled in **Prepare Page > Process > Others > Prime Tower**, but this may cause surface blemishes on the model.

Build Plate Detection

The live view camera checks the presence and type of the build plate on the heatbed. If the currently placed build plate does not match the print job settings, the print job will be paused to prevent failure.

The P2S printer comes with a Bambu Textured PEI Plate. To use the old Bambu Textured PEI Plate on P2S, disable build plate detection.

⚠ CAUTION

If build plate detection is disabled, the printer will not check if the build plate is in place before printing. Please verify manually to avoid print failure.

6.2 Adaptive Airflow System

The adaptive airflow system consists of an adaptive airflow switching unit and air filters. It automatically switches the air ducts based on the current air management mode to regulate the internal temperature and airflow, providing the optimal printing environment for different filaments.

🔗 NOTE

The printer can automatically select the appropriate air management mode for different filaments. This ensures that high-temperature filament does not cause warping and low-temperature filament does not clog. To manually select or adjust a specific mode, choose **air condition** on the printer Screen.

6.2.1 Cooling Mode

When printing heat-sensitive filaments such as PLA and TPU, the air management system is set to cooling mode.

In this mode, the built-in **auxiliary part cooling fan*** draws in external air to reduce the chamber temperature. Meanwhile, the exhaust air is expelled through **the vents at the back of the printer and the gaps around the purge wiper****.

🔗 NOTE

You can install another auxiliary part cooling fan on the printer's left side for a more balanced cooling effect. Visit the Bambu Lab Wiki (wiki.bambulab.com/home) and search for "Auxiliary Part Cooling Fan - Left - Installation Guide for the P2S" to access detailed instructions.

🔗 NOTE

You can install the external exhaust fan kit on the printer's back to actively expel fumes from inside the printer. Visit the Bambu Lab Wiki (wiki.bambulab.com/home) and search for "P2S External Exhaust Fan" to access detailed instructions and feature guides.

6.2.2 Heating Mode

Suitable for printing filaments with high heat resistance, such as PETG, ABS, ASA, PC, and PA. The air management system switches to Heating Mode.

In this mode, the airflow switching unit stops drawing in external air. Instead, it forms an internal circulation with the air filter, ensuring even warm air inside the chamber while filtering and purifying the air.

6.3 Auto Leveling and Calibration

The printer has multiple built-in calibration functions that detect and adjust key printing conditions before printing. These functions ensure the initial layer quality and extrusion performance, improving overall print quality.



Auto bed leveling and flow dynamics calibration are preset on the print job page. You can view or configure them in **advanced options** when initiating a print from the printer screen, Bambu Handy, or Bambu Studio.

i TIPS

- It's recommended to perform a full print calibration (see [print calibration](#)) after prolonged use or moving.
- To manually calibrate flow dynamics for filament, go to Flow Dynamics section on the Bambu Studio **Calibration page**, select **Manual Calibration**, and follow the prompts.

6.3.1 Auto Bed Leveling

Calibrate the flatness of the heatbed by contacting the nozzle with the build plate. It can make the extrusion height more uniform.

This feature has three states: Auto, On, and Off. In Auto mode, the printer first performs leveling detection (about 10 seconds). If the result is the same as last time, the process is skipped.

6.3.2 Flow Dynamics Calibration

This calibration can resolve issues like under-extrusion and over-extrusion caused by pressure lag inside the nozzle. It determines the optimal dynamic flow coefficient for the filament to maintain stable extrusion at fast print speed and improve print quality.

This feature has three states: Auto, On, and Off. In Auto mode, the printer checks if the filament has been calibrated recently. The process is skipped if calibration was done.

i NOTE

See [Filament Calibration](#) section for applicable scenarios and step-by-step guidance.

Chapter 7 Basic Controls and Functions

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

NOTE

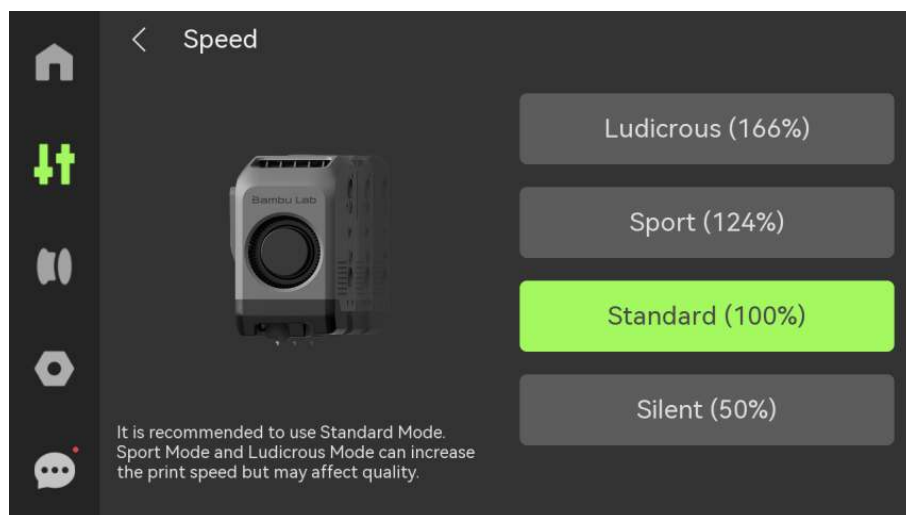
The interface is continuously updated to improve the user experience. Please refer to the actual interface.

7.1 Common Screen Operations

This printer is equipped with a full-color touchscreen for displaying the status of the printer and its main components. It also allows you to configure multiple printer settings.

7.1.1 Print Speed

Adjust the print speed in real time during printing. This affects print time and model surface quality. On the printer screen, select **⬆️⬆️** > **Speed** to adjust the print speed. Four modes are available:



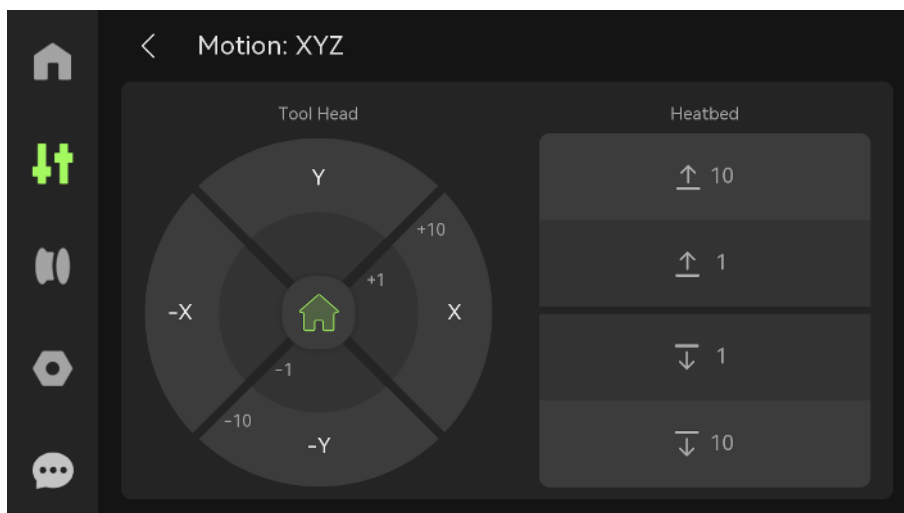
- **Ludicrous:** 166% of normal print speed and acceleration.
- **Motion:** 124% of normal print speed and acceleration.
- **Standard:** normal print speed and acceleration.
- **Silent:** 50% of normal print speed and acceleration.



You can flexibly adjust the print speed based on actual needs. Generally, increasing the speed will shorten the print time but may result in poorer surface quality. Reducing the speed improves surface quality and reduces printer noise but will increase print time.

7.1.2 XYZ Axes 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 **X/-X** and **Y/-Y** to move the toolhead. The inner ring (+1/-1) moves it in small increments, while the outer ring (+10/-10) moves it in large increments.
- **Heatbed:** Tap  or  to raise or lower the heatbed. 1 indicates a small movement, and 10 indicates a larger movement.


7.1.3 Nozzle and Extruder

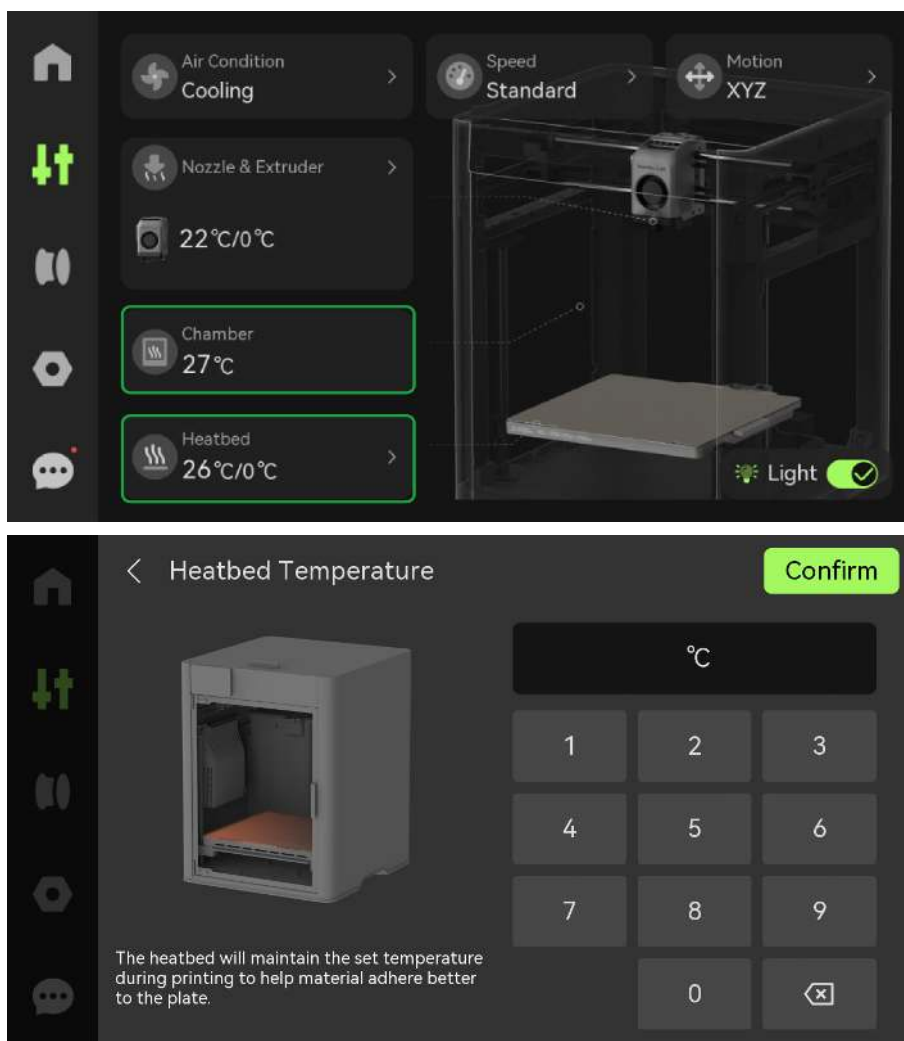
Use this page to set the nozzle temperature and information, view extruder load status, and manually extrude or retract filament. It is useful for routine maintenance, filament replacement, nozzle cleaning, and print preparation.

On the printer screen, select **Settings**  > **Nozzle & Extruder** to set the nozzle and extruder parameters.

1. **Extruder load status:** The extruder shows the color of the filament currently printing or loaded.
2. **Control the extruder:** Use the  or  button to extrude or retract 1 cm of filament.
3. **Set nozzle temperature:** Enter a value to set the nozzle temperature (°C).
4. **Set nozzle information:** After replacing the hotend, manually update its type, material, and diameter.


7.1.4 Heatbed and Chamber Temperature

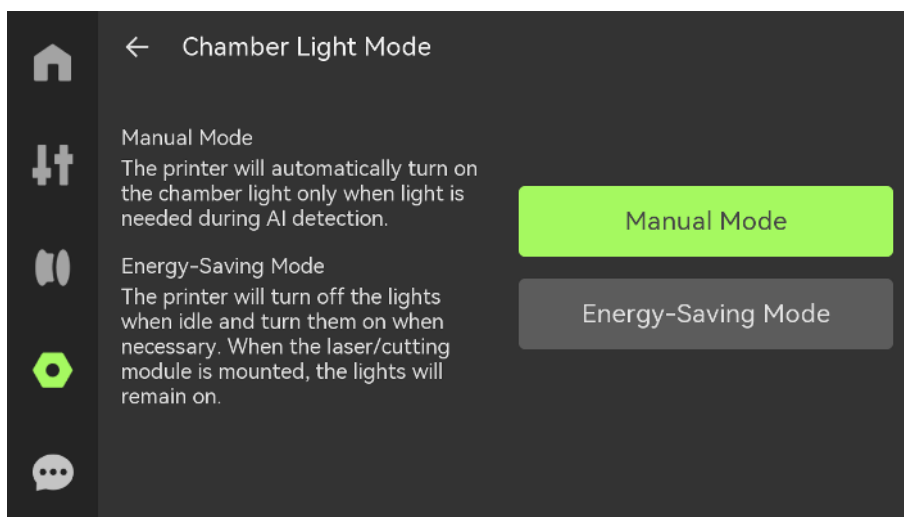
View the real-time chamber temperature and heatbed temperature directly on the  page. Click the **Heatbed** button, enter a desired value to manually set the temperature.



7.1.5 Chamber Light Mode

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

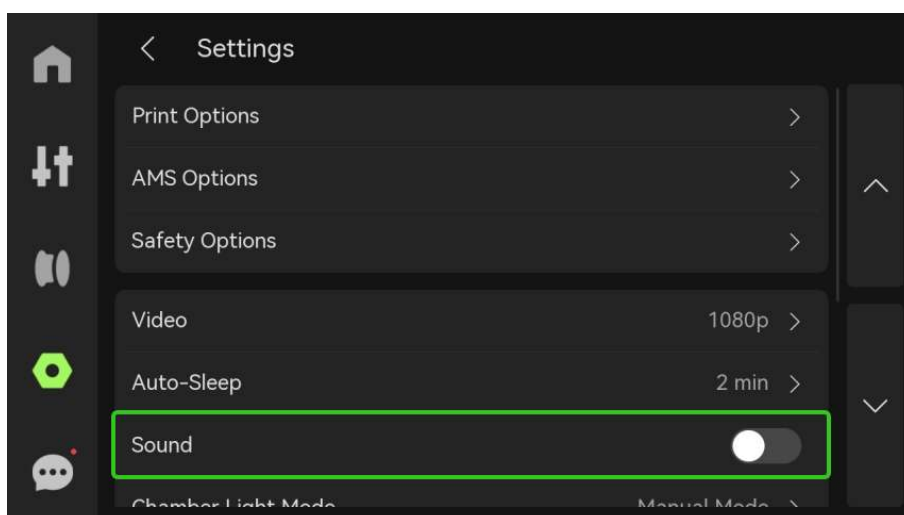
- **Manual Mode** :The light turns on automatically when required for AI detection. When checking print status or performing maintenance, you can manually switch the light on or off via  > **Light**.
- **Energy-Saving Mode**: The light automatically turns off when the printer is idle and only turns on when needed to save energy, such as during printing.



7.1.6 Sound

When enabled, the printer plays a notification sound when it powers on and when printing starts or finishes.

On the touchscreen, select  > **Settings** > **Sound** to turn it on or off.



7.2 Photo and Video

The printer is equipped with a 1920×1080 HD live view camera mounted at the upper left of the printer's front pane. It's used for real-time monitoring, time-lapse, photography, video recording, and intelligent detection. This chapter focuses on the printer's photo and video recording features.

TIPS

For privacy protection needs, you can visit MakerWorld (makerworld.com) and search for "P2S Privacy Cover" under the Bambu Lab official account. Print and install the live view camera cover.

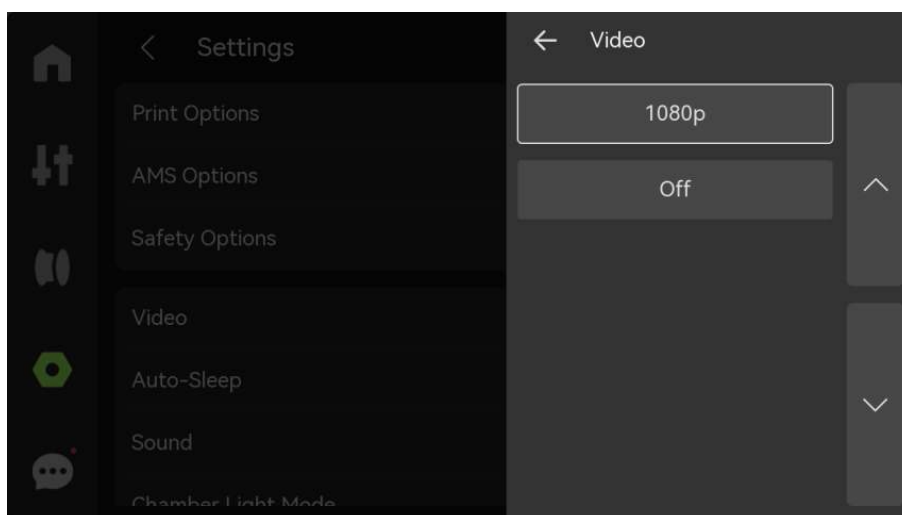
Note: After the privacy cover is installed, camera-related functions (such as photo, video, and visual detection) will be unavailable.

7.2.1 Video Recording

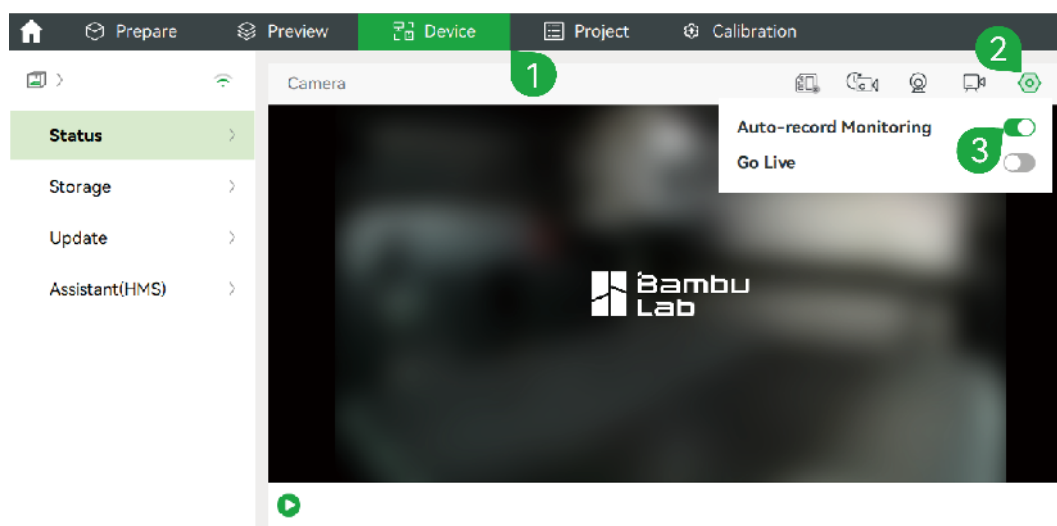
The printer supports video recording throughout the entire print process, making it easy to review print status at any time and providing useful reference for troubleshooting and after-sales support. The printer includes a USB port to connect a USB flash drive for saving video files.

Insert a flash drive before enabling this feature. You can enable or disable video recording using the following methods:

- On the printer screen, select  > **Settings** > **Video** to turn the recording feature on or off and adjust the resolution.





- In Bambu Studio, click **Device** > **Camera** >  > **Auto-record Monitoring** to enable/disable the video recording.

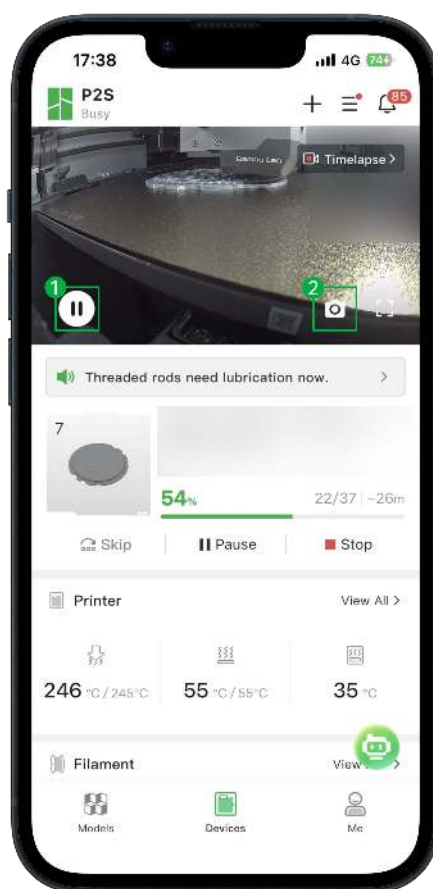


7.2.2 Photo

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. Go to **Device** page in Bambu Handy:

1. Tap  to view the camera preview.

2. Tap  at the bottom right of the camera preview to take a photo. The photo will be saved to your phone's photo album.



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. Before printing, insert your USB drive into the printer's USB port to store the time-lapse video. See [Connect USB Flash Drive](#) for specifications and usage tips.

Activation Methods

When starting a print job, you can choose whether to enable timelapse.

- **Printer Touchscreen:** On the **Plate to Print** page, expand **Advanced Options > Timelapse**, then tap **Enable**.
- **Bambu Studio:** After slicing, click **Print plate/Print all** in the upper-right corner. In the pop-up **send print job** window, find **Timelapse** and turn it **On**.
- **Bambu Handy:** On the **Prepare** page, go to **Options > Timelapse**, and tap to enable Timelapse.

NOTE

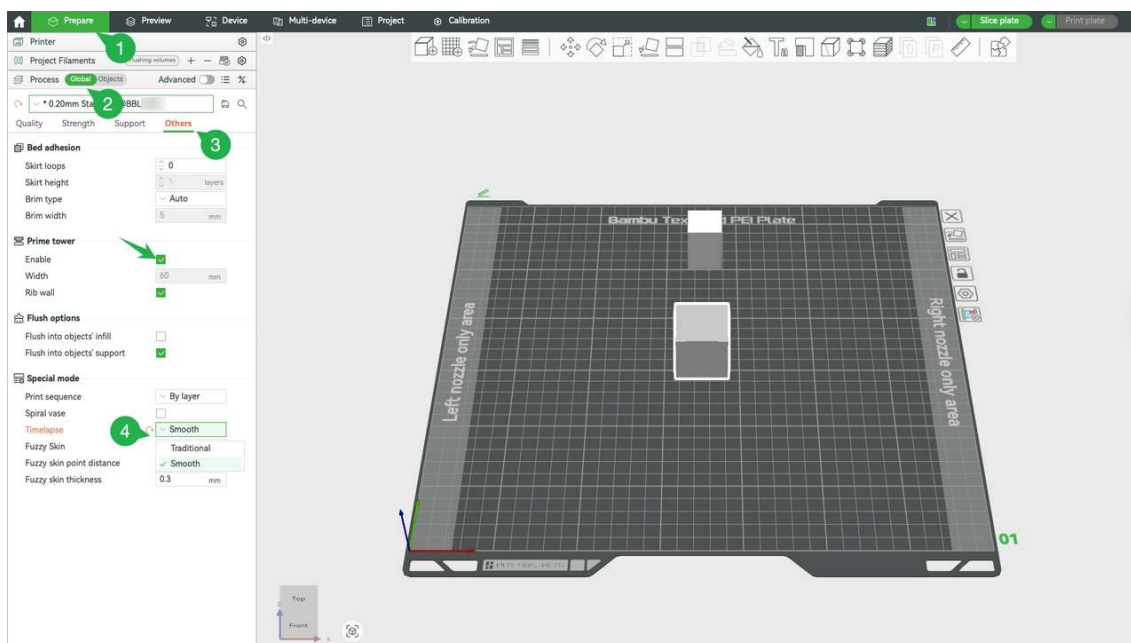
The interface is continuously updated to improve the user experience. Please refer to the actual interface.

Shooting Mode

There are two time-lapse modes:

You can switch the time-lapse photography mode on the **Prepare** page in Bambu Studio by navigating to **Process > Global > Others > Special Mode > Timelapse**.

- **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.
- **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.



TIPS

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

Download Methods

Once a print job completes, the time-lapse video is saved on the USB drive or printer's internal storage. You can also view and download time-lapse videos from the device page in Bambu Studio and Bambu Handy.

- **Download via USB Flash Drive:** On the touchscreen, tap **USB Storage > Eject**. Remove the USB drive and open the **timelapse** folder to find the video files.
- **Download via Bambu Studio:** On the device page, click **Storage > Timelapse**. Select the desired time-lapse video to download or delete.
- **Download via Bambu Handy:** On the device page, tap the **Timelapse > External Storage** icon at the top right corner of the video. Select the desired timelapse video to download or delete.

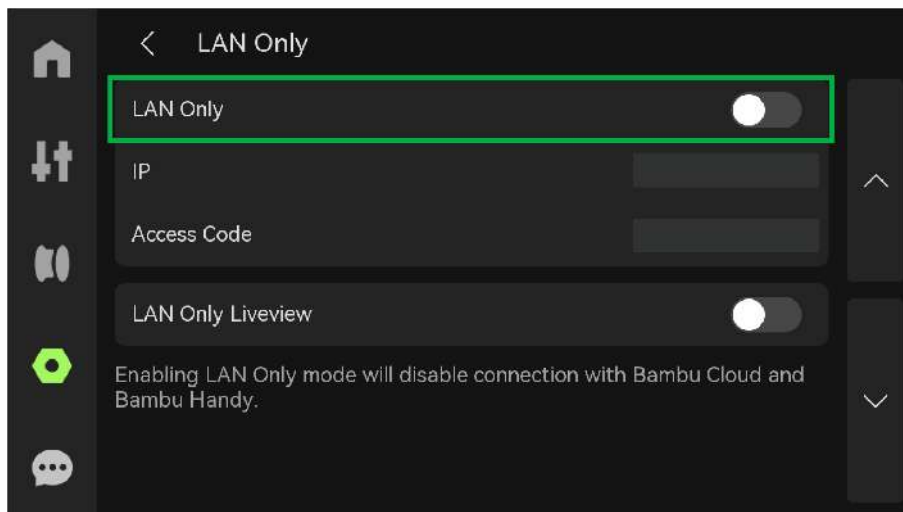
7.3 Connect to Network

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 users or enterprise environments with higher data security and privacy requirements.

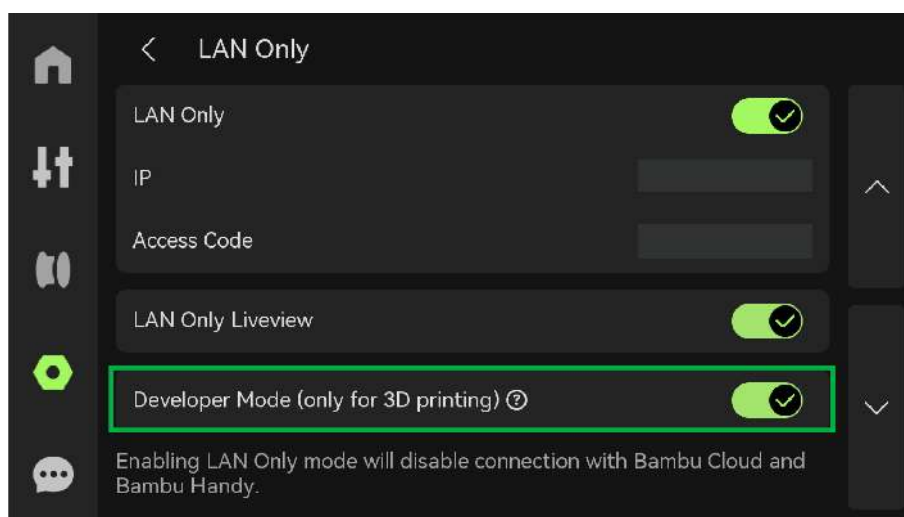
On the printer screen, select  > **Settings** > **LAN Only** to enable LAN Only mode.



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 customization but may involve security risks.

After enabling LAN Only mode, you can activate 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 Specifications

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

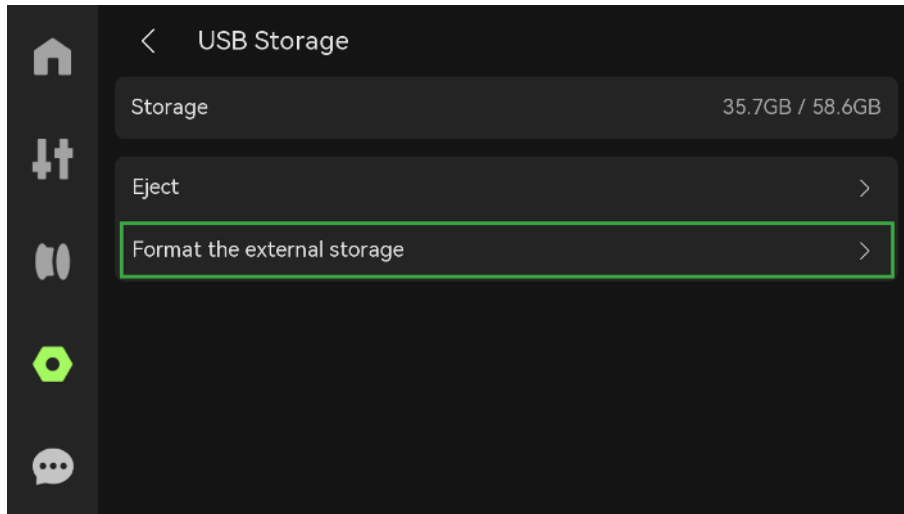
NOTE

- 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 USB drive as FAT32 (suitable for USB flash drives up to 2 TB).



7.4.3 Eject

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

- After the new firmware is released, a prompt will appear on the printer screen. Tap **Yes** and follow the instructions to complete the update.
- You can also update manually by going to  > **Firmware** > **Update**.

7.5.2 Initialization

Reset the printer to its factory default state. All custom settings and user data will be erased. This action is irreversible, so make sure no important data remains before proceeding.

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

Chapter 8 Filament Introduction and Usage

This chapter introduces common filament characteristics, pre-processing methods, and print requirements for specific filaments.

Choose the appropriate filament based on different scenarios. Then, check whether the print conditions are compatible with your current printer and AMS model. Pre-process the filament (drying and calibration) as needed before start a print to ensure stable and high quality results.

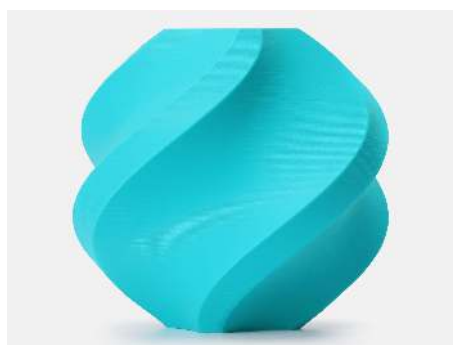
8.1 Common Filament Types

Different 3D printing filaments have their own physical properties and applications. Choosing the right filament based on your printing needs helps improve print quality and model performance. Below is a brief introduction to several common filaments. For detailed features and specifications, see the Bambu Lab Filament Guide (<https://bambulab.com/filament/guide>).

Basic Filaments

PLA

- The most common entry-level 3D printing filament. It is eco-friendly, non-toxic, biodegradable, and easy to print, but has limited mechanical performance.
- Suitable for everyday prototyping, home printing, and low-load applications.



PETG

- Offers good mechanical performance, with water, heat, and chemical resistance. However, it must be kept dry, and its smooth surface can show scratches easily.
- Suitable for containers, durable parts, and functional parts.



Aesthetic Filaments

PLA Silk/Silk+

- Has a silky texture and high gloss. However, it must be kept dry, as moisture can easily cause stringing.
- Suitable for models that require a silky texture or metallic sheen.



PLA Wood

- Contains natural wood particle additives for a matte, solid-wood-like texture. However, it must be kept dry, as moisture can easily cause discoloration.
- Suitable for natural or forest-style models. The matte texture also helps hide layer lines.



Engineering Filaments

ABS

- Offers good mechanical performance and high durability, but is prone to warping.
- Suitable for mechanical housings and functional parts.



ASA

- Similar to ABS but with superior UV resistance.
- Suitable for outdoor parts and automotive components.



PC

- Offers good heat resistance, strength, and rigidity, but is prone to warping and moisture absorption.
- Suitable for high-strength structural parts, high-temperature applications, and mechanical components.



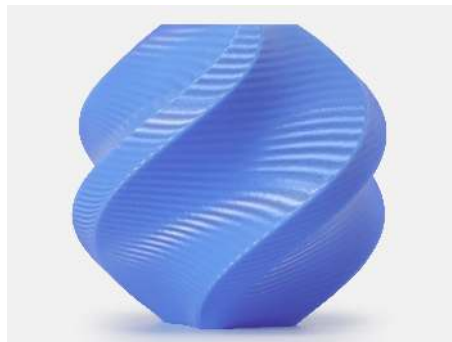
NOTE

Engineering filaments may emit irritating fumes during printing. Use enclosed printers and ensure good ventilation.

Flexible Filaments

TPU

- Highly elastic and wear-resistant, but relatively slow to print.
- Suitable for flexible models, such as shoe sole prototypes, seals, and cushioning pads.



NOTE

- The hardness of TPU filament is measured by the Shore A hardness. A lower value means the filament is softer, while a higher value means it is harder. Common hardness levels include 75A, 80A, 83A, 85A, 90A, and 95A.
- It is recommended to print with TPU of 80A hardness and above. Higher hardness improves printing stability and reduces the risk of print failure.

Fiber-Reinforced Filaments

Carbon Fiber Reinforced (-CF)

- Carbon fiber is added to filaments such as PLA or PETG to significantly improve rigidity and strength while keeping the material lightweight.
- Suitable for load-bearing structural parts and lightweight structural designs.



Glass Fiber Reinforced (-GF)

- Glass fiber is added to filaments such as ABS to improve toughness and wear resistance.
- Suitable for industrial mechanical parts and structural frame components.



Support Filaments

Support for PLA/PETG

- A breakaway support filament with good print compatibility, easy removal, and smooth contact surfaces.
- Designed specifically for use with PLA and PETG printing.



PVA (Water-soluble)

- A flexible, biodegradable polymer with strong moisture absorption. It absorbs water from the air and dissolves in water. Commonly used as a water-soluble support filament in 3D printing.
- Suitable for complex models or models where supports are difficult to remove manually.



NOTE

Support filaments are specially optimized for good stability. They should only be used for printing support structures. Using them for the model body may reduce model strength and surface quality.

8.2 Printing Conditions

The P2S is a fully enclosed 3D printer equipped with a hardened steel 0.4 mm hotend. The hotend supports a maximum temperature of 300°C, and the heatbed can reach up to 110°C, enabling stable printing from basic filaments to engineering filaments.

The following lists common printing conditions for Bambu filaments. Please confirm all settings before printing to ensure stable and reliable print quality. For detailed filament parameters and printing guides, visit Bambu Lab Wiki (wiki.bambulab.com/home) and search for the filament name, such as PLA.

Symbol Guide

NOTE

✓ indicates recommended, required, or mandatory actions, such as mandatory drying.

○ indicates allowed or suggested actions, such as recommended drying.

✗ indicates prohibited or incompatible actions, such as do not support hotend with 0.2 mm diameter nozzle.

Filament	Nozzle Requirements		Printing Requirements		AMS*
	Name	Diameter*	Material*	Temperature (°C)*	Drying
PLA Basic	All	All	190–230 °C	○	All
PLA Wood	× 0.2 mm	All	190–230 °C	○	All
PLA Silk/Silk+	× 0.2 mm	All	190–230 °C	○	All
PETG HF	All	All	230–260 °C	✓	All
TPU 90A	× 0.2 mm	Incompatible with High Flow versions	200–250 °C	✓	AMS HT only
ABS	All	All	240–280 °C	○	Incompatible with AMS lite
ASA	All	All	240–280 °C	○	All
PC	× 0.2 mm	All	260–290 °C	✓	All
PLA-CF	× 0.2 mm	Hardened steel nozzle only, High Flow version not recommended	210–240 °C	○	All
PETG-CF	× 0.2 mm	Hardened steel nozzle only	240–270 °C	○	Incompatible with AMS lite
ABS-GF	× 0.2 mm	Hardened steel nozzle only	240–280 °C	✓	Incompatible with AMS lite
PVA	All	All	190–240 °C	○	All
Support for PLA/PETG	All	All	190–240 °C	○	All

TIPS

- **Nozzle Size:** Includes nozzles with diameters of 0.2 mm, 0.4 mm, 0.6 mm, and 0.8 mm.
- **Nozzle Materials:** Includes hardened steel and stainless steel, as well as their high-flow versions.
- **AMS Compatibility:** Includes AMS 1st generation, AMS 2 Pro, AMS HT, and AMS lite.

⚠ CAUTION

Based on printing temperature, filaments can be classified as low-temperature, medium-temperature, or high-temperature materials. **When printing with multiple materials, follow these requirements to avoid extruder clogs or damage.**

- Do not mix high-temperature and low-temperature filaments in the same print. When starting a print from the printer screen, Bambu Handy, or Bambu Studio, the system automatically restricts this combination.
- When mixing high-temperature and medium-temperature filaments, or medium-temperature and low-temperature filaments, adjust the nozzle and chamber temperatures carefully to prevent lower-temperature filaments from softening too early.

8.3 Filament Preparation

Filament dryness and flow settings often affect print quality. For example, damp PLA Silk is more likely to cause stringing during printing. When using third-party filament, the preset flow settings may not provide the best print results.

This section explains when drying and calibration are needed and provides step-by-step instructions. You can prepare the filament as needed.

8.3.1 Filament Drying

When stored in a non-sealed environment, some filaments absorb moisture from the air. During printing, the moisture rapidly vaporizes in the hot nozzle, causing stringing, holes, reduced part strength, and other issues.



Filament not dried



Filament dried

For moisture-sensitive filaments, dry the filament before printing for more stable print quality:

- PLA: PLA Wood, PLA Silk/Silk+, PLA-CF, PLA Aero.
- PETG: PETG HF, PETG Translucent, etc.
- PVA, PC, PA, TPU, and others.
- All carbon fiber/glass fiber reinforced filaments.

NOTE

Please visit MakerWorld (makerworld.com), search for “stringing”, download a suitable test model, and print it to determine whether the filament needs drying based on the print quality.

Drying Filament with the Heatbed

To dry filament using the printer’s heatbed, prepare a container, such as the original filament package box or a **filament drying hood***.

TIPS

Please visit MakerWorld (makerworld.com), search for the “filament drying hood” model, and print it using high-temperature filaments, such as PC or PA-CF.

On the printer screen, select  > **Toolbox** > **Dry Filament**. Follow the on-screen instructions to complete the steps:

- Step 1. Remove any debris from the build plate and the bottom of the printer chamber.
- Step 2. Tap **Prepare**. The toolhead and heatbed will automatically move to the preset positions.
- Step 3. Place the filament on the build plate, then cover it with a paper box or filament drying cover.
- Step 4. Select the **filament type**. The printer will automatically set the heatbed temperature and drying duration. You can also set these manually.
- Step 5. Tap **Start** to begin drying.

NOTE

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

Filament Drying with AMS 2 Pro/AMS HT

AMS 2 Pro and AMS HT have built-in drying modules that heat and dry filament. During drying, the spool rotates automatically for more even results. When drying filament with AMS, follow the notes and steps below:

AMS Model	Filament Capacity	Max Temperature
AMS 2 Pro	4	65 °C
AMS HT	1	85 °C

⚠ CAUTION

- When using multiple AMS units for drying at the same time, only one unit can draw power from the printer. The other AMS units must be connected to a power outlet using the official power adapter.
- To enable “**printing while drying**,” upgrade the printer and AMS firmware to the latest version to activate this feature.
- If drying while the printer is idle, it's recommended to select "rotate spool while drying" for better effects. Pull the filament out of the AMS filament inlet and secure it to the inside of the spool with tape to prevent it from unwinding.

Step 1. Connect the AMS to the printer. See the Quick Start Guide or [Initial installation](#).

Step 2. Place the filament into the AMS slot, then close the AMS cover and locking tab.

Step 3. On the printer screen, go to the  filament page, then tap the  icon.

Step 4. Select the **filament type**. The printer automatically applies the preset drying temperature and duration. Tap **Start** to begin drying.



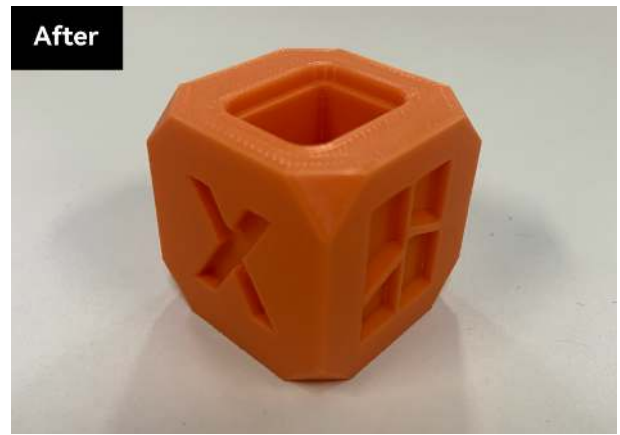
Other Drying Methods

You can use a forced-air dryer to dry filament, such as high-temperature filaments like PPS. Please visit Bambu Lab Wiki (wiki.bambulab.com/home) and search for “filament drying guide” to find drying parameter tables for various filaments.

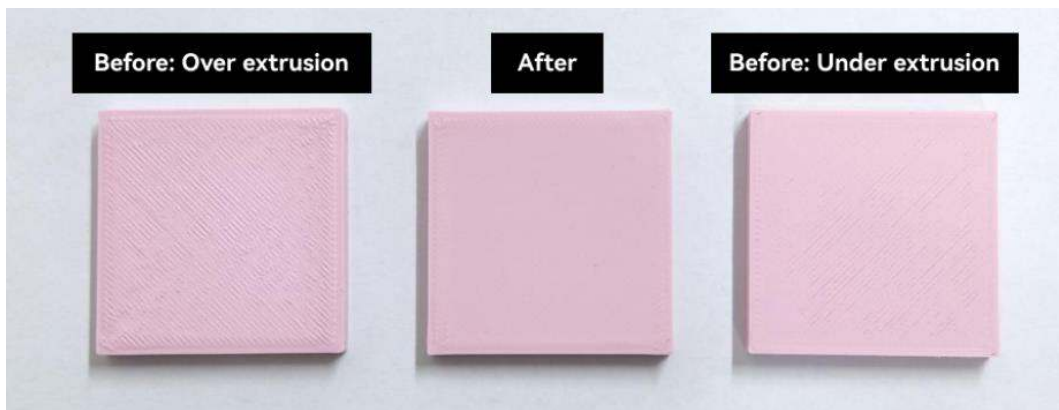
8.3.2 Filament Calibration

Filament calibration includes two dimensions: flow dynamics and flow rate. Visit Bambu Lab Wiki (wiki.bambulab.com/home) and search for "flow dynamics calibration" and "flow rate calibration" to learn about calibration principles and detailed steps.

- **Flow dynamics calibration** compensates for extrusion pressure lag and improves extrusion stability, such as print quality around model corners.



- **Flow rate calibration** helps reduce under-extrusion and over-extrusion, improving surface smoothness, layer adhesion, and more.



If you are using Bambu filament, you can directly apply the official preset parameters without frequent filament calibration. When **changing filament brands** or **hotends**, or encountering **specific print quality issues**, follow the applicable calibration scenarios and steps below.

Flow Dynamics Calibration

The printer supports both **automatic** and **manual** calibration. Flow Dynamics Calibration is recommended when using third-party filament, replacing the hotend, or changing filament settings, such as max volumetric speed or print temperature in Bambu Studio.

i TIPS

Visit Bambu Lab Wiki (wiki.bambulab.com/home) and search for "flow dynamics calibration" to learn how to determine the coefficient for manual calibration.

Calibration Type

Type	Stage	Platform	Method
Manual calibration	When the printer is idle	Bambu Studio only	Visually check the print quality of the test model and manually enter the corresponding coefficient.
Auto-calibration	When the printer is idle or starting a print job	Printer screen, Bambu Handy, and Bambu Studio	The printer automatically performs calibration and provides the recommended result.

Calibration Steps

- **Calibrate when starting a print job**

On the Send print job page, Enable flow dynamics calibration to automatically calibrate the filament used for printing. If you cannot find this option, expand Advanced Options.

- **Calibrate when the printer is idle:**

1. On the **Calibration** page in Bambu Studio, click **Flow Dynamics**, select **Auto-Calibration** or **Manual Calibration**.
2. Select the printer **preset** and wait for the printer to start calibration. Then enter the filament **coefficient (K)** based on the recommended automatic calibration result or your visual inspection of the printed test model.

Flow Rate Calibration

If the preset flow rate does not match the filament, under-extrusion or over-extrusion may occur, causing poor surface quality, weak layer bonding, and other issues. However, these issues are not always caused by an incorrect flow rate. Before running flow rate calibration, first check the printer's basic calibration items and the model structure. See [Print Quality Issues and Solutions](#).

If these issues have been ruled out and the print defect is mainly related to a specific filament, follow the notes and steps below:

- Step 1. On the **Calibration** page in Bambu Studio, click **Flow Rate** and select **Manual Calibration**.
- Step 2. Select the printer **Preset** and **Calibration Type**. **Complete Calibration** is recommended for first-time use.
- Step 3. Wait for the printer to start calibration, then enter the filament coefficient (K) based on the recommended automatic calibration result or your visual inspection of the test model.

TIPS

Visit Bambu Lab Wiki (wiki.bambulab.com/home) and search for "flow rate calibration" to learn how to determine the calibration coefficient.

8.4 Special Filament Printing Requirements

8.4.1 TPU

Hardness Classification

TPU is a flexible filament that easily absorbs moisture. It has higher requirements for storage, dryness, and loading method. TPU filaments with different hardness levels also vary in printer compatibility and loading method. Common TPU hardness levels and supported feeding methods are listed below:


Hardness Level		Bambu Lab Printer	AMS	Supported Loading Method
68D (TPU for AMS)	High	Compatible	Compatible	Filament buffer, PTFE tube bracket (printer filament inlet)
95A	Relatively high	Compatible	Incompatible	Printer filament inlet
90A, 85A	Medium	Compatible	Incompatible	Toolhead filament inlet
Below 85A	Low	No	No	/


CAUTION

TPU filaments, except TPU for AMS, cannot be loaded through the printer's filament buffer.

Print Preparation

You can visit Bambu Lab Wiki (wiki.bambulab.com/home) and search for "Printing Guides for TPU 85A and 90A" to get detailed instructions and illustrated guidance. The following general TPU checklist can help you prepare before printing:

Stage	Preparation	Steps
Before printing	Dry the filament	Select the drying equipment and settings based on the filament type
	Check printer configuration	Make sure the current AMS, build plate, and hotend support TPU printing
	Clean the printer	<ul style="list-style-type: none"> Clean and dry the build plate Use a new hotend if possible, or perform cold pull maintenance (do not use fiber reinforced filament) Check the extruder gear for debris and clean it if needed
Load filament	Confirm the loading method	Determine the supported filament inlet method based on TPU hardness, such as printer PTFE tube bracket or toolhead filament inlet (recommended)
	* Prepare a filament sealed box and spool shaft	For longer prints, place the filament in a sealed container, such as a rice storage container, and print a spool shaft holder for filament loading
	* Use the filament spool holder rack	Print a holder rack for the filament spool or sealed box, and make sure the box outlet is higher than the toolhead
	Extrude manually	On the printer screen, tap  at low frequency to manually extrude

Stage	Preparation	Steps
Initiate a Print	Adjust filament parameters and model position	<ul style="list-style-type: none"> Check whether the filament print temperature and heated bed temperature is appropriate, then disable the Flow Dynamics Calibration option when starting the print job If the model is pre-positioned at the corner of the build plate, move the model to the center in Bambu Studio
After printing	Unload Filament	Heat up the hotend, tap  at low frequency on the screen and manually pull out the filament
	Store the filament	Store the filament in a sealed box with desiccant, and keep the ambient humidity below 20% RH.

 **NOTE**

When printing TPU that is not compatible with AMS, prepare a filament sealed box and sealed container yourself. Visit MakerWorld (makerworld.com), search for "top spool holder TPU," download a suitable holder model, print and install it according to the model instructions.

8.4.2 High-Temperature Filament

High-temperature filaments, such as ABS, ASA, PC, and nylon, offer excellent strength and heat resistance, but require higher nozzle temperatures, chamber temperatures, and better print environment control. Before printing, check the configuration and adjust the slicing settings to reduce the risk of warping, cracking, and nozzle clogging. Use the following general checklist to prepare:

Stage	Preparation	Steps
Before printing	Dry the filament	Select the drying equipment and settings based on the filament type
	Check printer configuration	Make sure the current build plate and hotend support high-temperature filaments. Do not use the Bambu Cool Plate SuperTack
	Clean the printer	<ul style="list-style-type: none"> Clean and dry the build plate, then apply Bambu glue stick or adhesive to improve stability Perform cold pull maintenance on the hotend
Initiate a Print	Adjust model settings in Bambu Studio	<ul style="list-style-type: none"> Set an appropriate layer height. The default 0.2 mm is recommended For multiple models, select By object for print sequence

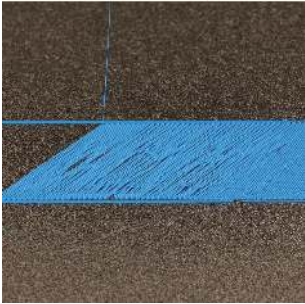
Stage	Preparation	Steps
	Preheat the printer	<p>Set the required chamber temperature in advance according to filament type. Confirm the air management is set to Heating Mode</p> <p>* You may also adjust the heatbed temperature to help heat up the chamber temperature</p>
After printing	Remove the model	<ul style="list-style-type: none"> • Wait until the model has fully cooled to room temperature before removing it • Remove supports within 2 hours after printing to prevent moisture absorption
	Post-process the model	<ul style="list-style-type: none"> • Annealing is recommended for models printed with some filaments • For moisture-sensitive filaments, apply a water-resistant coating such as paint or wax
	Future use	To use low-temperature filaments later, set the nozzle temperature to 250–300 °C and manually extrude filament to ensure smooth printing

 **NOTE**

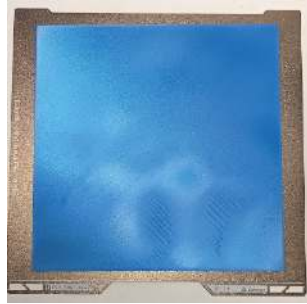
Visit Bambu Lab Wiki (wiki.bambulab.com/home) and search for “high-temperature filament” for detailed instructions and illustrated guides.

Chapter 9 Print Quality Issues and Solutions

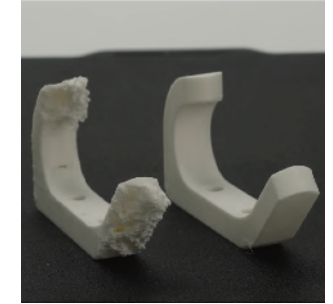
9.1 Common Print Quality Issue Overview



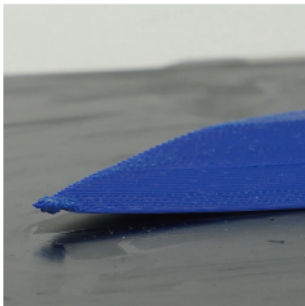
First layer not sticking



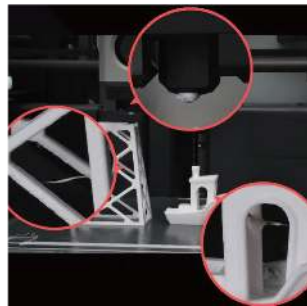
First layer too high/too low



Poor overhang quality



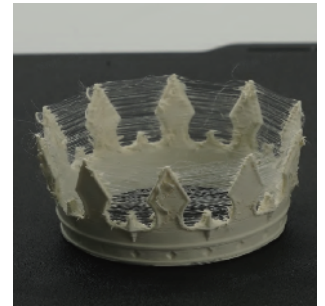
Model warping, falling off, or collapse



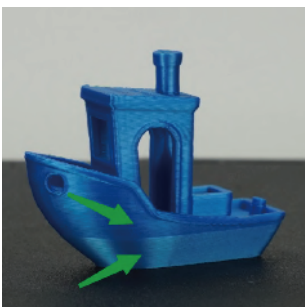
Filament sticking to the nozzle



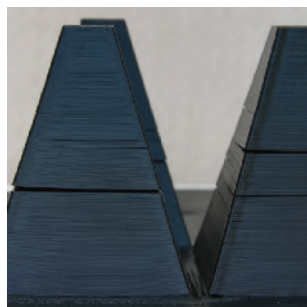
Under-extrusion



Stringing or oozing



Gloss difference



Interlayer cracking

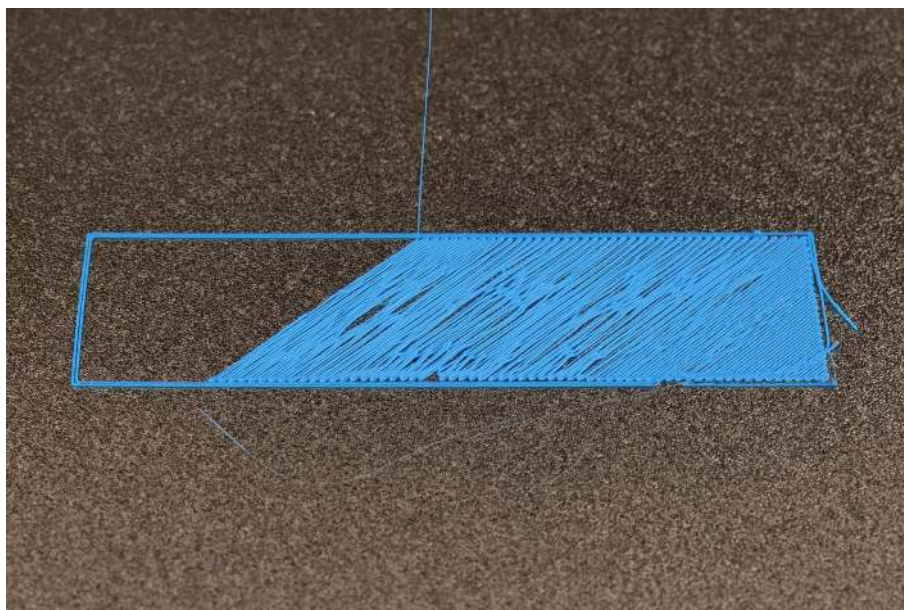


Seam



Top layer gaps

9.2 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.

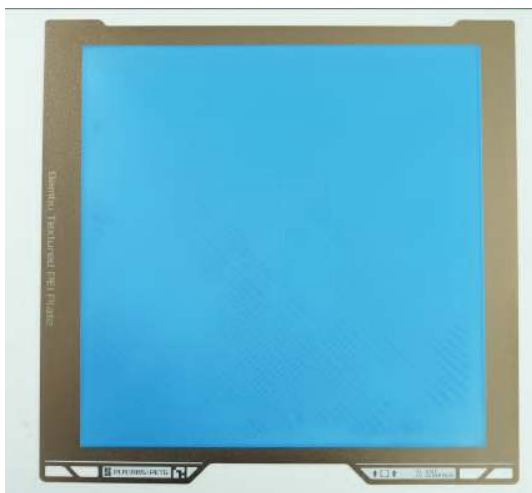
2. Dirty build plate surface

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

3. Bed leveling not performed

On the printer screen, select **Calibration** > **Print Calibration** > **Bed Leveling** to perform heatbed leveling. Alternatively, on the Device page in Bambu Studio, select **Calibration** > **Bed Leveling** to perform heatbed leveling.

9.3 First Layer Too High/Too Low




Full-Plate 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 "P2S First-Layer" to obtain a detailed tutorial.

9.4 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 the angle from vertical is greater than 45°, add support structures.

2. Print speed too high

Reduce the print speed, or enable the Slow down for overhangs function.

3. Insufficient cooling

Lower the nozzle temperature as needed, increase the auxiliary part cooling fan and part cooling fan speeds, and open the printer front door and top glass cover to improve heat dissipation.

9.5 Model Warping, Falling Off, or Collapse

During printing, if the model shrinks locally or does not adhere firmly to the build plate, it may warp, detach completely, or even collapse. Warped areas are often accompanied by a visible horizontal ridge. This happens because the warped area moves closer to the nozzle, causing the extrusion lines to be flattened and overflow.



Possible Causes and Solutions

1. **Tall model with unstable center of gravity**

Add supports and reduce print speed and acceleration. Lay the model flat whenever possible, or split the model before printing if needed.

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**

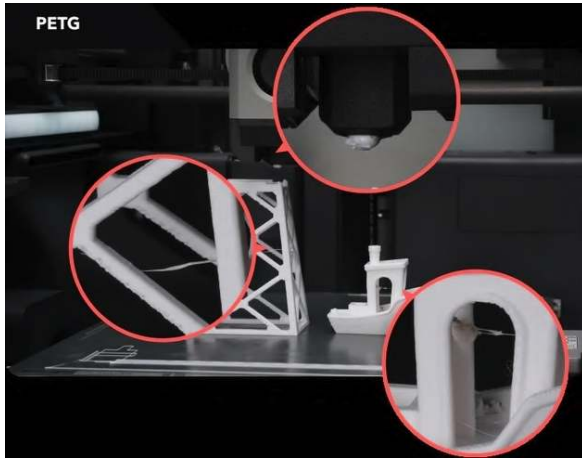
Raise the heatbed temperature appropriately, close the printer's front door and top glass cover, reduce fan speed, lower sparse infill density, and select non-crossing infill patterns like Gyroid.

5. **Insufficient build plate 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.

9.6 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 like PETG.



Sticking nozzle



Clean nozzle

Possible Causes and Solutions

1. Damp 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 is too slow or the temperature is 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 toolhead against it after powering off. Repair or replace it if abnormalities are found.

5. Too many travel paths in the model

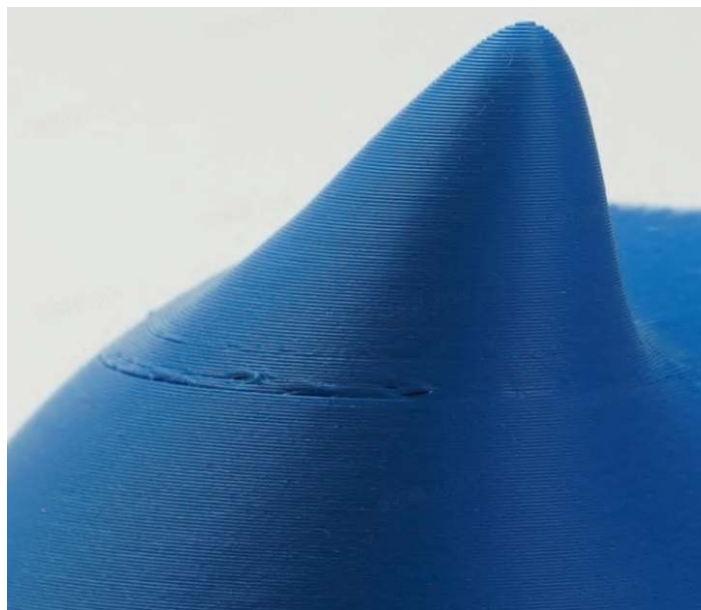
Optimize the print layout path in Bambu Studio to avoid excessive travel paths between model sections.

9.7 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**.



Overall 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.

9.8 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.

Possible Causes and Solutions

1. Damp filament

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

2. Long travel distance and insufficient retraction length

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

3. Special model structure or improper plate arrangement

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 (e.g., PLA Aero) without reducing temperature or flow rate**

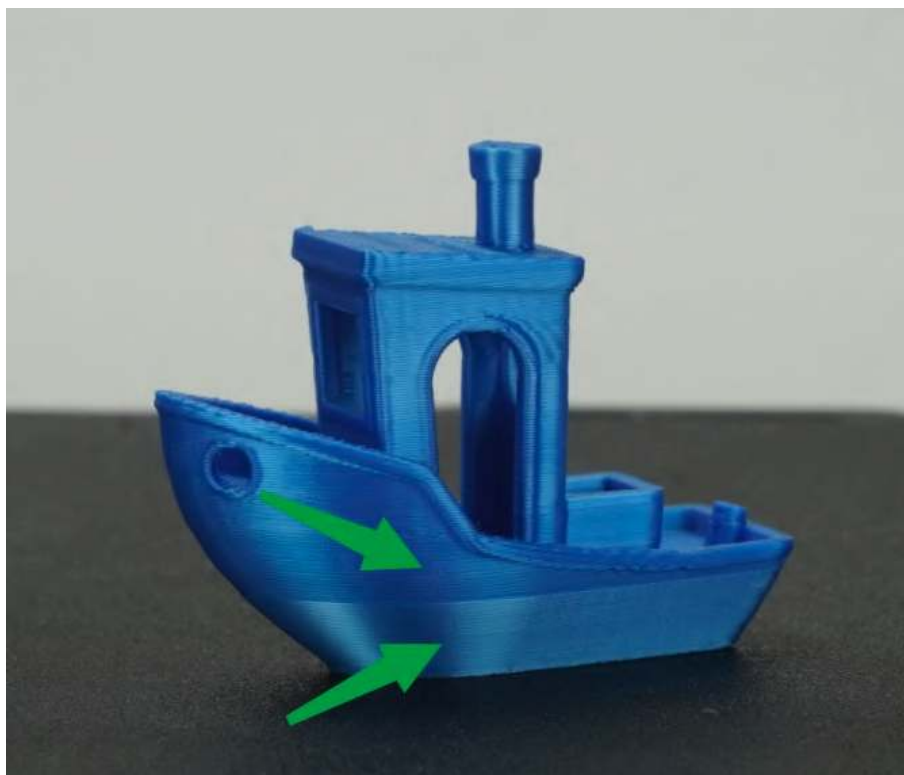
Lower the print temperature appropriately and adjust the flow rate to the range of 0.5 – 0.7.

6. **Worn or oversized nozzle**

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

9.9 Gloss Difference

At the same temperature, lower print speeds produce smoother surfaces. At the same speed, higher temperatures produce smoother surfaces. If uneven gloss appears on the model surface during printing, it is usually caused by differences in filament melting or cooling rate, which affect flow and lead to changes in surface roughness. This issue is more noticeable on highly reflective filaments.



Possible Causes and Solutions

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

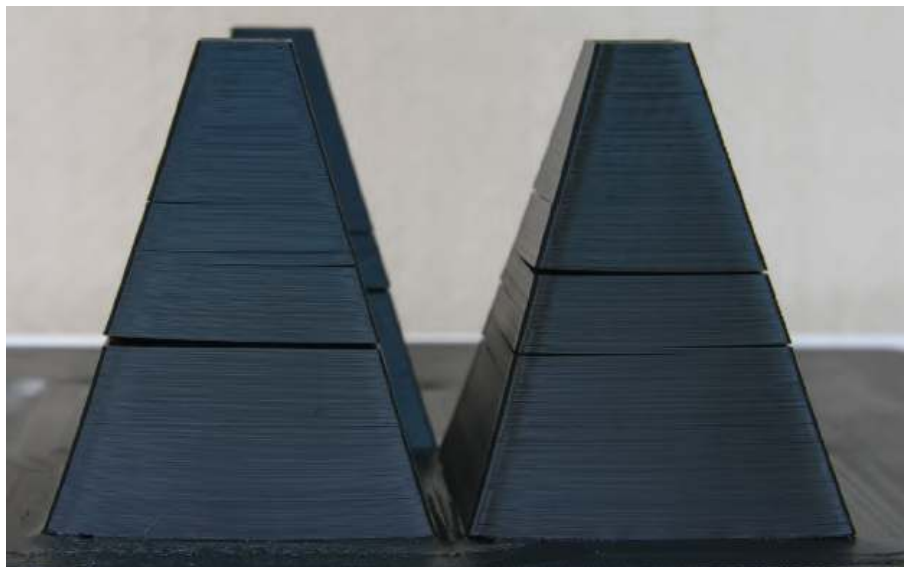
For large models, reduce the outer wall speed as needed. For small models, reduce the overall print speed. If necessary, disable Slow down for overhangs.

2. **High print speed when printing at a low layer height, such as 0.08 mm**

This may cause scale-like gloss differences on the model surface. Consider increasing the layer height, lowering print speed, or raising print temperature accordingly.

9.10 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**

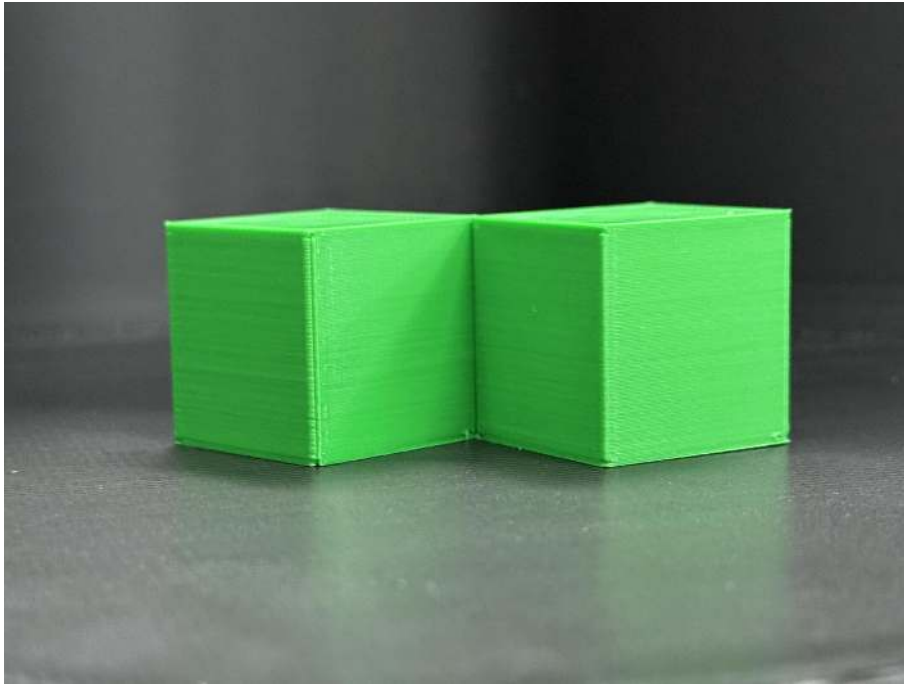
Increase the number of model wall loops or raise the sparse infill density to enhance strength.

3. **Excessive cooling**

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

9.11 Seam

In FDM 3D printing, the start and end points of each layer naturally form a seam. This is normal. You can use the following methods to reduce seam visibility and improve the model's appearance.



Optimization Methods

1. Set the wall loops to 3.
2. Print the model separately to avoid seam buildup caused by printing multiple models at the same time.
3. Increase the nozzle temperature appropriately and reduce the outer wall speed.
4. For ring-shaped or rotationally symmetric models, try enabling **Spiral vase** mode.

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 clog**
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 Printer Issues and Troubleshooting

10.1 Toolhead Clog Troubleshooting

If error messages like “extruder motor overload” or “air print” appear during printing, or if the toolhead moves normally but no filament is extruded from the nozzle (or the extruded line is too thin), this indicates a clog in the **extruder** or **hotend** inside the toolhead. Follow the steps below to troubleshoot.

Initial Check: Unload Filament

Step 1. On the printer screen Homepage, tap **Stop** to end the print job.

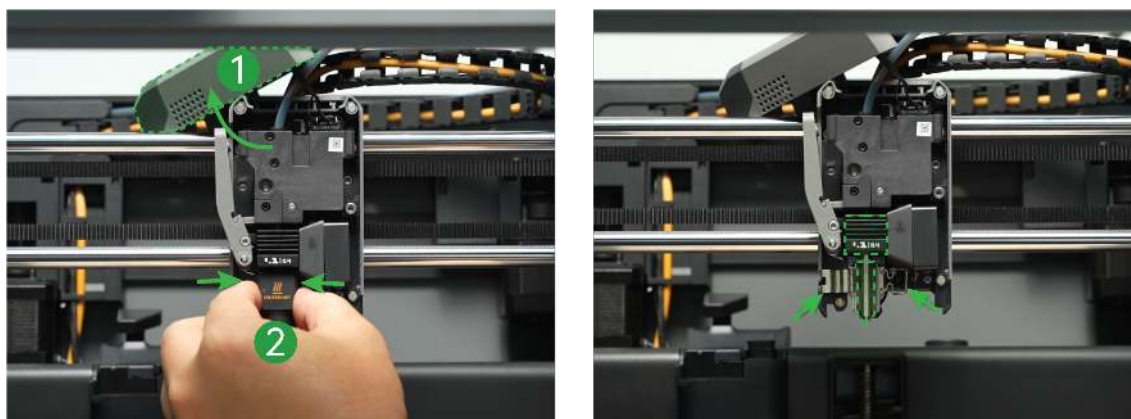
Step 2. On the  page, select the corresponding filament then tap **Unload**.

NOTE

If the unload process is abnormal, it can be determined that **the extruder is clogged (or the hotend is also clogged at the same time)**. If the process is normal, proceed to **the next troubleshooting step**.


Further Check: Manual Extrusion

Step 3. Remove the silicone cover and hotend.



Step 4. Press the black outer ring of the toolhead filament inlet and pull out the PTFE tube.



Step 5. On the printer screen, tap  > **Nozzle & Extruder** > , then manually push forward the filament.

 **NOTE**

If the extrusion process is normal, the clog is confirmed to be **in the hotend**; if extrusion fails, it indicates **<1>an extruder clog (or a clog in both the extruder and the hotend)**.

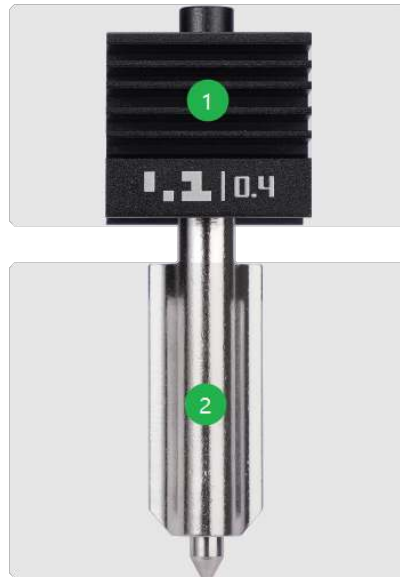
Unclogging Method

- Extruder cleaning is complex. Please visit the Bambu Lab Wiki (wiki.bambulab.com/home) and search for “P2S Extruder Clog Cleaning Guide” for detailed disassembly and cleaning instructions.
- If it's a hotend clog, see [Hotend Unclogging](#) for the next steps.

10.2 Hotend Unclogging


10.2.1 Operation Overview

Hotend clogs typically occur in the hotend cold zone (①) or the nozzle (②). They may appear as clog error messages on the printer, or as clear under-extrusion in prints, such as gaps or layer separation.



Before unclogging, identify where the clog is located:

Clog Location	Possible Causes	Unclogging Method
Hotend (inlet and cold zone)	Filament diameter too large (> 1.8 mm) or inconsistent.	Unclog with an allen key
	Filament softening or swelling caused by heat creep, such as high chamber temperature or printing with low-temperature and high-temperature filaments together.	
Nozzle	Nozzle temperature is set too low, causing the filament to melt slowly and accumulate in the nozzle.	Unclog by manual extrusion
	Filament contains particles, such as carbon fiber or glow-in-the-dark filament, or impurities are stuck inside the nozzle.	Unclog with an unclogging pin, unclog by cold pull
	High-temperature and low-temperature filaments are used together or in succession, causing the residual filament to remain unmelted and unable to extrude.	

- Step 1. On the  page, select the corresponding filament, then tap **Unload**. If an error appears, ignore it.
- Step 2. **Remove the hotend.**
- Step 3. Check if the filament at the hotend inlet is swollen or jammed.
- Step 4. If the filament is stuck at the inlet, use an allen key to clear it. If the filament is not stuck, reinstall the hotend onto the toolhead, and follow the nozzle unclogging methods listed above.

10.2.2 Unclog by Manual Extrusion

Heat the nozzle to a higher temperature and extrude filament to clear the clog by increasing filament flow.

- Step 1. On the printer screen, select **⚙ > Nozzle & Extruder**, set the nozzle temperature slightly above the recommended printing temperature, then tap **Confirm** to start heating.

i TIPS

See [Printing Conditions](#) for the recommended nozzle temperature range. For example, set to 220°C for PLA filament.

- Step 2. After the nozzle heats up, tap **▶** to extrude filament and check whether molten filament comes out of the nozzle.

⚠ CAUTION

When using TPU filament, tap the **▶** extrude button no more than 3 times, and avoid rapid and continuous tapping.



If the filament cannot be extruded normally, or if the extruded filament is too thin and brittle, see [Unclog with Unclogging Pin](#) for the next step.

10.2.3 Unclog with Unclogging Pin

Tools: heat-resistant gloves, unclogging pin (included in the toolbox).

Heat the nozzle to a higher temperature, then use a fine-tipped unclogging pin to clear the nozzle from bottom to top. This helps the molten filament inside the nozzle extrude smoothly.

i TIPS

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

⚠ DANGER

Wear protective gloves when using the unclogging pin. Keep your head away from the printer chamber to avoid contact with molten filament that may eject from the nozzle.

- Step 1. On the printer screen, select **⬆️** > **Nozzle & Extruder**, set the nozzle temperature slightly above the recommended printing temperature, then tap **Confirm** to start heating.

i TIPS

See [Printing Conditions](#) for the recommended nozzle temperature range. For example, set to 220°C for PLA filament.

- Step 2. After the nozzle heats up, insert the unclogging pin and move it up and down several times.



After that, tap **<** to extrude filament. If the filament does not extrude normally, or if the extruded filament is too thin or breaks easily, see [Unclog by Cold Pull](#) for the next step.

10.2.4 Unclog by Cold Pull

Cold Pull is a common nozzle maintenance method. During the process, the nozzle is heated to soften the filament, then the filament is pulled out after it cools and partially solidifies. Repeat the cold pull several times to better remove clogs and residue inside the nozzle.

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

- 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. Manually pull out the filament. Repeat the process based on actual needs.
- Step 5. After the cold pull is complete, reinstall the PTFE tube on the toolhead.



10.2.5 Unclog with Allen Key

Tools: heat-resistant gloves, allen key H1.5, pliers, lighter.

If a swollen filament is stuck inside the hotend and cannot be extruded or pulled out, insert a heated allen key into the filament to soften it and let it wrap around the key, then pull them out together.

⚠ CAUTION

This method only applies to allen key H1.5.

⚠ DANGER

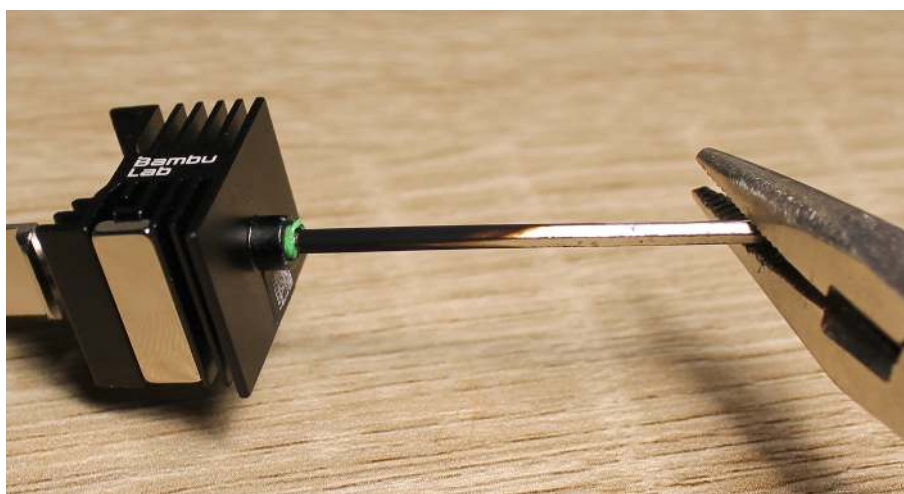
Always wear protective gloves while operating to prevent burns from dripping molten filament.

Step 1. **Remove the hotend.**

Step 2. Hold the allen key with pliers and heat its end for about 10 seconds. Be careful to avoid burns.



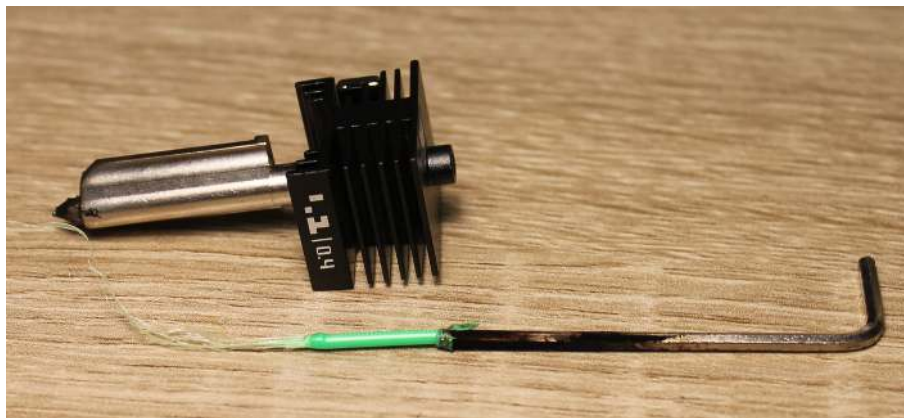
Step 3. Quickly insert the allen key into the hotend and ensure it is fully covered by the filament. Push forward until the allen key passes through the heatsink (as shown below), then wait about 30 seconds to cool.



Step 4. Heat the nozzle tip with a lighter for about 20 seconds. Slowly pull out the allen key to remove the filament and clog from inside the hotend.

⚠ DANGER

- Do not overheat the nozzle or use high-power butane torches. A regular gas lighter is recommended.
- The nozzle only needs to be hot enough to pull out the filament and allen key. Overheating the nozzle may cause the filament to drip or burst. Always keep the nozzle tip pointed away from yourself.



Step 5. Once finished, reinstall the hotend.

10.3 Loose Belt

Tool: allen key H2.0.

After long-term use, the printer belt, also known as the timing belt, may wear or loosen, causing print quality issues, such as circles printing as ovals, or HMS error messages. In this case, maintain and tension the belt.

⚠ CAUTION

Make sure the printer is powered off during maintenance.

10.3.1 XY Belt Tension

📘 TIPS

After completing all maintenance steps, run vibration compensation calibration.

Step 1. Loosen the **4** tension screws by 1 – 2 turns.



Step 2. Move the toolhead along the printer frame for 3 – 5 rotations, then position it at the **middle rear of the printer** (green circle).



Step 3. Retighten the **4** screws.

⚠ CAUTION

Do not overtighten the screws, as this may damage the threads.

10.3.2 Z Belt Tension

⚠ CAUTION

- Make sure the printer is powered off during maintenance.
- The printer is heavy. Move it with two people if possible to avoid injury or printer damage.

i TIPS

After completing all maintenance steps, run vibration compensation calibration.

Step 1. Remove the top cover and lay the printer flat.



Step 2. Loosen the tensioning screw by 1 – 2 turns.



Step 3. Pull the Z belt through 3 – 5 cycles. Ensure the movement is smooth.



Step 4. Retighten the screw.

Step 5. After completing these steps, place the printer on a stable, level surface and replace the cover.

Chapter 11 Regular Care and Maintenance

DANGER

- Before maintenance or disassembly, power off the printer and disconnect it from the power supply to avoid safety hazards.
- Before maintenance or replacement, ensure that the hotend and heatbed have cooled to room temperature. If you must work while they're hot, wear heat-resistant gloves to protect yourself.

11.1 Regular Maintenance

11.1.1 Maintenance Frequency

The printer can assess its cleanliness based on task type and print duration, and provide maintenance reminders. After receiving the printer, update the firmware to the latest version to enable this feature.


CAUTION

If you print high-temperature or engineering filaments continuously for long periods, or if the printer runs for more than 8 hours per day on average, increase the maintenance frequency:

- Parts maintained every three months should be **maintained monthly**.
- Parts maintained monthly should be **maintained weekly**.

DANGER

Clean and maintain all printer parts regularly as instructed. Failure to do so may damage the printer and create safety risks.

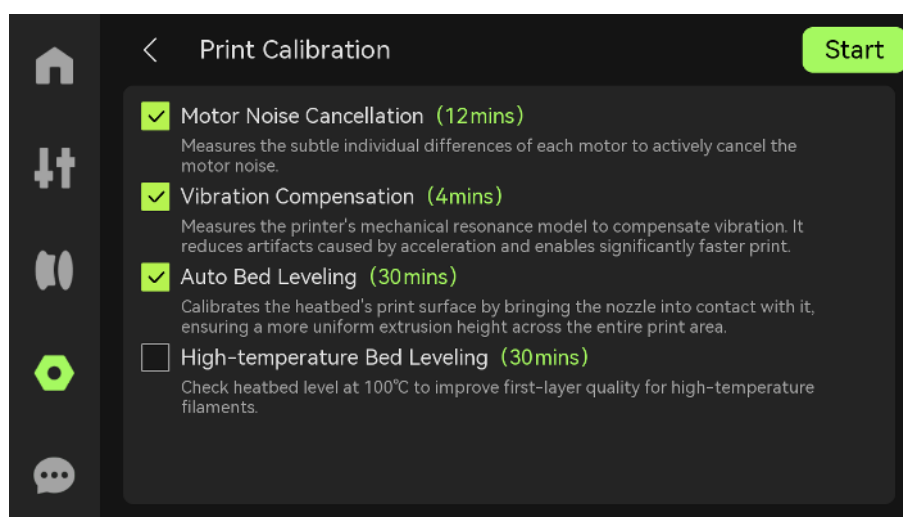
Component		Type	Frequency	Operating Steps
Printer	Printer system	Calibration	After transport or maintenance	On the printer screen, select  > Calibration > Print Calibration .
Chamber	Live view camera	Clean	1 month	Wipe with a cotton swab or non-woven cloth dampened with alcohol
	Build plate	Clean	1 week	Wash the surface with warm water and dish detergent, then wipe it dry.
	Air filter	Clean	3 months	Wipe the filter cover with a non-woven cloth dampened with alcohol, and replace the activated carbon filter regularly.
XYZ Axes	X-Axis	Clean	1 month	Wipe with a non-woven cloth dampened with alcohol

	Y-Axis	Clean and lubricate	3 months	Wipe with a non-woven cloth dampened with alcohol, then apply lubricant grease to the lead screws and lubricant oil to the linear rods after cleaning.
	Z-Axis			
Toolhead	Extruder	Clean and lubricate	1 month	Remove internal debris and apply lubricant grease to the gears.
	Hotend	Clean	1 month	Cold pull


11.1.2 Print Calibration

To ensure printing accuracy and stable operation, perform a full calibration after moving the printer or maintaining key components such as the XYZ axes, belts, and toolhead.

On the printer screen, tap  > **Calibration** > **Print Calibration**, then select the calibration task as needed.



You can also start calibration tasks via Bambu Handy or Bambu Studio. Follow the steps below:

- Bambu Studio: Go to the **Device** page, select **Status**. Click **Calibration**, choose the calibration task as needed, then click **Start Calibration**.
- Bambu Handy: Go to the **Devices** page, select  > **Calibration**, choose the calibration task as needed, then tap **Start Calibration**.

11.1.3 Air Filter

Tools: non-woven cloth, alcohol, dust brush, and a new activated carbon filter.

Regularly clean dust from the air filter cover. If the filter surface is visibly dirty or saturated, replace the activated carbon filter.

Step 1. On the printer screen, tap  > **Motion** >  to lower the heatbed to the bottom.

Step 2. Remove the filter cover from the side clips. Use a dust brush to clean the surface, then wipe it with a non-woven cloth dampened with alcohol.

⚠ CAUTION

If the cover is heavily soiled, you can rinse it directly with water. Make sure it is completely dry before reinstalling it, otherwise other electronic components may be affected.

- Step 3. Hold the filter handles and pull outward to remove the old filter and replace it with a new one.



11.1.4 XYZ Axes

⚠ CAUTION

Make sure the printer is powered off during maintenance.

Clean and Lubricate the X-Axis

Tools: non-woven cloth, alcohol, lubricant oil.

- Step 1. Use a non-woven cloth dampened with alcohol to wipe the X-axis linear rods and belts.
- Step 2. Manually move the toolhead to clean the remaining areas.
- Step 3. Apply lubricant oil dropwise onto the linear rods. Manually move the toolhead along the X-axis **slowly** 3 – 5 times to ensure even distribution.



Clean and Lubricate the Y-Axis

Tools: non-woven cloth, alcohol, lubricant oil.

- Step 1. Use a non-woven cloth dampened with alcohol to wipe the Y-axis linear rods on both sides.
- Step 2. Manually move the toolhead to clean the remaining areas.
- Step 3. Apply lubricant oil dropwise onto the linear rods. Manually move the toolhead along the Y-Axis 3 – 5 times to ensure even distribution.



Clean and Lubricate the Z-Axis

Tools: non-woven cloth, alcohol, lubricant oil.

The Z-Axis includes three sets of lead screws and linear rods: one at the left front, one at the right front, and one at the rear center. The front sets include both a lead screw and a linear rod, while the rear set includes only a lead screw. During cleaning, raise the heatbed from the printer screen to access the lower sections.

⚠ CAUTION

1. When wiping or lubricating the Z-Axis, make sure the printer is powered off. Only power on the printer when raising the heatbed to clean the lower sections.
2. Linear rods and lead screws use different lubricants. Do not mix them up.

- Step 1. Use a non-woven cloth dampened with alcohol to wipe the lead screws and linear rods on both sides and the rear.
- Step 2. Apply lubricant oil to the linear rods (dashed lines) and the bottom bearings of the lead screws. Apply lubricant grease to the main bodies of the lead screws (solid lines).



Step 3. Connect the power. On the printer screen, tap **⬆️** > **Motion**, then move the heatbed up and down 3 – 5 times to spread the lubricant evenly.

11.1.5 Heatbed

Tools: non-woven cloth, alcohol.

- Step 1. On the printer screen, tap **⬆️** > **Motion** > **⬆️** to raise the heatbed. Clean any leftover filament underneath.
- Step 2. Spray alcohol evenly on the bottom of the chamber, then wipe clean with a non-woven cloth.
- Step 3. On the printer screen, lower the heatbed and remove the build plate. Use a non-woven cloth dampened with alcohol to wipe the heatbed surface.
- Step 4. After the alcohol has fully evaporated, place the build plate back properly.

⚠️ CAUTION

Before placing the build plate, check and clean the heatbed surface to ensure no debris remains.

11.1.6 Build Plate

Tools: dish detergent, sponge (or brush).

Wash the build plate with warm water and dish detergent. Dry it completely before attaching it to the heatbed for use.



11.1.7 Toolhead

Tools: pointed tweezers, H2.0 and H1.5 allen keys, lubricant grease.

The gear assembly inside the extruder may wear after prolonged use, causing printing issues. Regularly apply lubricant to ensure smooth printing.

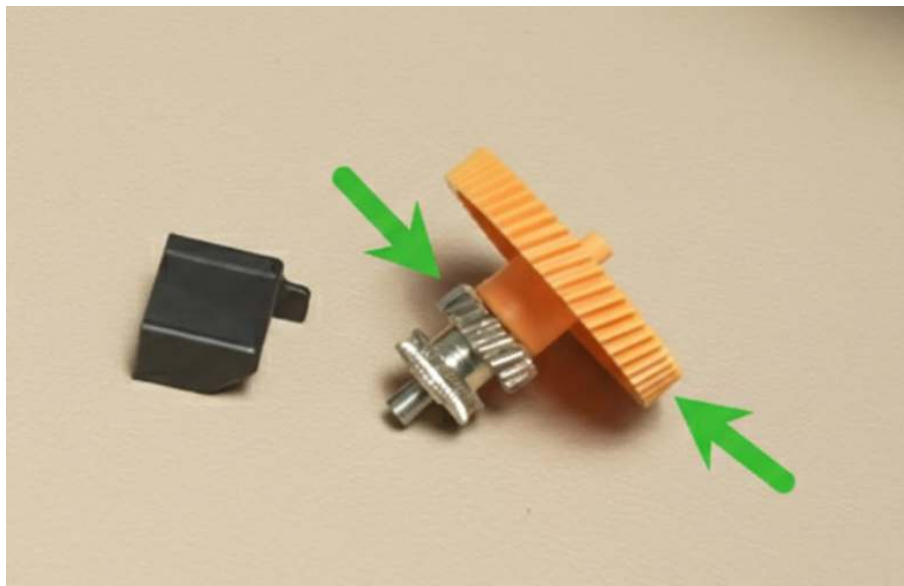
NOTE

To maintain the extruder, visit Bambu Lab Wiki (wiki.bambulab.com/home) and search for "Extruder Components Replacement Guide for the P2S " for detailed disassembly and assembly instructions.

CAUTION

Place the extruder on a clean surface during disassembly to prevent small parts from falling.

- Step 1. Remove the extruder, detach the Hall sensor assembly, then separate the extruder gear components.
- Step 2. Apply lubricant grease to the 2 gear locations shown in the illustration.

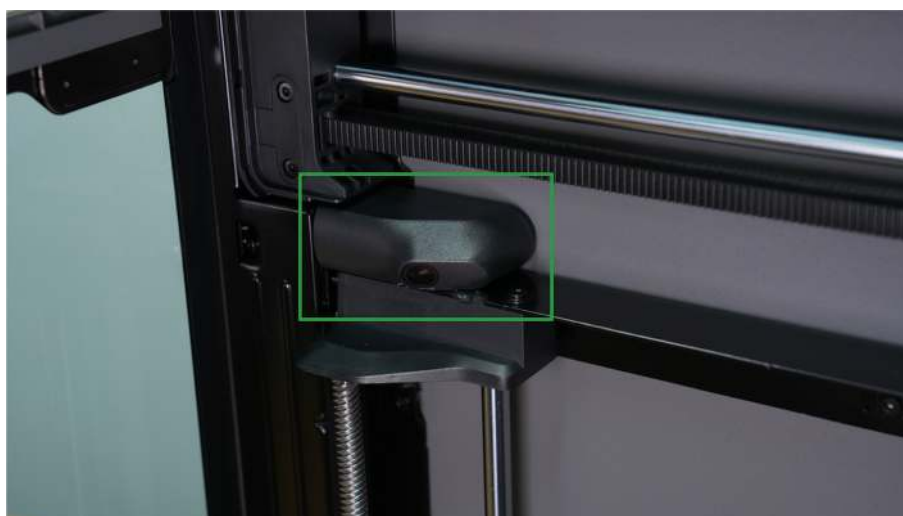


11.1.8 Live View Camera

Tools: cotton swab (or non-woven cloth), alcohol

After prolonged use, volatile particles may build up on the surface of the live view camera. This causes blurry video and reduces visual detection sensitivity.

Dip a cotton swab in a small amount of alcohol and wipe the camera surface.



11.2 Replace Accessories

Over time, wear or aging of certain printer accessories can affect overall printing performance. Regularly check the condition of accessories and replace them as needed. This chapter explains the replacement requirements and steps for common consumable parts. You can also visit Bambu Lab Wiki (wiki.bambulab.com/p2s) and see "**Part Replacement Guide**" for more tutorials.

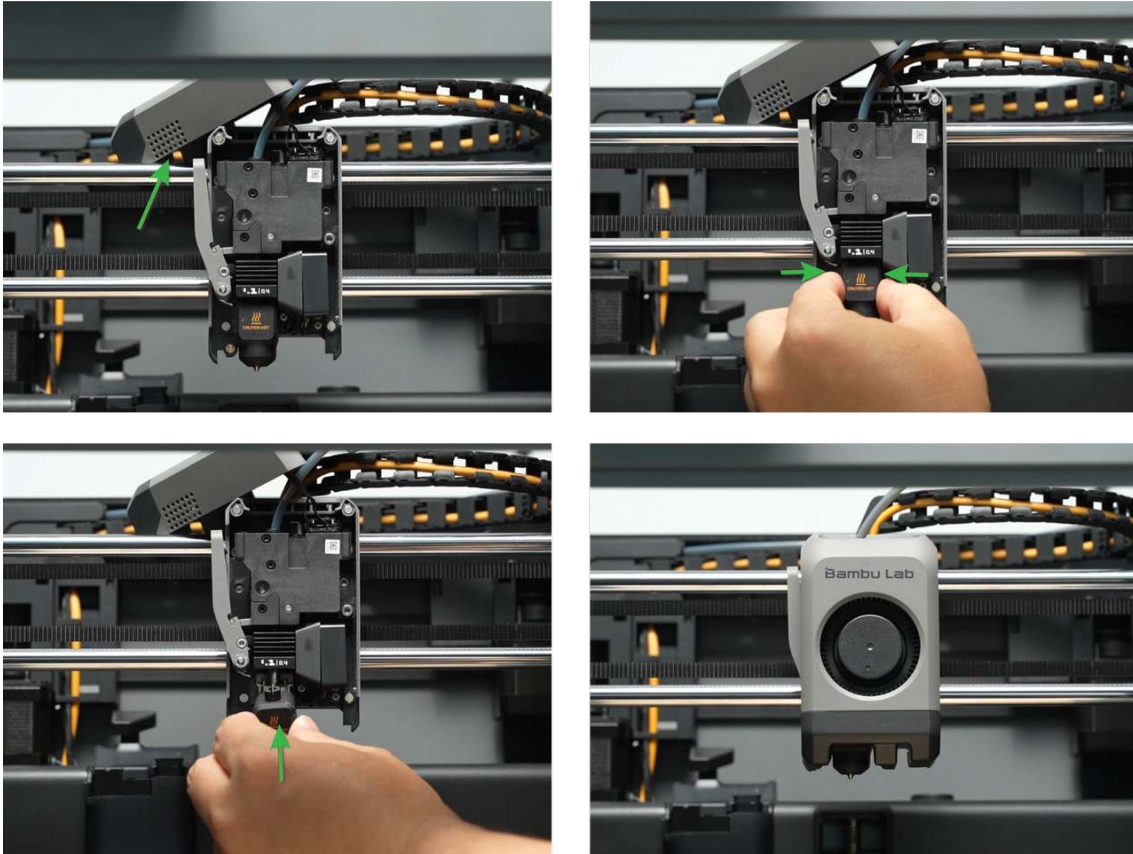
11.2.1 Hotend Silicone Sock

Tools and Accessories: new hotend silicone sock (included in the toolbox).

The silicone sock is installed around the hotend nozzle to maintain a stable nozzle temperature. Replace it if it is damaged or no longer stays securely in place.

⚠ DANGER

Before replacement, ensure the nozzle temperature has cooled down to room temperature to prevent burns.



- Step 1. Remove the toolhead front cover and hang it aside.
- Step 2. Pinch the top of the silicone sock and pull it down diagonally to remove.
- Step 3. Install the replacement silicone sock onto the hotend from bottom to top.
- Step 4. Gently pull on the silicone sock to ensure it fits properly, then snap the toolhead front cover back into place.

11.2.2 Hotend


Tools and accessories: heat-resistant gloves and a new hotend.

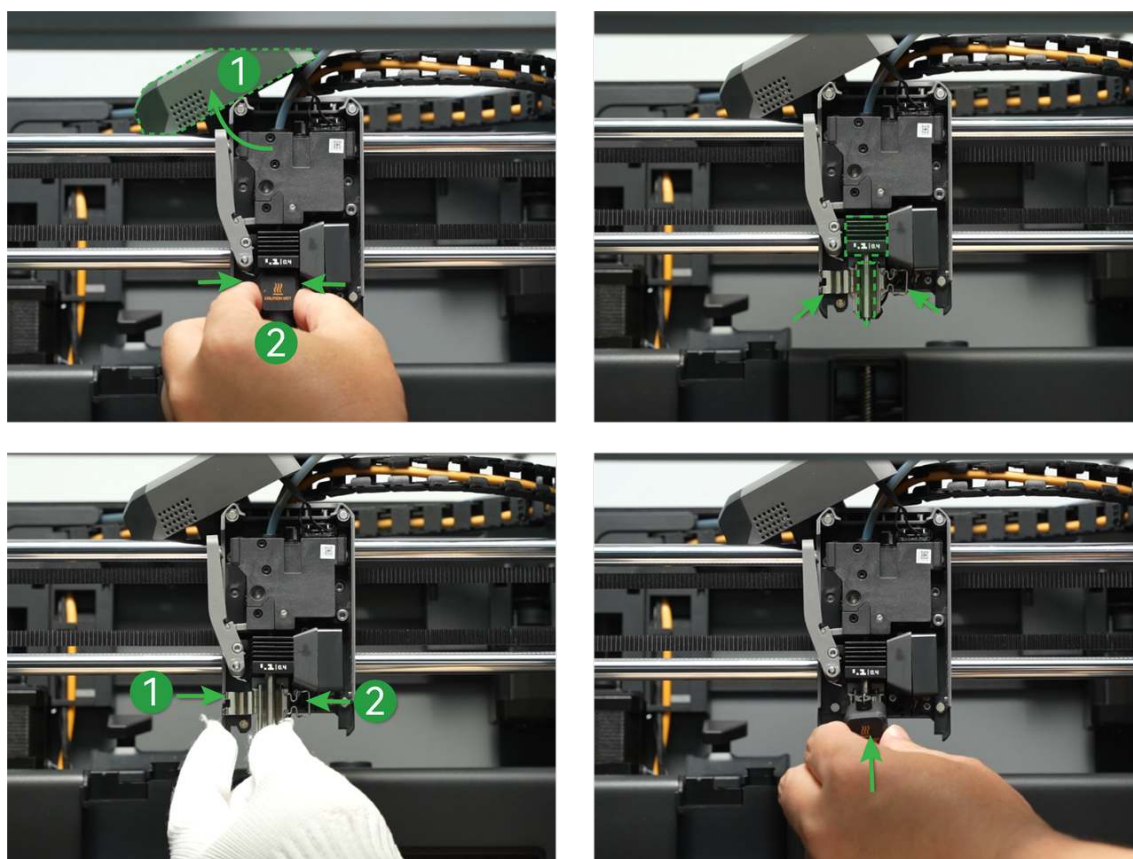
Replace the hotend when it is clogged or damaged and standard unclogging steps (see [Hotend Unclogging](#)) can no longer resolve the issue.

⚠ DANGER

Before replacement, make sure the nozzle has cooled to room temperature to avoid burns. If you must work while it is still hot, wear heat-resistant gloves for safety.


TIPS

Before replacement, go to the filament page on the printer screen , select the corresponding filament spool icon, and tap **Unload**.



- Step 1. Remove the toolhead front cover and **the hotend silicone sock**.
- Step 2. Unlock the locking tabs on both sides, **right first then left** and remove the hotend.
- Step 3. Insert the replacement hotend. Secure the hotend locking tabs, **left first and then right**, ensuring they fit snugly.
- Step 4. Reinstall the hotend silicone sock and the toolhead front cover.

NOTE

If you replaced the hotend with a different material or diameter, tap  > **Nozzle & Extruder** on the printer screen and select the correct nozzle type.

11.2.3 Filament Cutter Blade

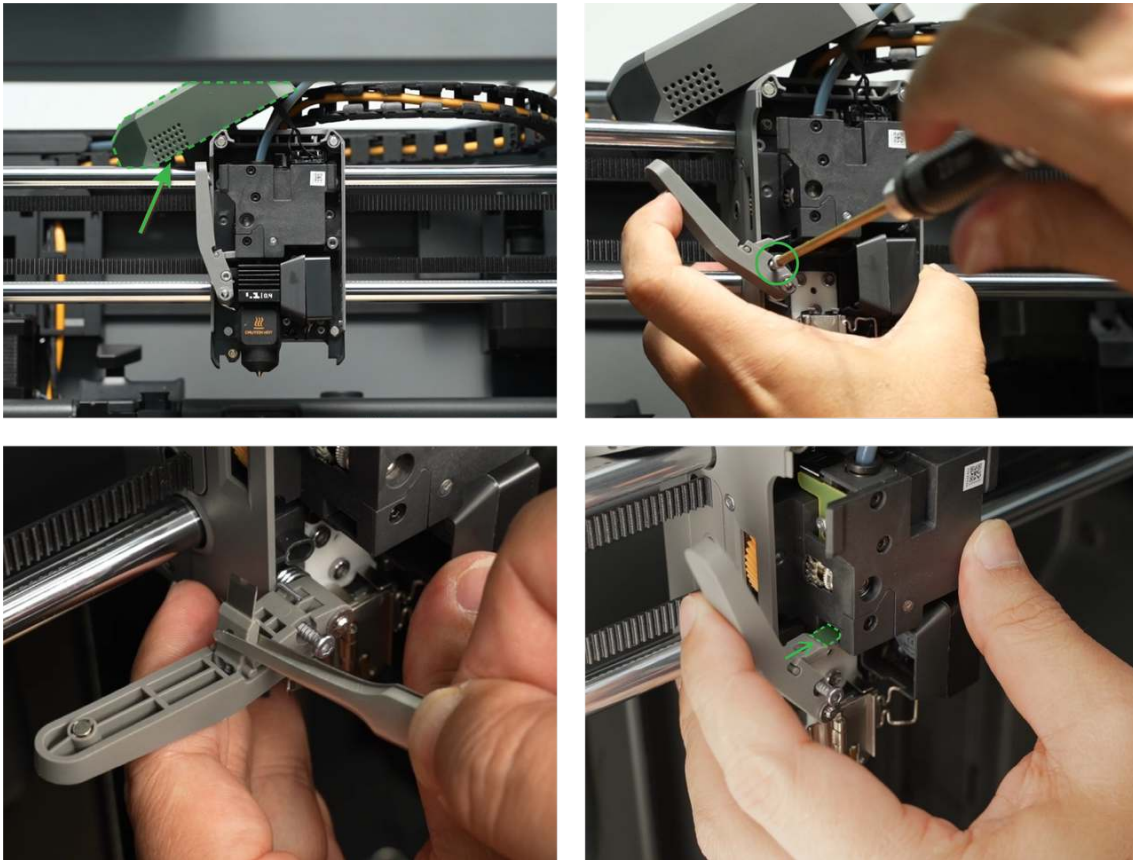
Tools and accessories: new filament cutter blade (included in the toolbox), allen key H2.0, and needle-nose pliers or tweezers.

The filament cutter is installed on the side of the toolhead. It consists of the filament cutter lever, blade, and other parts. During filament unloading and changing, the filament cutter lever is driven to press the filament cutter stopper and cut the filament.

Regularly check the blade condition and replace it if worn or damaged.

⚠ DANGER

The blade edge is sharp and may cause cuts. Handle with care.



- Step 1. Remove the toolhead front cover.
- Step 2. Hold the filament cutter lever and loosen the screw below to let the lever hang naturally.
- Step 3. Remove the old blade and insert a new one.
- Step 4. Hold the lever, insert the blade into the slot on the side of the extruder, and tighten the screw.
- Step 5. Snap the toolhead front cover back into place.

11.2.4 Nozzle Wiping Pad

Tools and accessories: new nozzle wiping pad (included in accessory box).

The nozzle wiping pad is installed on the purge wiper and is used to remove residual waste material from the nozzle surface. Replace it when it is damaged or dislodged.

i TIPS

The nozzle wiping pad rarely comes loose. However, if the toolhead follows an abnormal path, such as due to a G-code error or firmware fault, the pad may be scraped off. Follow the steps below to reinstall it for normal use.



- Step 1. Remove the old nozzle wiping pad from the purge wiper.
- Step 2. Insert the spare nozzle wiping pad. Align the nozzle wiping pad with the groove on the purge wiper, then press down until it is installed securely.

Enjoy!

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



bambulab.com/support/documentation