PRUSA RESEARCH
by JOSEF PRUSA

ENG

ORIGINAL INSTRUCTIONS

3D PRINTING HANDBOOK

FOR THE PRUSA PRO HT90 3D PRINTER

To download the handbook in other languages and see a complete overview of help articles, visit help.prusa3d.com/drivers or scan the QR at the bottom of this page.

Prusa PRO HT90 Official Handbook v1.00 - June 15th, 2024

Safety Information

Before any operation or handling of the Prusa Pro HT90 printer, please thoroughly read this manual. If you have any questions or concerns, contact technical support. Follow the safety instructions provided in this guide.

Orange boxes indicate an important notice, tip, hint, or information for easier printing.

This section of text is very important, please read it carefully! It is directly related to the proper operation of the printer and its safe use.

Technical support

Various situations are described in this handbook and more extensively at prusa.io/ht90. Scan the QR code on the right to visit the section of our website. In case your problem is not listed, you can reach our tech support via Live Chat on our website or a dedicated e-mail address ht90@prusa3d.com.

1. About Prusa PRO and Prusa Research

Prusa Pro is the professional division of Prusa Research, a 3D printing manufacturer based in Prague, Czech Republic. Prusa Pro develops and manufactures industrial-grade 3D printers, accessories, and software designed for a wide range of demanding applications.

Prusa Research, founded by Josef Prusa in 2012, is a renowned Czech company specializing in 3D printing technology. The company grew from a small one-man startup project into one of the most important companies in the additive manufacturing industry, garnering a global reputation for its high-quality, reliable, and user-friendly 3D printers. The core philosophy of Prusa Research revolves around open-source principles, innovation, and community engagement, which have collectively driven its rapid growth and success.

Table of contents

2. Prusa Pro HT90: Product Information

Product name:	Prusa Pro HT90
Manufacturer:	Prusa Research a.s., Partyzánská 188/7a, Holešovice, 170 00, Prague, Czech republic
Manufacturer contacts:	Phone: +420 222 263 718, e-mail: info@prusa3d.com
EEE category:	3 (IT / telecommunications equipment)
Power supply:	220-240 VAC, 10 A max., 50-60 Hz
Operating temperature range	20 °C - 32 °C
Maximal air humidity:	85 %, non-condensing
Printer dimensions:	Width: 530 mm (without the filament holder)
	Depth: 530 mm
	Height: 1050 mm (1470 mm with the door opened)
Maximal print dimensions:	Ø 300 mm (X, Y) × 400 mm (Z)
Preinstalled nozzle diameter:	0,4 mm (High Flow), 0,6 mm (High Temp)
Filament diameter:	1,75 mm
Weight (incl. packaging / without packaging):	50 kg / 43,5 kg

The product's serial number is located on a type plate placed next to the power connector.

This device is meant exclusively for indoor use, where it is protected from extreme conditions. The device is also meant to be operated in an industrial environment; it may cause interference in a residential environment.

3. Prusa Pro HT90: Warranty information

The Prusa Pro HT90 3D printer is covered by a 24-month warranty for end customers in the EU, a 12-month warranty for end customers in the rest of the world and a 12-month warranty for all business customers. Consumable materials and parts subject to wear and tear are excluded from this warranty. The warranty period begins on the day the customer receives the goods. Failure to follow official instructions during installation, upgrade, operation and configuration may result in loss of warranty.

The seller is not liable for damages caused by the improper handling of purchased products or for damages caused by handling not in accordance with the information and recommendations provided in the official manuals and handbooks. The warranty is also void in the event of non-professional tampering or use of unofficial modifications to the hardware and software.

Safety Symbols

Caution: strong magnetic fields. Do not place objects sensitive to magnetic fields on the marked areas, as they could be irreversibly damaged.

Attention: approaching mechanical parts. Take care not to injure your hands if you are in close proximity to equipment with mechanical parts.

Information on function, operation and service can be found in this manual or online at help.prusa3d.com.

Before any servicing, the product must first be disconnected from the power supply and the instructions read.

Take extra care when handling or touching parts marked with this symbol and avoid other hazards listed for specific symbols, such as hot surface hazards - burns may occur.

Warning: hot surface! An object marked in this way may be hot and extra care must be taken when touching it to avoid burns.

Attention: moving parts. Unprotected moving mechanical parts can cause injury, take extra care.

Do not print directly on this surface.

Warning: risk of electric shock. Never tamper with the parts of the product marked as such.

This product comprises components that must be disposed of per the Waste Electrical and Electronic Equipment Directive, so take it to an e-waste collection point.

4. Safety Information

Please, read the following information thoroughly. Ensure that anyone who operates the Prusa PRO HT90 knows these information.

4.1. Electrical safety:

- The printer can only be powered through a standard 230 VAC, 50 Hz power outlet. Never use alternative power cords, they may cause problems or even damage the printer.
- Do not use the printer if the power cord is damaged in any way - damaged cords can lead to electric shock.
- When unplugging the power cord from the power outlet, do not pull on the cord but on the plug, this will reduce the risk of damaging the plug or the power outlet
- Never disassemble the printer, it contains no parts that could be repaired by untrained personnel. Always have the printer repaired by a qualified service technician.
- Improper tampering with the printer can result in damage to the printer and an increased risk of electrical shock.
- Disconnect the device from the mains by pulling the plug. The electrical outlet must be easily accessible.
- The printer is equipped with a replaceable fuse, which is located in the fuse holder near the power supply connector and protects the entire printer. Before replacing the fuse, turn off the printer and disconnect the power supply by removing the power cord from the power outlet. Use a flat-blade screwdriver to unscrew the fuse holder, remove the fuse, and insert the new fuse. Screw the fuse holder back into place. Always make sure that the new fuse has the same value as the label. If the fuse repeatedly blows, contact technical support.

4.2. Mechanical risk

- Moving mechanical parts of the printer can cause injury. Never tamper with the internal parts of the printer while it is connected to power or in operation - there is a risk of injury from mechanical parts or electric shock.
- Prevent unauthorized persons from handling the printer - even when the printer is not printing.

4.3. Burn Risk

- Do not touch heated parts of the printer - heated beds, print sheets, and parts of the print head. There is a risk of burns.
- Warning! During printing, parts of the print head and print bed can heat up to very high temperatures! Do not touch them until printing is finished and the printer has cooled down - burns may occur.

4.4. Safety Instructions and Disclaimer

- Acting contrary to the information provided in the manual may lead to injury, poor print results, or damage to the 3D printer. Ensure everyone working with the printer is familiar with the manual and understands the instructions.
- Since we cannot control the conditions under which the 3D printer is used, we do not accept any liability and explicitly disclaim any responsibility for losses, injuries, damages, or expenses arising from or related to the assembly, handling, storage, use, or disposal of the product.
- Information is provided without any warranties, express or implied. Be cautious when handling the 3D printer. It is an electrical device with moving and heated parts.

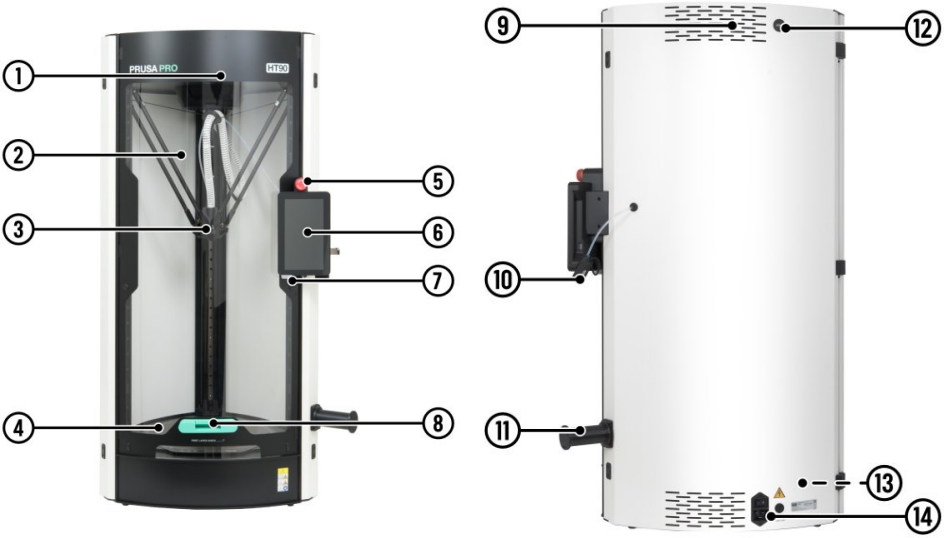
4.5. Placement and Basic Use

- Ensure the printer is placed and operated in a suitable location to avoid potential risks. The surface must be level and stable.
- The device is intended for indoor use only. Do not expose the device to water or snow. Contact with water and other liquids can lead to electronic damage, short circuits, and other types of damage. Always operate the printer in a dry environment.
- The device is designed for industrial environments; it may cause interference in residential environments. Ensure that none of the vents/fans are blocked. Insufficient cooling can lead to overheating and serious damage to the printer (risk of electronic damage, fire).
- Make sure the printer is positioned to prevent it from tipping over or falling to the ground. If the printer has suffered physical damage, do not use it - damaged parts of the printer can pose a safety risk.
- Power and other cables must be placed so they cannot be tripped over, stepped on, or otherwise damaged. Ensure the power cable is not damaged. If it is, immediately stop using the device and replace the cable. Damaged cables pose a safety risk - there is a risk of electric shock.
- Do not leave the printer unattended! The printer is equipped with temperature control and a range of intelligent safety features, but using the device contrary to the instructions in this manual or in the event of an unexpected component failure can pose a fire risk.
- Never interfere with the print area when the printer is in operation. Also, prevent foreign objects from entering the print area - this could cause a collision.

5. Prusa Pro HT90 Package Contents

- IEC 320 C19 Power Cable (plug is automatically selected based on the country of destination)
- Wi-Fi adapter - manufacturer: TP-Link, type: TL-WN821N
- USB flash drive
- A spool of Prusament filament
- Textured Printpad
- Alcohol-soaked pad
- Side cutters PLATE170
- High-temperature print head
- Filament holder

6. Prusa Pro HT90: Parts Description



1. Surveillance camera (inside the chamber)
2. Sliding door
3. Print head
4. Heated bed with a print sheet
5. Power on/off button
6. Touch control display with USB port (for USB drive with print projects)
7. Status indicator light
8. Door handle with lock - press to unlock
9. HEPA filter
10. Filament input and sensor
11. Spoolholder
12. USB port for Wi-Fi module
13. USB port for Wi-Fi module and LAN connector
14. Power connector and power switch

7. Unpacking and Setting Up the Printer

The printer must be placed in a location with sufficient load-bearing capacity (at least 50 kg) and space around it, especially upwards, where the doors slide open. The bottom of the printer has cutouts for transport straps.

For transport, the printer is packed in a cardboard box with pull-out handles. Remove the handle lid (4x), pull the handles out, lift the box upwards and remove the safety padding. Now, carry the printer to a location with sufficient load-bearing capacity.

We recommend keeping the transport packaging for easier and safer handling during any transport.

8. Preparing for Operation

After placing the printer on a suitable surface, it needs to be prepared for operation. First, connect the printer to the power supply. Before plugging in, ensure the main switch is in the off position (O).

8.1. Power Supply

Use the included power cable and connect it to the power connector as shown in the photos below. Note that the power cable has a security latch.

Now plug the cable into an electrical outlet.

8.2. Spoolholder

Next, prepare the adjustable spoolholder. It can only be inserted in one correct direction, as shown below - the narrow part goes into the gap first.

To adjust the required width for a wider spool, turn the end of the holder. Turning counterclockwise increases the width. Turning clockwise tightens (shortens) the holder.

After adjusting the width, the spool holder is ready. Place a spool of filament on the holder.

If you have the official DryBox accessory, do not place the spool on the spoolholder yet. The DryBox setup is described in the following chapters.

8.3. Network Connection

The last step before turning on the printer is connecting to the internet. It is not necessary, **the printer can operate in a fully offline mode,** however, without a network connection, you cannot utilize all the advantages.

8.3.1. Wired Connection

The recommended method is connecting via an Ethernet cable (RJ45 connector). Locate the port on the back of the printer and plug in the cable. If you have a standard DHCP-enabled network, connection setup is needed for this option.

8.3.2. Wireless Connection

If you cannot use a cable, you will find a Wi-Fi adapter in the package. Insert it into one of the two USB ports on the back of the printer.

Do not insert the Wi-Fi module into the USB port intended for the flash drive with print data on the side of the display!

To connect, you need to enter login details in the Settings. More details are provided in the Network Connection chapter.

This concludes the initial setup. The printer is now fully prepared for the first start.

Please keep in mind that we are not able to control the conditions in which your printer connects to your network. The HT90 is equipped to handle all standard network conditions, however, if your corporate network requires a specific configuration, please contact your IT administrator for specific instructions.

The quality of the wireless connection may vary depending on your wireless router's performance. If you encounter issues with a poor signal and frequent disconnecting, consider using a standard ethernet (RJ45) cable to connect the printer.

8.4. Power on the Printer

Flip the switch on the back of the printer to start the system.

This concludes the initial setup. Now it's time for the first print.

9. Your First Print

To use the printer correctly, pay attention to the information in the following chapters.

9.1. Basic Operation

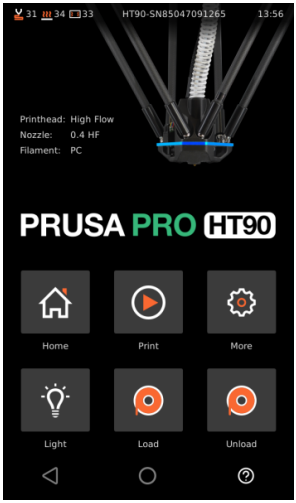
Emergency Stop

If needed, there is **an Emergency Stop** right above the display that will immediately stop all printer activities.

Pressing it will display an information screen (see below). After the emergency is over, turn the physical button in the direction of the arrows (i.e., clockwise) to release it.

9.1.1. Touch Display

After turning on the 3D printer, the **Home Screen** appears with basic instructions and an overview of the printer's status.



At the top of the Home Screen is a menu bar that contains the following information (from left to right):

- Nozzle temperature
- Heated bed temperature
- Chamber temperature
- Serial number
- Current time (obtained from the network if available)

Right below the status bar are details about the installed print head (loaded automatically after installation). Information includes nozzle type, diameter, and the last used material. Information is color-coded for a quick overview. A blue stripe indicates a High Flow head (up to 300°C), and red indicates a High-Temperature head (500°C, standard flow).

At the bottom of the home screen are basic control instructions for the 3D printer - Home, Print, More, Light, Load, and Unload.

Each menu option will be explained in the following chapters.

9.2. Status LED

At the bottom of the display, there is an indicator light for quickly checking the printer's status. You can encounter 4 different colors:

- White indicates the printer is in Idle mode.
- Green indicates the printer is performing an operation other than a print job.
- Blue is for a print job.
- Red indicates an error has been detected.

9.3. Sliding Door

The Prusa Pro HT90 printer has a sliding door and a safety sensor. Use the green handle to operate it.

First, **press the handle,** then pull the door to the desired height.

Now you can release the handle; the door will stay in place thanks to the spring system.

At the bottom of the door is a pressure sensor. An error message will appear on the display if the door is not properly closed. If needed, the door sensor can be completely disabled. This option is

available in the printer Settings, but unless you have a very specific reason, we recommend not disabling the sensor.

The door can only be raised to half the height of the printer. There are stops in the sliding mechanism.

9.4. Print Sheets

The Prusa Pro HT90 can use two types of print sheets - standard (for printing all materials) and special (for printing filaments based on polyamides, such as PA and Nylon).

When you unpack the printer for the first time, the print sheet should be already installed on the heated bed. However, it might have moved during transportation and you will also need to remove it regularly - e.g., when you finish a print or when you need to clean the sheet.

Before installing the print sheet, ensure no debris (plastic scraps, filament pieces, etc.) is on the heated bed. It could affect the calibration of the first layer or damage the bed's surface. Do not drag or move the print sheet on the heated bed, as this could scratch and damage it. If you need to align the print sheet, lift it by the front part, adjust its position, and place it back.

Never print directly on the heated bed! See the picture below - this is how the printer looks WITHOUT the print sheet.

The flexible print sheet must be placed on the heatbed before you start any calibration or print!

9.4.1. Placing the Print Sheets

The print sheet contains a special surface that makes removing printed objects fast and easy. It's important to keep it grease-free (try not to touch the print area with your fingers).

Place the print sheet on the bed by first aligning the rear cutout with the locking screws on the back of the heated bed.

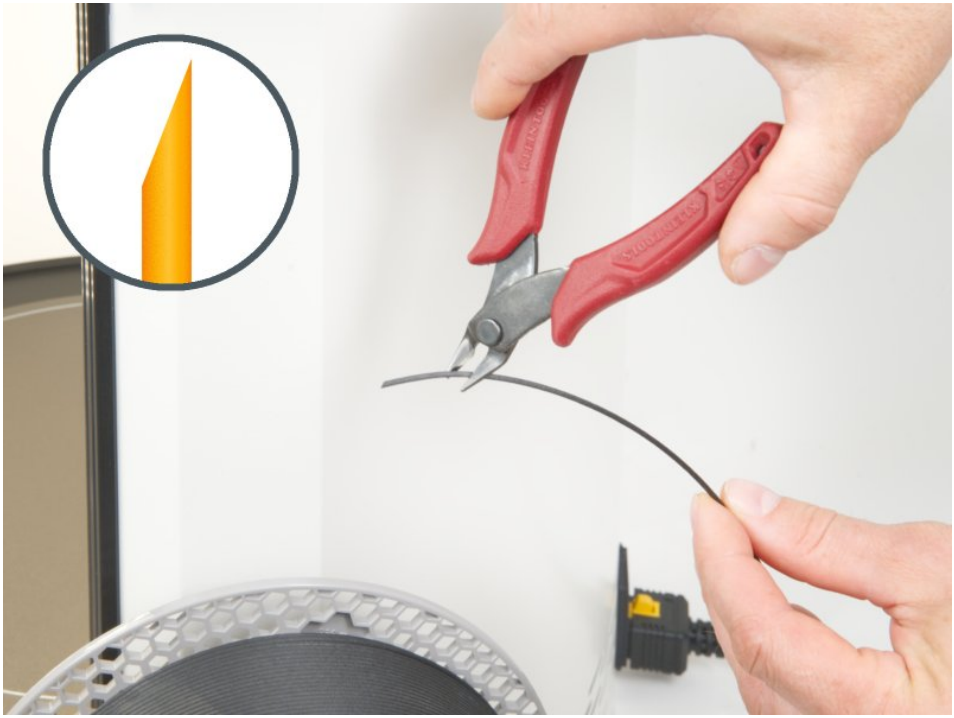
Hold the sheet by the front part and slowly place it on the heated bed. Be careful and watch your fingers.

For the first print, wipe the print sheet with the supplied cleaning wipes soaked in isopropyl alcohol. You can do this immediately, but be careful not to touch the surface afterward.

9.5. Loading Filament

First, prepare a filament spool - we strongly recommend using the ASA filament included in the printer package. Take the spool and ensure the filament end is properly secured. You have two options where to place the filament spool. Either you can use the side spoolholder, or you can use an optional accessory - an external filament dryer. You can visit our e-shop to learn more about this optional accessory. Both the side spoolholder and dryer methods are described in the following two chapters.

In both cases ensure that the tip of the filament is cut to a sharp point as depicted in the photo below.



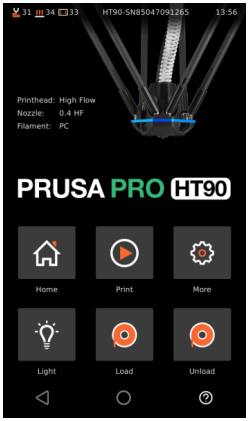
For proper loading, the filament tip must be sharp, and there must not be any visible bumps or other deformations on the filament strand.

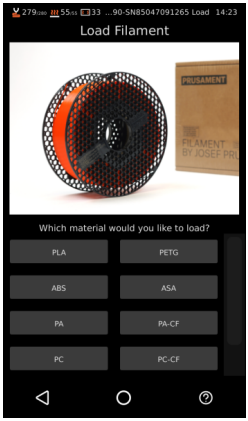
9.5.1. Filament Spool Placed on the Side Spoolholder

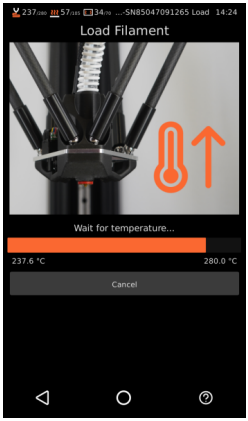
Start by trimming (cutting) the filament end to a sharp point.

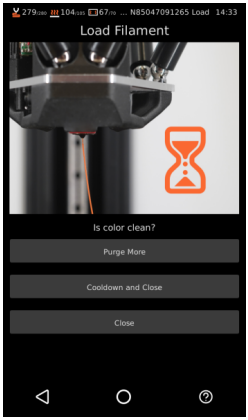
1. On the printer's display, select LOAD FILAMENT.
2. Choose the correct type of filament
3. The print head will start heating to the required temperature.
4. Once the required temperature is reached, insert the filament fully into the print head and press CONTINUE.
5. The extruder will start pushing the filament through the nozzle.
6. Once the loading is complete, the printer will ask if the color of the extruded material is correct. Select one of the options:
    - Purge More - if you are not satisfied, select PURGE MORE as many times as necessary until you are satisfied with the color.
    - Cooldown and Close
    - Close

Clean the nozzle of the filament residue by pulling it off - use pliers, do not use your fingers!









9.5.2. Loading Filament from an External DryBox (Official Accessory)

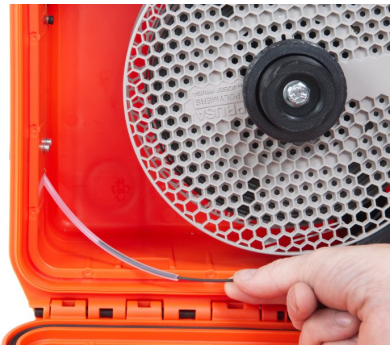
The second option where to place a filament spool is an external box that serves as a filament dryer. Some materials are hygroscopic, meaning it is necessary to dry them before printing and keep them in an environment with as low humidity as possible during printing.

This is why using an external dryer with an integrated hygrometer and thermometer is recommended. We're offering two accessories for the HT90 - the DryBox and the industrial filament dryer. You can see the details at our eshop at prusa3d.com. On the next page is the spool installation process for the DryBox.

Open the dryer by unlocking the two hinges on the top.

Place the spool with the filament on the holder in the center and release the filament end.

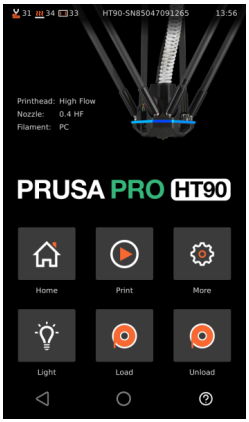
Thread the filament completely through the white PTFE tube and close the dryer by snapping both hinges shut.

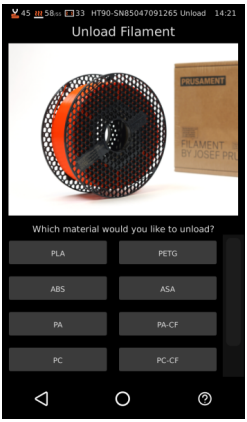


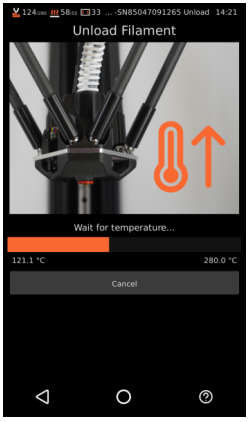
Now proceed as you would with the side spoolholder.

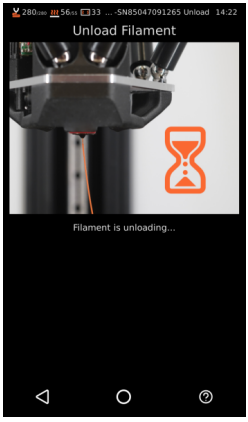
9.6. Unloading Filament

1. On the display, select UNLOAD FILAMENT.
2. The printer will ask if the filament type matches the one it remembers.
3. If yes, the print head will start heating. If not, select the correct material type.
4. The printer will heat the nozzle as needed.
5. The extruder will push out some material and then completely remove it from the nozzle.
6. After the process is complete, choose how to continue:
   - To load new material, completely remove the current filament and select Load new material.
   - To finish and cool down, select Cooldown and Close.
   - To finish but keep the nozzle heated, select Close.
   - If the filament was not successfully removed, select the last option.









9.7. Filament Sensor

The printer is equipped with a filament sensor that detects when the filament on the spool runs out. If this happens and the filament end passes through the sensor, the printer will pause the print and remove the filament from the nozzle. You can then pull it out from the filament insertion point by hand.

If the filament does not pass back through the sensor after removal, simply remove its rear part held by a magnet and manually remove the filament. This situation may happen when printing with a 0.8 mm nozzle and fast profiles.

9.8. Starting the First Print

Final checklist before starting the first print:

- The printer is completely unpacked and free from any transportation/packing material
- The print sheet is installed and free of grease (use isopropyl alcohol to clean it if necessary)
- There is only one print sheet on the heatbed
- Filament is loaded
- The sliding door is closed

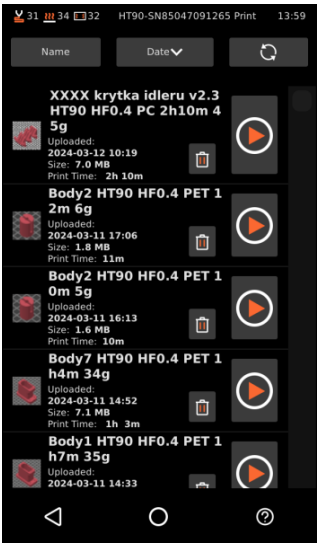
9.8.1. Printing from Internal Memory

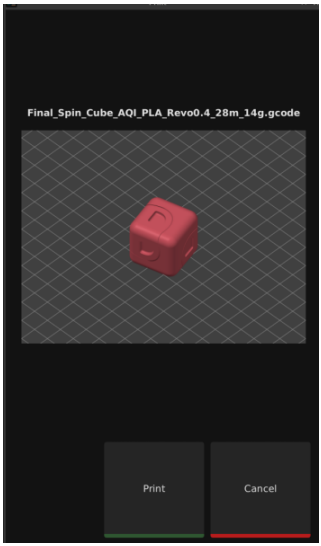
In the LCD's Home Screen, select Print. Then **select the desired file by pressing the arrow button** (see the left image below).

Once the file is selected, confirm (PRINT) or cancel (CANCEL) the file selection (center image). After confirming the print, the printer will heat up as needed for the material used. After heating, it will calibrate for the correct first layer height.

Once the automatic pre-print setup is ready, the HT90 will start printing the model (right image).

For the first print, we recommend using one of the sample models uploaded in the printer's storage. These objects have been prepared by Prusa Research and they have been fully tested. Please note that each model may require a different material. Never print with an incompatible material - e.g., an object that requires PLA filament must not be printed with PCCF filament. The name of the required filament can be found in the name of each file.





9.8.2. Printing from a USB Flash Drive

If the print file is on a USB flash drive, insert it into the USB connector on the side of the display.

In the Print menu, select USB and choose the desired print file.

The rest of the process is the same as for printing a file from the internal memory.

The printer can only recognize a compatible G-code format. It cannot print STL, OBJ, 3MF and other files directly. 3D objects need to be sliced (converted) for use with the HT90. Please see the Printing Your Models chapter for additional information.

9.8.3. Options During Printing

During the print job, several options are available. You can pause the print (Pause), completely stop it (Cancel), adjust print parameters (Tuning), and modify additional settings (Settings).

Pausing the Print

If necessary, you can pause the print by pressing the Pause button. The print head will move away from the printed object, and the printer will wait for the next command.

Canceling the Print

If you need to cancel the print completely, press the Cancel button. The printer will cancel the print job, return the head to the home position, and turn off the heating.

Tuning

During printing, you can adjust several print parameters by pressing the Tuning button.

The first parameter is the first layer height. We strongly recommend monitoring the first layer print through the cutout under the handle. **The first layer is crucial for a successful print.**

If the first layer needs adjustment, use the Z- and Z+ buttons.

The menu contains other items, but we recommend leaving them at default values. The correct values for the material are already preset in PrusaSlicer. These include:

- Print speed
- Extrusion amount
- Nozzle, bed, and chamber temperature

9.8.4. Settings (during a running print)

During printing, you can access additional printer settings.

Available options include:

- Turning the light on/off
- Change filament (during printing) - displays a confirmation screen, the printer then guides you through the filament change
- Adjust temperature (nozzle, bed, chamber)
- System - checks for updates. If the printer finds an update, it can be downloaded even during printing. The update will be installed after the print job is finished.
- Network - available options for connecting to your network. Detailed explanation in the Network Connection chapter.
- Settings - in this section, you can set the printer name, (de)activate door and filament sensors, set the screen timeout, password protection, privacy, the time zone, and restore factory settings.

9.8.5. Removing the Printed Object from the Bed

After completing the print job, wait for the heated bed and chamber to cool down. The print sheet and heated bed may exceed 100°C depending on the settings - contact with unprotected skin can cause burns, so check the heated bed temperature before touching!

Depending on the material type, the print may automatically detach from the print sheet after cooling.

If not, there are several ways to remove the print from the bed.

The first method is to carefully bend the print sheet by its front part.

If the print does not come off, **remove the print sheet and gently bend it on both sides.** Then rotate it 90° and repeat the bending. Be careful not to bend the print sheet too much.

The last method is to use a spatula. If using a metal spatula, as shown in the picture, use it as parallel to the bed as possible to avoid damaging (scratching) the bed.

Ensure all remaining pieces of plastic are removed - don't forget the introductory line printed on the edge.

If there are plastic residues on the print bed, do not remove them with your fingernails, as you might drive a piece of plastic under your nail. Use a spatula to remove plastic residues.

9.9. Sample Models

Prusa Pro HT90 comes with several sample test files (G-codes). These files have been prepared (sliced) and thoroughly tested. If you ever encounter print quality issues, try selecting and printing one of the sample files - especially the Prusa logo keychain and the first layer test. These sample files are designed to test the basic functionality of your Prusa Pro HT90 3D printer.

If a print fails, but the sample files from Prusa Research print correctly, it likely indicates an issue with how your files are sliced. Try preparing them again with the default PrusaSlicer settings and check for common issues:

- Incorrect printer/nozzle profile
- Incorrect material settings
- Missing supports
- Incorrectly configured infill
- The model is not in contact with the print bed

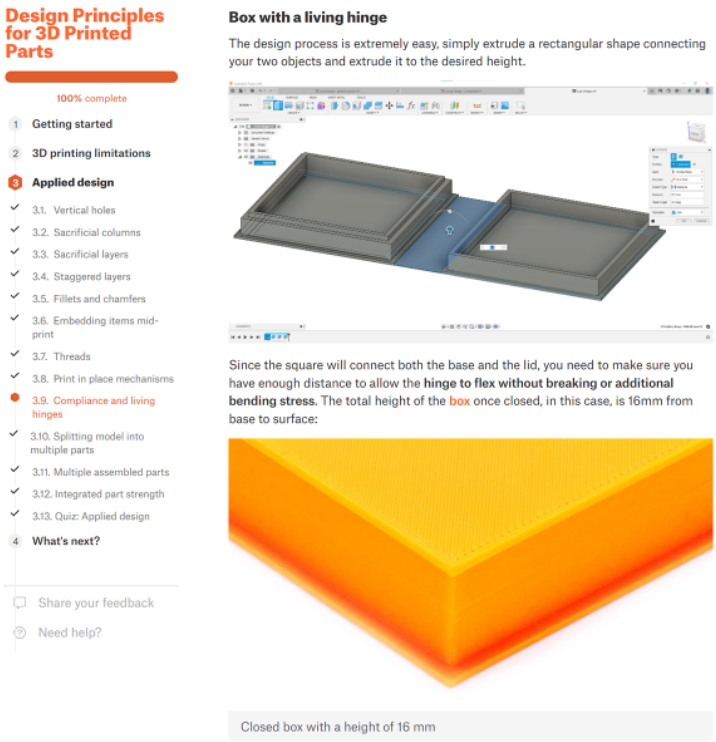
If the sample files do not print correctly, check the Troubleshooting section, our knowledge base at help.prusa3d.com, or contact our technical support.

10. Advanced Guides

This part of the manual covers everything you need to know once you successfully print your first sample object. Network connection, printing your own models, slicing - all this and more are described in the following pages.

Prusa Academy Courses

Become an expert in 3D printing! Prusa Academy offers comprehensive online courses on various 3D printing topics. Each course includes understandable texts with many images and short videos, links for inspiration and further study, quizzes to test your knowledge, and a completion certificate! With our online courses, you will quickly learn to model your own designs and master advanced 3D printing techniques. Visit academy.prusa3d.com.



10.1. Network Connection

The Prusa Pro HT90 3D printer has a built-in Ethernet (RJ45) interface and can be expanded with a Wi-Fi module for wireless network connection. The Wi-Fi module is included in the package.

There are three modes available:

- **Fully offline mode**
- **Local network management** - you effectively connect only to the printer via a local network, no internet connection is necessary
- **Full remote management** - using PrusaSlicer, a secure internet-based remote management system

10.1.1. Establishing a Connection

To connect the printer to the network, connect either an Ethernet cable or the Wi-Fi module as described below.

After connecting the cable/Wi-Fi module, press the More button on the home screen, then press the Network button.

Here you will see information about the printer's network connection. Primarily, these are details about the connection type (Ethernet / Wi-Fi), IPv4 protocol with IP address, and MAC address.

If you wish to connect to the printer via local network, note the IP address of the printer.

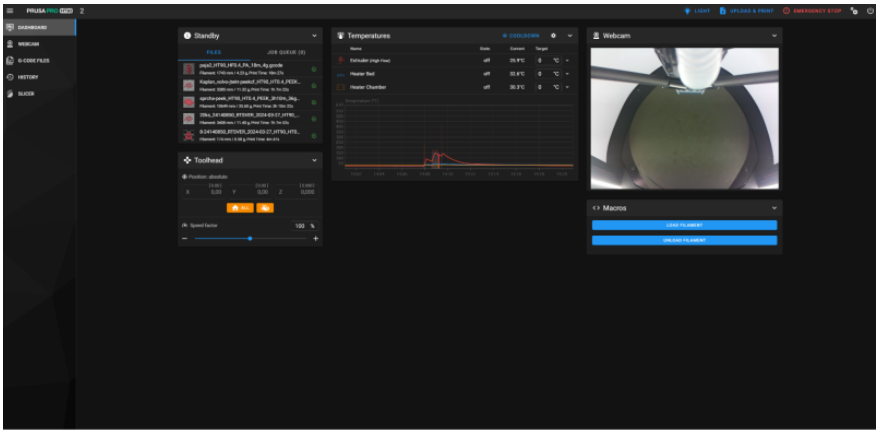
10.1.2. Detailed Connection Settings

Clicking on the Settings gear icon displays a list of connection profiles.

The selected type can be deleted using the Trash button, and after clicking on Settings, details are displayed.

10.2. Web Interface (Local)

Open a web browser of your choice and enter the printer's IP address into the address field. See the previous chapter on how to obtain the printer's IP address. Once you confirm the address, you will be presented with the web interface with a complete overview of the printer. We recommend using Google Chrome.



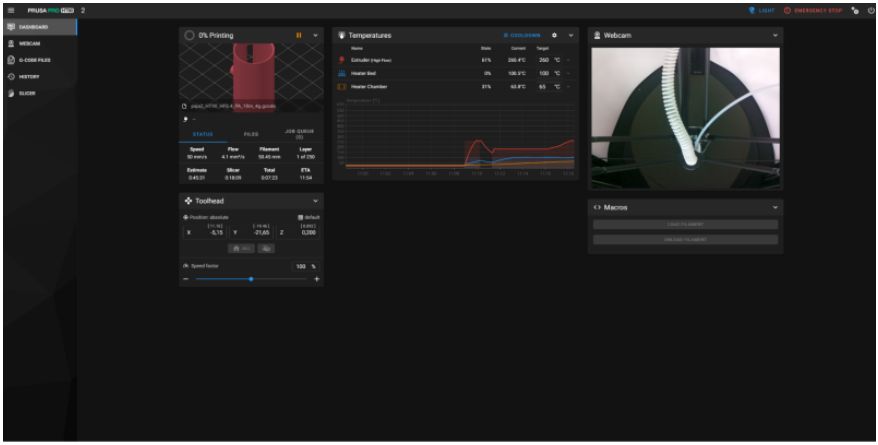
Home Screen

After connecting to the web interface, the Home Screen shows a complete overview of the current printer status.

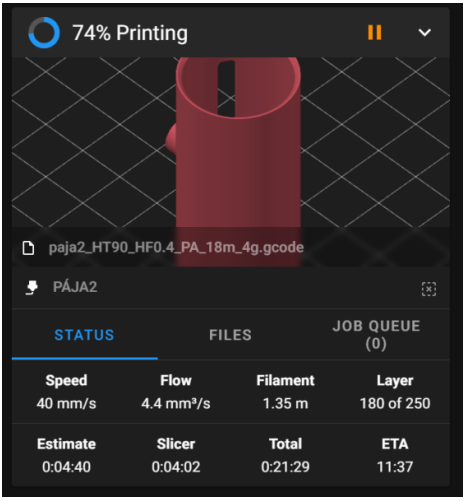
On all the tabs, the top gray bar stays the same. On the left, it shows the printer name, and on the right, buttons for lighting, uploading and starting prints, emergency stop, settings, and shutdown.

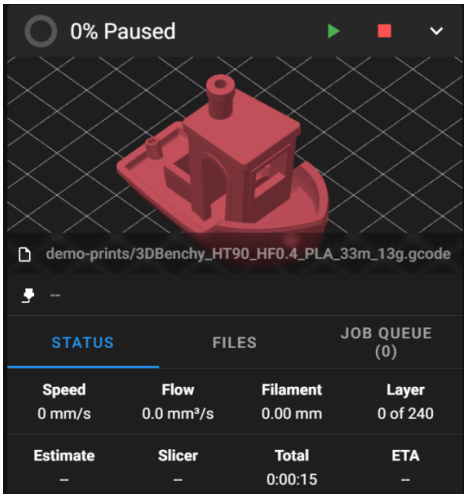
Dashboard

This card serves as an overall overview of the current activities.

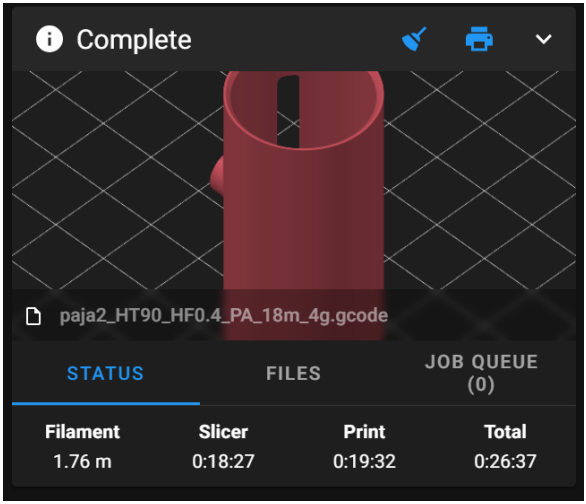


Let's take a detailed look. **The first block on the left** provides a visualization of the printed file, options to pause (orange button), restart/resume (green button), or stop (red button) the print, file name, and current print speed (Speed), material flow (Flow), used filament (Filament), layer (Layer), estimated time to completion by the printer (Estimate), calculated slicer time (Slicer), total time (Total), and task completion time (ETA).

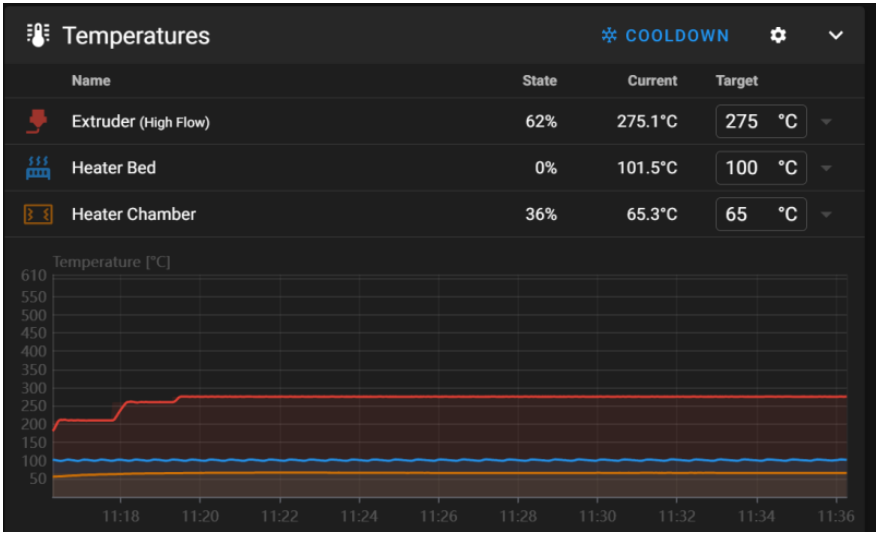




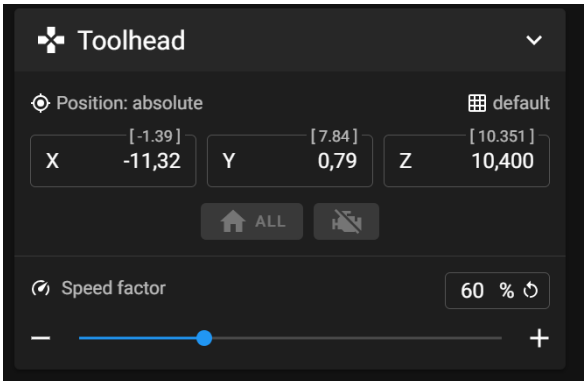
After completing the task, the status changes and you will be presented with the options to clear statistics and reprint the previous task. The final filament consumption and individual times are shown at the bottom.



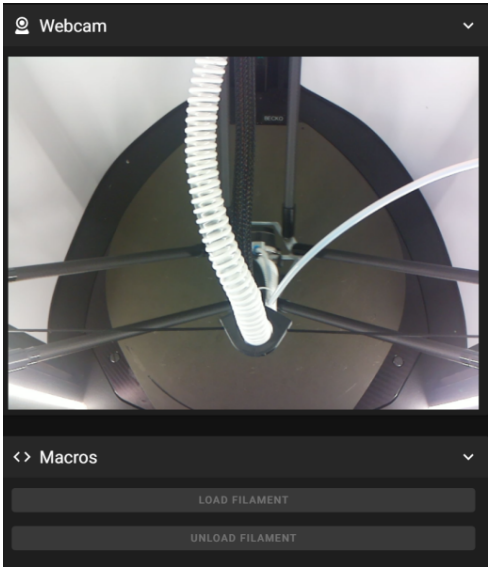
The middle section (Temperatures) provides an overview of the nozzle temperature (Extruder), Heatbed, and Heated Chamber. The State column shows the current heater power value, the Current column shows the current temperature and the Target column shows the desired value. At the bottom, there is a temperature graph showing the last 20 minutes. The red curve corresponds to the nozzle temperature, the blue one to the bed, and the orange one to the chamber temperature.



The next section, Toolhead, shows the current coordinates of the extruder. If no print task is in progress, buttons for returning the print head to the home position and turning off the motors are available. The last item is the Speed factor settings, but we recommend not changing it. The default value provides the optimal performance.

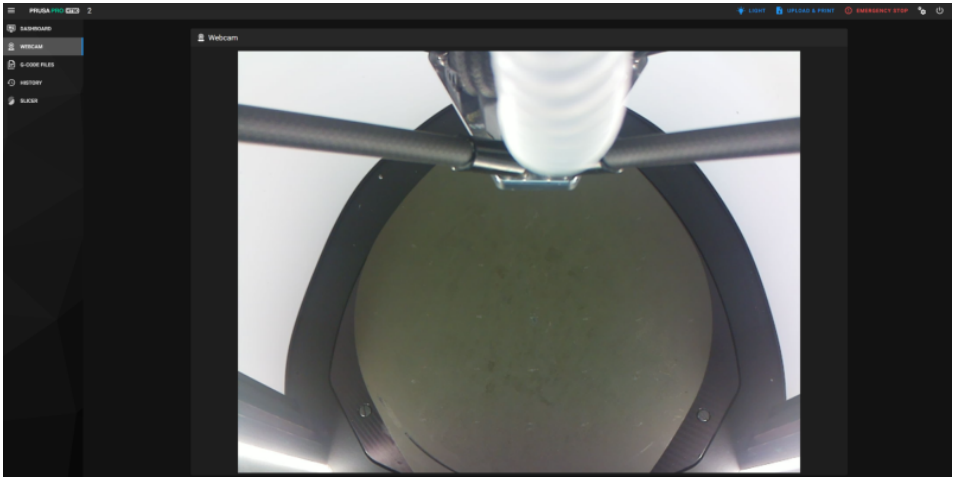


In the last section, Webcam, you can access the camera image. Also, two actions for loading (Load) and unloading (Unload) filament are available when a print job is not running.



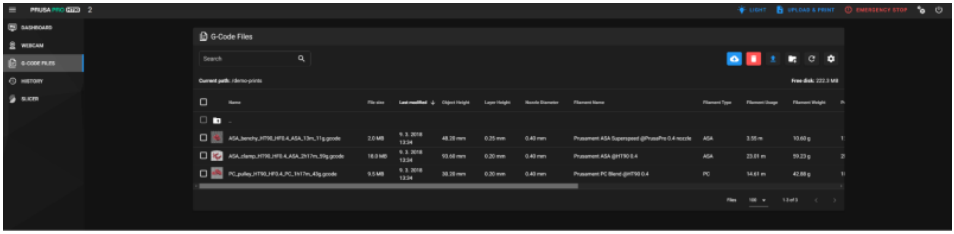
Webcam Tab

This tab displays the top-down camera image. If you need an additional view, connect an external webcam via the USB port and restart the printer.



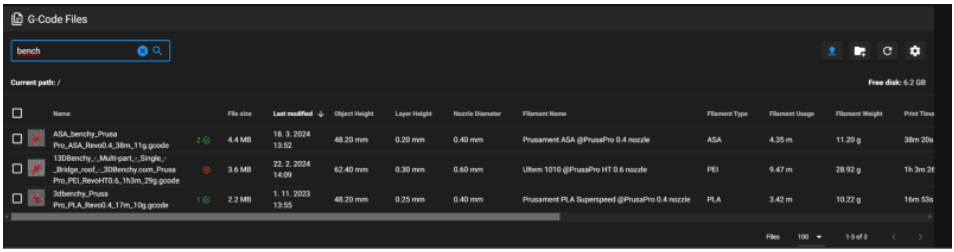
G-code Files Tab

This card manages print files, providing an overview of all folders and files in the printer memory.

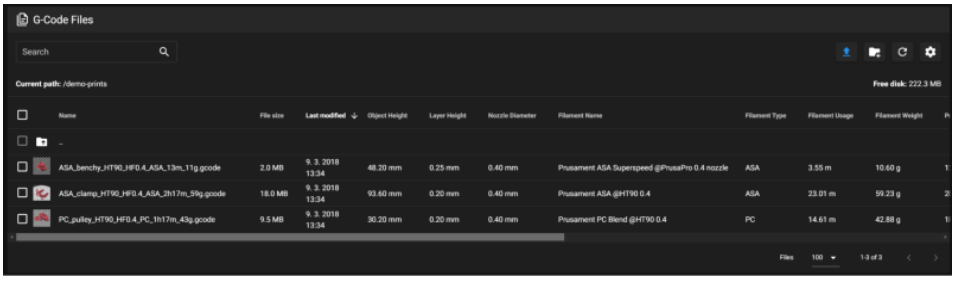


A total of 11.5 GB of internal memory is available.

At the top of this card is a quick search bar. Enter text to automatically search for all matching folders and files.

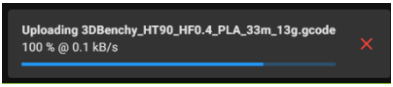


Located in the right corner of this block are buttons for uploading a file (blue arrow up), creating a folder, reloading files, and setting display details. More than 15 details are available, and you can choose which ones to display.



Each print file always provides a preview, name, and print count. We recommend selecting additional information that you find the most useful. Print files can be sorted by various properties, such as the last modification.

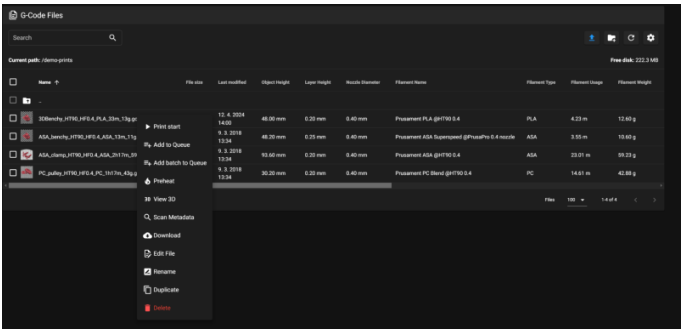
To upload a file to the printer, click the Upload New G-code button (blue arrow up), select the required file, and confirm it to start uploading to internal memory, as shown in the image below.



After a successful upload, a confirmation will appear.

To start printing, click on the file and confirm by clicking Print.

For other actions, right-click on the file to display more options.



The Print Start option in the context menu sends the file to be printed, Add to Queue adds the file to the print queue, Add Batch to Queue adds the selected files to the print queue, Preheat starts preheating the printer according to the specifications in the file. View 3D shows a preview of the model.

Be careful, selecting Delete immediately deletes the file from the printer memory!

History

This card provides an overview of print history, print time statistics, print completion types, a graph of filament lengths, and print times for individual files.

Slicer

Here you can download the latest version of PrusaSlicer with prepared profiles for your printer. Click the Download button to download the installation file.

10.3. Prusa Connect

Prusa Connect is our web service that allows you to manage 3D printers from anywhere. This includes uploading print files, controlling and monitoring multiple printers simultaneously, tracking the status of print jobs using cameras, or tracking print history and detailed printer statistics, including filament consumption. Many features are already implemented, and many more are in preparation!

To access the Connect application, visit connect.prusa3d.com and log in with your Prusa Account (free account, required). Then click Add Printer, select Prusa Pro HT90 from the list, and enter the code found in the printer menu in Settings - Network - Prusa Connect - Add Printer to Connect.

Prusa Connect can be used with any number of printers, from one machine to an entire print farm. Each registered printer has a print queue that can be filled with the desired print files. This means that if you are printing a larger project, you can prepare and send all G-codes to the printer at once. After completing the print job, simply collect the finished model and confirm that the printer is ready for the next job in the queue. The print queue has a timeline to visualize when individual print jobs will be completed, making it easy to track print status across all printers simultaneously.

You can also remotely control and adjust printer settings. This includes initiating the homing procedure, setting temperatures and speeds, moving any axis including the extruder, turning off the motors, or even restarting the entire printer.

11. Printing Your Models

Your Prusa Pro HT90 3D printer should now have its first print successfully completed. Now you want to print your own models.

Let's go through the necessary steps briefly - everything is explained on the following pages. If you need help with slicing, check out our knowledge base at help.prusa3d.com or our Prusa Academy courses at academy.prusa3d.com, which will teach you everything about 3D printing.

- Find or create a suitable object for 3D printing (usually in .STL, .3MF, or .OBJ format).
- Import the object into PrusaSlicer - download it from www.prusaslicer.com
- Select the desired print profile depending on the level of quality, material, and speed you require.
- Use built-in tools to move, rotate, and resize the object. Try to find the optimal orientation for the object, i.e.:
  - A large flat base that sits on the print sheet.
  - Minimal overhangs to reduce the number of supports needed for printing.
  - If supports are necessary, try to orient the object so that the supports are not in direct contact with areas that need high print quality (e.g., a face).
- Choose the infill type and its density.
- Slice the object.
- Check the preview.
- Export the G-code and start printing.

11.1. What is a G-code File?

3D models you create or download from the internet need to be converted from their original format (.stl, .obj, .3mf, etc.) into a file that contains specific instructions for the printer - the G-code. This is the only format recognized by 3D printers. This file contains instructions about nozzle movement, filament extrusion amount, temperature settings, and fan speeds. To turn a 3D model into a G-Code, our internally developed software PrusaSlicer is the ideal choice.

11.2. PrusaSlicer

As the name suggests, PrusaSlicer (prusaslicer.com) is a slicing software developed by Prusa Research which is based on the open-source project Slic3r. PrusaSlicer is open-source, packed with useful features, and frequently updated, so it includes everything you need to export perfect print files (not only) for Prusa Pro HT90. PrusaSlicer boasts features like:

- Clear and simple user interface
- Fine-tuned print and material profiles with automatic updates
- Print time analysis and breakdown
- User-defined supports and modifiers
- Shape Gallery
- Variable layer height
- Color printing
- Detailed print settings

PrusaSlicer is constantly evolving and bringing new features and improvements, often based on user feedback. Even small updates can significantly impact your 3D printing experience - whether it's improving quality, shortening print time, or minimizing filament consumption.

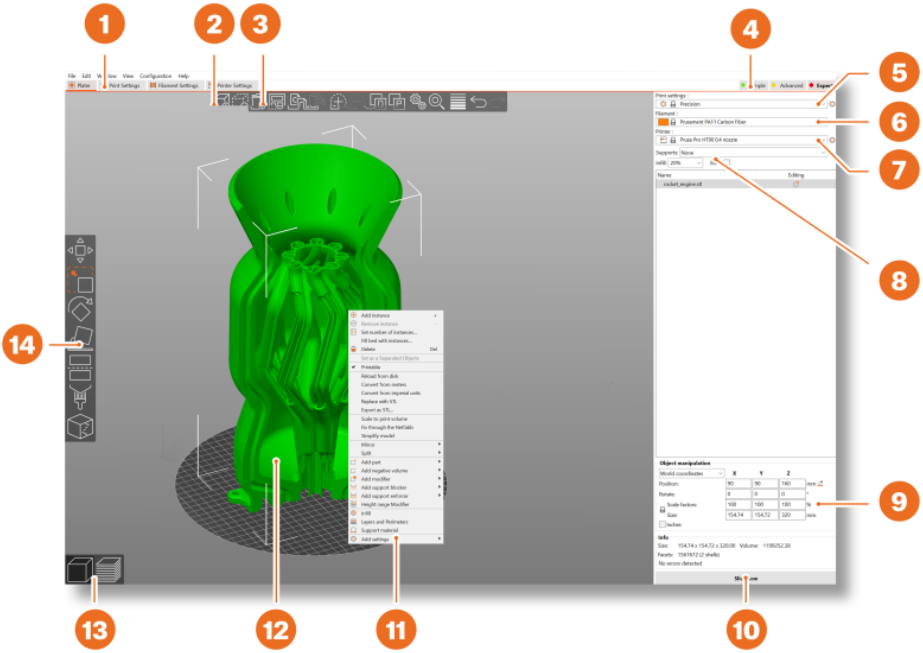
PrusaSlicer comes with a G-code viewer, a simple application for quickly previewing G-codes from all popular slicers. It behaves identically to the PrusaSlicer preview (using the same code), but you can load external G-code files.

Currently, we analyze and interpret G-codes from applications PrusaSlicer, Slic3r, Slic3r PE, CURA, ideaMaker, Simplify3D, Craftware, and KISSSlicer to some extent. The PrusaSlicer G-code Viewer is part of the PrusaSlicer installation package. Just download the latest version of PrusaSlicer, and the standalone G-code Viewer will automatically install with it.

Download PrusaSlicer!

The latest stable version is always available at prusaslicer.com. Development alpha/beta versions can be downloaded from github.com/prusa3d/PrusaSlicer - these are less stable (non-final) versions but with the latest features.

11.3. PrusaSlicer Interface



1. Tabs with detailed print, filament, and printer settings
2. Add button - import a 3D model into the scene.
3. Delete and Delete all buttons - remove model(s) from PrusaSlicer
4. Simple, Advanced, and Expert modes toggle
5. Print quality and speed settings
6. Material selection
7. Printer selection
8. Quick settings for Infill density, Supports, and Brim
9. Information about model size and print time (displayed after slicing)
10. Slice button (turns into Export button after slicing)
11. Right-clicking on a model opens the context menu
12. Model preview in 3D
13. Toggles between the 3D editor and Preview mode
14. Move, Scale, Rotate, Place on bed, Cut, Paint supports, Paint seam position tools

11.4. Initial Setup and General Procedures

PrusaSlicer allows you to import objects in STL, OBJ, AMF, STEP and 3MF formats - these are the most common types of 3D files you can find on the internet. You can either drag them directly into the 3D editor window, or use the Add... button from the top bar.

Upon launching PrusaSlicer, a Configuration Wizard should start. If it doesn't start automatically, open it from Configuration -> Wizard. Use the Wizard to add the HT90 and materials of your choice into PrusaSlicer. You can launch this wizard any time to add more materials in the future.

In the main PrusaSlicer window, look for the **Printer** drop-down menu on the right (no. 7 in the illustration above). **Make sure Prusa PRO HT90 is selected.**

Then select **Print Quality** from the top drop-down menu (Print Settings - no. 5 in the illustration above).

A vast majority of objects require **an infill and supports.** A good value for infill is between 5-15 % depending on how durable the object should be.

Supports require slightly more work. There are objects that can be printed completely without supports, but if there are steep overhangs or part of the object that would start mid-air, supports are required. More information can be found in the following chapter.

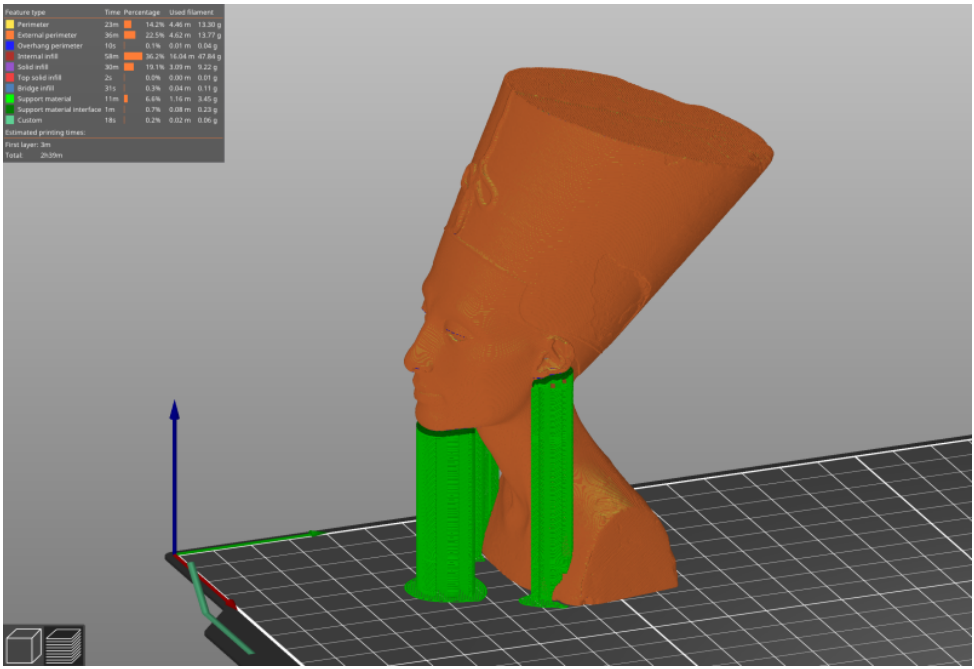
To modify the model, **use the tools in the left sidebar,** i.e. Move, Scale and Rotate. If an object is blue, it means it does not fit into the print bed and it needs to be moved or scaled down. There is no universal way to place the model on the bed, it always depends on the specific shape. However, a general rule is that the bigger the flat surface of the model that touches the bed, the better it will hold - so try to orient the largest flat surface of the model downwards.

11.5. Using Supports

Supports are printed structures resembling scaffolding. They are used for printing complex objects. After printing, they can be easily separated from the output.

You can find (or design) objects that can be printed without supports - just place them in the right orientation on the bed, slice them and you can print. Not all objects, however, can be printed without supports.

If you are printing an object with walls that rise at an angle less than 45°, these overhangs will cause issues with print quality. Also, keep in mind that the printer cannot start printing mid-air. In such cases, supports are necessary. You can enable safe default values by choosing "Everywhere" in the Supports drop-down menu in PrusaSlicer.



How to tell whether an object needs supports?

The shortest answer is: it comes with experience. With your first prints, stick to the default PrusaSlicer values ("Supports Everywhere"). Once you feel comfortable with printing complex shapes using default support settings, try playing around with the Overhang threshold option in the settings. We have an extensive list of detailed tutorials available at help.prusa3d.com/category/prusaslicer_204 if you need guidance.

You have three options to choose from when selecting support generation:

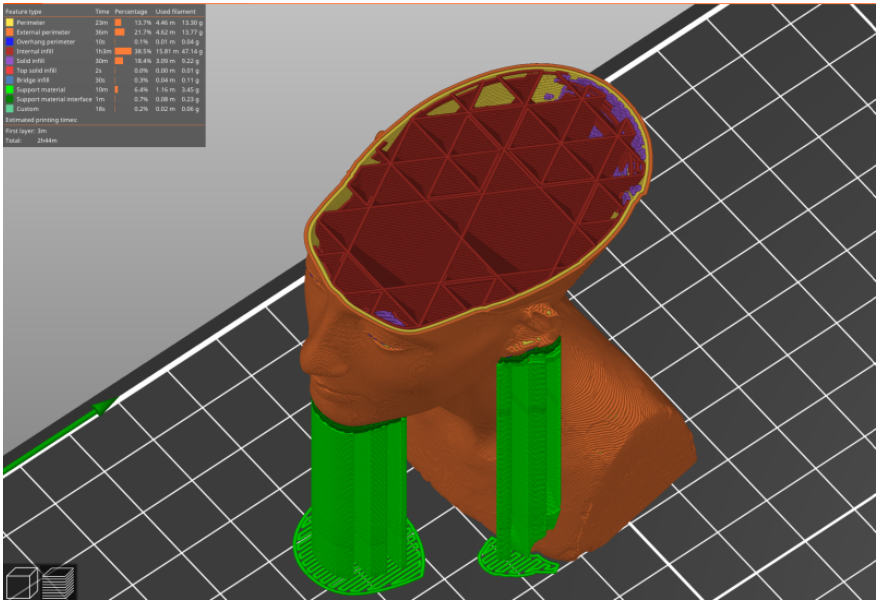
- Support on Build Plate Only - generates supports only in the space between the object and the print bed.
- For Support Enforcers Only - generates supports only where enforced by placed modifiers.
- Everywhere - generates supports everywhere.

The default support pattern usually works correctly, but if you need to modify the places where the supports will be generated, just go to the Print Settings tab and select Support Material.

- Check the Generate Support Material box.
- The Overhang Threshold allows you to set the minimum angle for printing the support material. Setting the value to zero will enable an automatic calculation. Try generating supports with different angle settings to see which value works best for your object.
- Enforce Supports is an option mainly used for small models or models with a small base to prevent them from being broken or detached from the print bed during the printing process. Wherever the supports touch the model, they are usually associated with a lower surface quality. Try to reduce or even avoid the need for supports by rotating or shifting the model accordingly.

11.6. Infill

Another parameter that affects the properties of the printed object is Infill. It affects the printing speed, strength and appearance of the object. Objects printed with the FFF/FDM method usually do not have 100% density. Instead, they contain a certain geometric structure inside. It can take various forms, from simple square grids or hexagons to more complex patterns. The purpose of the infill is to stiffen the object from the inside. Most models are printed with 10-15% infill, but if you need a really solid structure, you can choose a higher density.



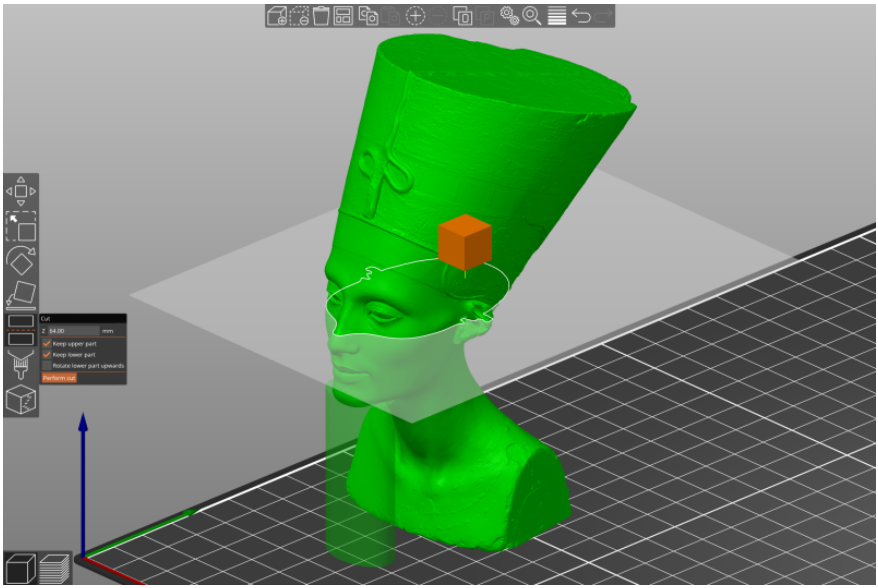
11.7. Brim

The brim serves **to increase adhesion to the bed,** reducing the risk of warping. A wider first layer is printed around the model. This makes sense especially if the model only touches the bed in a small area. This function can be enabled in PrusaSlicer by checking the "Brim" box in the menu in the right column. After the printing is finished, the brim can usually be removed easily by hand, or you can use a knife or scalpel.

11.8. Printing Objects Larger than the Print Volume

The Prusa Pro HT90 has a print area of Ø 300 mm and maximum Z-height of 400 mm. If this is not enough for your project, you can use PrusaSlicer's built-in tools to find a solution.

Of course, you can also resize the imported model to fit the bed. The Scale tool is there to help you with that. If you want to print an object that is too large in its original size for the print bed, you can cut it into several smaller parts. **Use the Cut tool** from the left menu (or press the letter C). Place the cutting plane manually or set an exact height using the Cut tool dialog. Choose whether you want to keep only the part above the cut, below it, or both.

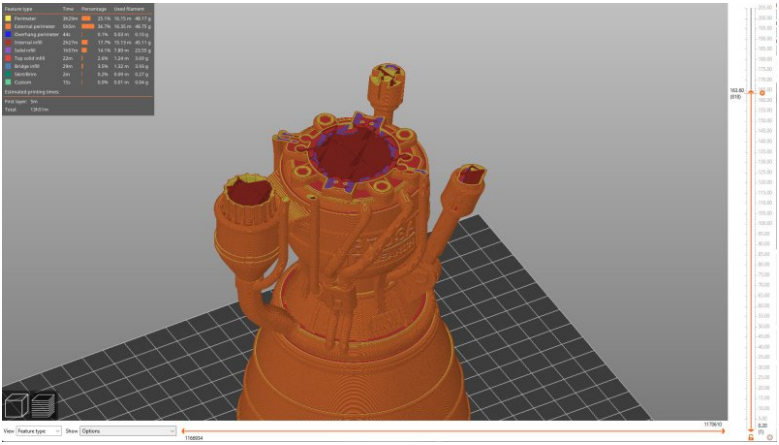


11.9. Slicing and Exporting

One of the most important phases of the slicing process is the final check of the sliced object in the Preview.

Once your object is scaled, rotated, and supported, the infill is set and the correct material is selected in the menu, you can press the Slice button so that PrusaSlicer can process the 3D model. Once this is done, a Preview is displayed.

Using the slider on the right, review all the print layers of the object one by one. This will help you identify problematic spots - for example, if the bottom of the object doesn't stick well to the bed or if some of the parts are missing supports and are "hanging in the air". This is the best way to prevent potential print issues. If you are sure that there are no problematic spots, export the file as a G-code.



Before you export the model as G-code and upload it to the USB drive, always check it in the Preview first. It's the best way to avoid mistakes during printing.

12. Material Guide

Complete material table available online! Due to the limited space in this guide, we can only provide a brief overview of popular materials. Visit **help.prusa3d.com/materials** for a detailed overview of a wide range of printing materials. The Prusa Pro HT90 is compatible with almost all available filaments.

Print with Prusament!

Prusament (prusament.com) is our brand of high-quality filaments that we develop and manufacture completely in-house. We were not satisfied with the quality of filaments on the market, so we decided to make our own! The entire production process is carefully monitored, and each spool is thoroughly tested – we measure diameter, color consistency, and mechanical properties.

We are the only manufacturer that allows customers to check the parameters of each produced spool of filament thoroughly. Simply scan the QR code on the spool and you will see all the tested parameters of that spool on our website. We offer a wide range of materials that are continuously expanding at prusa3d.com.

12.1. PLA

PLA is the most commonly used material for 3D printing. It is easy to print with and prints from PLA are very hard. It is the perfect choice for printing large objects due to its low thermal expansion (prints do not warp on the print bed) and for printing detailed small models.

Advantages

- Easy to print, suitable for beginners
- Printing small, detailed models
- Seamless printing of larger objects
- Almost odorless
- Affordable
- Wide range of colors

Disadvantages

- Brittle and inflexible
- Low thermal resistance (50-60°C)
- Difficult post-processing
- Not suitable for outdoor use (low UV/temperature resistance)

Typical Uses: Prototypes, toys, figurines, jewelry and small detailed models in general, architectural models, etc.

PLA can be printed on a standard print sheet. For post-processing, wet sanding is best for achieving better results. Using dry sandpaper may cause heat from friction to deform the printed object. PLA can only be dissolved in chemicals like chloroform or heated benzene. For bonding, use high-quality superglue; some types of PLA can also be bonded with acetone.

Nozzle temperature: 215°C **Heated bed temperature:** 50-60°C **Print surface:** Ensure the print bed is clean according to the instructions in the Regular Maintenance chapter.

12.2. PETG

PETG is one of the most popular materials for 3D printing. It is a great choice for parts that will be subjected to mechanical stress. Compared to PLA, it has higher thermal resistance, is more flexible and less brittle. Due to its low thermal expansion, it adheres well to the print bed and does not deform. Printing with it is almost as easy as with PLA, but unlike PLA, it offers much better mechanical properties. We print parts of our printers from PETG.

Advantages

- High thermal resistance
- Easy to print
- Low thermal expansion
- Durable and tough, good layer adhesion
- Easy to process (sanding)
- Almost odorless printing

Disadvantages

- Not suitable for printing small/very detailed models
- The nozzle may leave thin threads ("stringing")
- Problematic bridging and overhangs
- Strong adhesion to the print bed
- Cannot be smoothed with common solvents, only soluble in hazardous chemicals.
- Removing supports can be difficult

Typical Uses: Mechanical parts, holders and cases, waterproof prints (flower pots).

PETG requires a higher heated bed temperature (85°C). PETG usually has poorer quality when bridging two points and tends to string - leaving fine plastic threads on the surface of the print (which can be easily removed). Stringing can be minimized by adjusting appropriate retractions and using lower print temperatures - we recommend sticking to the values provided in PrusaSlicer profiles. Printing should be well-cooled for better details and to prevent stringing. However, for the strongest print, try turning off the print fan. Higher temperature causes layers to adhere better, leading to better mechanical resistance. Generally, we recommend printing the first few layers with the fan off (for adhesion) and then turning it on to 50% power.

Nozzle temperature: 240°C **Bed temperature:** 70-90°C **Print surface:** A standard print sheet does not require any special preparation, just keep it clean and free of grease.

12.3. ASA (ABS)

ASA and ABS are very similar materials. ASA is better than ABS in many ways. ASA is UV stable and shrinks less during printing compared to ABS. In terms of post-processing, both materials are similar. We will discuss ASA, but the same applies to ABS.

ASA is a strong and versatile material. A higher melting temperature than PLA ensures good thermal resistance, so your prints won't show signs of deformation up to around 100°C. However, compared to PLA, ASA has very high thermal expansion, complicating printing, especially for larger models. Even with a heated bed set to 100°C, prints can deform and separate from the bed. ASA also produces a significant odor during printing.

Advantages

- High impact and wear resistance (but lower than PETG)
- Very good thermal resistance
- Suitable for outdoor use - UV stable
- Soluble in acetone - can be used for bonding or smoothing
- Detailed prints without "stringing" (leaving threads on the surface of the print)
- Easy post-processing (e.g., sanding, cutting, etc.)

Disadvantages

- More challenging to print
- Tends to warp (recommended to print in an enclosed box)
- Unpleasant odor during printing (contains styrene)

Typical Uses: Cases and protective covers, prototypes, spare parts, toys and figurines, parts suitable for outdoor use.

Printing with ASA/ABS is significantly easier if the printer is placed inside an enclosure. During printing, the heated bed quickly warms the enclosure, reducing thermal shock to the extruded filament. Consequently, both warping and layer separation are significantly reduced. Using acetone makes it easy to bond multiple prints together. Simply coat the contact surfaces lightly and press the parts together. Additionally, prints can be smoothed with acetone vapors to achieve a perfectly glossy surface. Handle acetone carefully!

Nozzle temperature: 220-275°C **Bed temperature:** 90-110°C (larger objects require higher temperature) **Print surface:** ASA and ABS can be printed on a standard print sheet, which does not require any special preparation - just keep it clean and grease-free.

12.4. PC (Polycarbonate) and PC Blend

Polycarbonate (PC) is a technical material with excellent toughness, tensile strength, and high-temperature resistance. However, it is very challenging to print, suitable mainly for advanced users. This does not apply to our Prusament PC Blend, which is much easier to print compared to other polycarbonates. Polycarbonate surpasses all previously mentioned materials in mechanical, chemical, and thermal resistance.

Advantages

- High thermal resistance
- High toughness and tensile strength
- Clear polycarbonate is transparent
- Good electrical insulation properties
-

Disadvantages

- Pure polycarbonate is highly hygroscopic
- High nozzle and bed temperature
- Strong warping, especially for large models
- Slight odor during printing
- A separating layer is recommended
- High cost

Typical Uses: Polycarbonate is suitable primarily for technical components requiring high resistance to mechanical wear and high temperatures.

Consider printing in an enclosed chamber (printer box/enclosure) to prevent deformation of printed objects; enable the "Shield" function in PrusaSlicer - a protective outline is printed around the object throughout its height, creating a better microclimate. Do not print in a room with low temperature.

Nozzle temperature: 270-275°C **Bed temperature:** 110°C for the first layer, 115°C for subsequent layers. **Print surface:** PC and PC Blend adhere best to standard print sheets.

12.5. PVB

Polyvinyl butyral (PVB) is a material that can be easily smoothed using isopropyl alcohol (IPA). When properly set, prints are translucent, making PVB suitable for printing vases, lampshades, and other decorative models. Print settings are similar to PLA, but PVB has slightly better mechanical properties.

Advantages

- Similar print settings to PLA
- Transparent filament
- Suitable for decorative models - vases, lampshades, etc.
- Chemical smoothing with IPA
- Good toughness and tensile strength
- Less prone to warping (less than PLA)
- Suitable for use with a 0.8mm nozzle

Disadvantages

- Lower layer adhesion
- Hygroscopic material (absorbs moisture)
- Higher cost

Typical Uses: PVB is best used for printing transparent models - e.g., jewelry, vases, lampshades, etc.

PVB adheres best to standard print sheets. If you want to print transparent prints that you will later smooth with isopropyl alcohol, we recommend using a larger nozzle (0.8mm) and enabling the "Spiral Vase" mode in PrusaSlicer. When printing with multiple perimeters, individual layers will be clearly visible even after smoothing with isopropyl alcohol.

Store the filament in a dry environment - PVB is extremely hygroscopic, which negatively affects print quality. Always store the filament back in a resealable plastic bag and add silica gel. Alternatively, dry it for 4 hours at 60°C before printing.

The main advantage of PVB is that it can be smoothed with isopropyl alcohol (IPA). Models printed from PVB can be smoothed in IPA vapors, by immersion in an IPA bath, or by directly applying IPA to the surface of the object (using a spray or brush).

Nozzle temperature: 215±10°C **Bed temperature:** 75°C **Print surface:** PVB adheres best to standard print sheet.

12.6. Flexible Materials

Flexible filaments are generally very strong and flexible materials. In many cases, rigid plastic (PLA, PETG) may not be ideal or even completely unsuitable for printing a model. Whether you are printing a phone case, an action camera housing, or wheels for a remote-controlled car, it is better to use a flexible material. Flexible filaments are useful, but printing with them can be more challenging. Before starting to print with Flex, clean the nozzle of the previous material by inserting PLA into the preheated extruder and extruding all the previous material.

Advantages

- Flexibility and elasticity
- Minimal shrinkage
- Excellent layer adhesion
- High wear resistance

Disadvantages

- Careful handling of the filament is required
- Problematic bridging and overhangs
- Lower printing speed required
- Higher cost
- Absorbs moisture - must be stored in a dry environment

For the Prusa Pro HT90 3D printer, we recommend using Semiflex or Flexfill 98A filaments, Filatech FilaFlex40, or any filament for which we have tuned profiles in PrusaSlicer.

Nozzle temperature: 220 - 260°C **Bed temperature:** 40 - 85°C (larger objects require higher temperature) **Print surface:** Flexible materials adhere best to standard print sheets.

12.7. PA (Polyamide) / PA11CF

Polyamide (known as Nylon) is a versatile material known for its durability and is commonly used for 3D printing special models despite high print difficulty (does not apply to PA11CF) and higher costs. There are several types of polyamide that differ in properties such as temperature resistance, water absorption, and adhesion to different types of surfaces. Prusament PA11CF has high-temperature resistance (up to 192°C), strong resistance to many chemicals, and is easy to print. Some polyamides, including PA11CF, are reinforced with carbon fibers to reduce shrinkage. We recommend PA11CF for printing extremely stressed parts such as plastic engine components, etc.

Advantages

- High-temperature resistance (up to 192°C)
- Resistance to many chemicals
- Hard and durable in thick layers, flexible in thin layers
- Smooth glossy surface of pure PA
- Excellent layer adhesion
- Suitable insulating material

Disadvantages

- Not suitable for printing small/very detailed models
- Prone to warping (not applicable to PA11CF)
- Hygroscopic material (absorbs moisture)
- Strong (or too weak) adhesion to the surface
- Cannot be smoothed with common solvents, and dissolves only in hazardous chemicals.
- Removing supports can be difficult

Typical Uses: Mechanical parts, holders and cases, electrical insulating parts, moving parts, and parts requiring high-temperature resistance.

It is essential to keep the filament dry, otherwise, its adhesion and overall print quality will significantly deteriorate. Therefore, we recommend drying the filament for at least 4 hours at a maximum temperature of 90°C before printing. A dried polyamide print should have a smooth and glossy surface, while carbon fiber-reinforced materials have a matte surface. When printing polyamides, we recommend using the printer in an enclosed box (enclosure) with active filtration or having the printer in a well-ventilated room. Not only do (all) PA release potentially hazardous particles during printing, but higher ambient temperature also reduces warping and improves layer adhesion. Carbon-reinforced polyamides can be printed without a covered printer, but due to internal tension caused by sudden temperature changes, the finished prints may have slightly worse mechanical properties.

Nozzle temperature: 240 - 285°C **Bed temperature:** 70 - 115°C **Print surface:** For printing most polyamides, we recommend using our special nylon print sheet, which ensures ideal adhesion even when cleaned with water only. However, some types of polyamides may have too high adhesion, which can lead to damage to the sheet, so we recommend checking compatibility in our material table (help.prusa3d.com/materials).

12.8. PEI (Polyetherimide) / ULTEM

Ultem, also known as PEI (Polyetherimide), is a high-performance thermoplastic material ideal for 3D printing. Its extreme resistance to heat and chemicals makes it a great choice for demanding applications. With a temperature range from -70°C to over 200°C, Ultem offers high stability and strength in a wide range of environments. In 3D printing, it requires stable print conditions and specialized equipment, but the results are precise and durable. Ideal for the automotive industry, aerospace industry, medical devices, and prototyping, Ultem is a material that can handle challenges even in the most demanding environments.

Advantages

- High-temperature resistance (over 200°C)
- Exceptional chemical resistance
- Resistance to radiation
- Suitable insulating material
- Self-extinguishing material
- Dimensional stability during printing

Disadvantages

- High cost
- Prone to warping
- Highly hygroscopic material (absorbs moisture) - requires drying at 130°C and use of a dry box
- Requires adhesives to increase adhesion to the print bed
- Removing supports can be difficult

Typical Uses: Alternative to aluminum, gearbox components, throttle bodies, ignition components, sensors, and thermostat housings, biomedical device components, holders for sterilizing medical instruments, laboratory equipment

It is essential to keep the filament dry, otherwise, its mechanical properties will significantly deteriorate. Ultem is extremely hygroscopic and begins to absorb moisture from the air within 10 minutes, affecting the resulting quality and strength. Therefore, we recommend drying the filament for at least 4 hours at 130°C before printing and then using a dry box. Printing this material requires an enclosed and heated chamber with a minimum temperature of 85°C. After printing, the material shrinks during the gradual cooling of the chamber. Therefore, it is important to remove the print from the printer as soon as possible after printing.

Nozzle temperature: 380 - 420°C **Bed temperature:** 145 - 155°C **Print surface:** For printing, it is necessary to apply a special adhesive to the print bed (e.g., MAGIGOO HT) for the perfect first layer. We recommend applying the adhesive in multiple layers.

12.9. PPS (Polyphenylene sulfide)

PPS or Polyphenylene sulfide is a technical thermoplastic material suitable for 3D printing characterized by high resistance to heat and chemicals. With thermal stability from -100°C to over 200°C, PPS is an ideal choice for applications requiring resistance in extreme temperature conditions. This material has excellent mechanical properties, including high toughness, strength, and wear resistance. Due to its resistance to chemicals, PPS is suitable for a wide range of industrial applications, including chemical, automotive, and electrical industries.

Advantages

- High-temperature resistance
- Exceptional chemical resistance - not soluble in any solvent below 200°C
- Resistance to radiation
- Suitable insulating material
- Self-extinguishing material
- Dimensional stability during printing
- Low moisture absorption

Disadvantages

- Higher cost
- Prone to warping
- Requires drying
- Requires adhesives to increase adhesion to the print bed
- Removing supports can be difficult

Typical Uses: Valve components, pipes, pumps, sensors, seals, insulation boards, relays, connectors, and other electronic components.

It is essential to keep the filament dry, otherwise, its mechanical properties will significantly deteriorate. Despite the low moisture absorption, we recommend drying the filament for at least 4 hours at 110°C before printing and then using a dry box. Printing this material requires an enclosed and heated chamber with a minimum temperature of 65°C.

Nozzle temperature: 330 - 370°C **Bed temperature:** 120 - 150°C **Print surface:** For printing, it is necessary to apply a special adhesive to the print bed (e.g., MAGIGOO HT) for the perfect first layer. We recommend applying the adhesive in multiple layers.

12.10. PPSU (Polyphenylsulfone)

PPSU, Polyphenylsulfone is a technical thermoplastic material with excellent resistance to heat and chemicals, ideal for applications requiring extreme stability and durability. Its properties include high mechanical strength, toughness, and wear resistance. With a temperature range from -100°C to over 200°C, PPSU is suitable for applications where extreme temperature resistance is needed.

Advantages

- Excellent temperature resistance (better than Ultem)
- Good chemical resistance
- Resistance to radiation
- Possibility of sterilization
- Self-extinguishing material
- Dimensional stability during printing

Disadvantages

- Higher cost
- Prone to warping
- Requires drying
- Requires adhesives to increase adhesion to the print bed
- Removing supports can be difficult

Typical Uses: Sterilized and chemically resistant components in medicine - casings, surgical instruments, pipes, valves. Additionally, connectors, relays, and aerospace components.

It is essential to keep the filament dry, otherwise, its mechanical properties will significantly deteriorate. Despite the low moisture absorption, we recommend drying the filament for at least 4 hours at 110°C before printing and then using a dry box. Printing this material requires an enclosed and heated chamber with a minimum temperature of 65°C.

Nozzle temperature: 370 - 410°C **Bed temperature:** 140 - 155°C **Print surface:** For printing, it is necessary to apply a special adhesive to the print bed (e.g., MAGIGOO HT) for the perfect first layer. We recommend applying the adhesive in multiple layers.

12.11. PES (Polyethersulfone)

PES, short for Polyethersulfone, is a technical thermoplastic material whose excellent combination of properties makes it an ideal choice for a variety of applications, including 3D printing. Its high resistance to heat, chemicals, and mechanical stress guarantees reliability and performance in all conditions.

This material has excellent mechanical properties such as high strength, toughness, and wear resistance, making it suitable for applications requiring durability in harsh environments.

Advantages

- Excellent temperature resistance (better than Ultem)
- Good chemical resistance
- Resistance to radiation
- Possibility of sterilization
- Self-extinguishing material
- Dimensional stability during printing

Disadvantages

- Higher cost
- Prone to warping
- Requires drying
- Requires adhesives to increase adhesion to the print bed
- Removing supports can be difficult

Typical uses: Sterilized and chemically resistant components in medicine - packaging, surgical instruments, pipes, valves. Also connectors, relays, parts of airplanes and space shuttles.

It is essential to keep the filament dry, otherwise, its mechanical properties will significantly deteriorate. Despite the low moisture absorption of PES, we recommend drying the filament for at least 4 hours at 110°C before printing and then using a dry box. Printing this material requires an enclosed and heated chamber with a minimum temperature of 65°C.

Nozzle temperature: 360 - 400°C **Bed temperature:** 140 - 155°C **Print surface:** For printing, it is necessary to apply a special adhesive to the print bed (e.g., MAGIGOO HT) for the perfect first layer. We recommend applying the adhesive in multiple layers.

12.12. PSU (Polysulfone)

PSU, or Polysulfone, is a technical thermoplastic material with an excellent combination of properties that make it suitable for a variety of applications including 3D printing. Its high resistance to temperature, chemicals, and mechanical stresses ensures reliability and performance in demanding conditions.

This material has excellent mechanical properties such as high strength, toughness, and wear resistance, making it an ideal choice for applications requiring durability and reliability.

Advantages

- Excellent temperature resistance (better than Ultem)
- Good chemical resistance
- Resistance to radiation
- Possibility of sterilization
- Self-extinguishing material
- Dimensional stability during printing

Disadvantages

- Higher cost
- Prone to warping
- Requires drying
- Requires adhesives to increase adhesion to the print bed
- Removing supports can be difficult

Typical uses: Sterilised and chemically resistant components in medicine - packaging, surgical instruments, pipes, valves. Also connectors, relays, aircraft and shuttle components.

It is essential to keep the filament dry, otherwise its mechanical properties will deteriorate significantly. Even though the PSU absorbs little moisture, we recommend drying the filament at 110°C for at least 4 hours before printing and then using a drybox. Printing this material requires a closed and heated chamber with a minimum temperature of 65 °C.

Nozzle temperature: 360 - 400°C **Bed temperature:** 140 - 155°C **Print surface:** For printing, it is necessary to apply a special adhesive to the print bed (e.g., MAGIGOO HT) for the perfect first layer. We recommend applying the adhesive in multiple layers.

12.13. PEKK-CF (Polyetherketoneketone)

PEKK-CF is an advanced thermoplastic filament that combines excellent mechanical properties with high-temperature resistance and chemical stability. This material is based on polyetherketoneketone (PEKK) with added carbon fibers, improving its strength and toughness. PEKK-CF filament offers excellent resistance to temperature, chemicals, and wear, making it an ideal choice for applications requiring high performance and reliability. Due to its ability to print complex geometries with high precision, PEKK-CF filament is suitable for industrial and engineering applications such as aerospace components, automotive parts, medical implants, and prototypes.

Advantages

- Extraordinary temperature resistance
- Excellent chemical and mechanical resistance
- High precision and dimensional stability
- Low weight due to the presence of carbon fibers
- Self-extinguishing material
- Excellent insulating properties

Disadvantages

- High cost of filament
- Requires adhesives to increase adhesion to the print bed
- Removing supports can be difficult
- Requires drying

Typical Uses: Lightweight and strong components in aerospace, engine parts and coolers in automotive, medical implants and tools, functional parts of machinery, valves, and pipes.

It is essential to keep the filament dry, otherwise, its mechanical properties will significantly deteriorate. PEKK-CF is extremely hygroscopic and begins to absorb moisture from the air within 10 minutes, affecting the resulting quality and strength. Therefore, we recommend drying the filament for at least 4 hours at 130°C before printing and then using a dry box. Printing this material requires an enclosed and heated chamber with a minimum temperature of 85°C or higher.

Nozzle temperature: 380 - 440°C **Bed temperature:** 135 - 150°C **Print surface:** For printing, it is necessary to apply a special adhesive to the print bed (e.g., MAGIGOO HT) for the perfect first layer. We recommend applying the adhesive in multiple layers.

12.14. PEEK-CF (Polyetheretherketone)

PEEK-CF is an advanced thermoplastic filament with excellent mechanical properties and high-temperature resistance. This material is based on polyetheretherketone (PEEK) with added carbon fibers, improving its strength, toughness, and thermal resistance. PEEK-CF filament is known for its high resistance to temperature, chemicals, and wear, making it an ideal choice for applications requiring extreme operating conditions. Due to its ability to print complex geometries with high precision, PEEK-CF filament is suitable for industrial and engineering applications, including aerospace components, automotive parts, medical implants, and prototyping.

Advantages

- Extraordinary temperature resistance
- Excellent chemical and mechanical resistance
- High precision and dimensional stability
- Low weight due to the presence of carbon fibers
- Self-extinguishing material
- Excellent insulating properties

Disadvantages

- High cost of filament
- Requires adhesives to increase adhesion to the print bed
- Removing supports can be difficult
- Requires drying

Typical Uses: Lightweight and strong components in aerospace, engine parts and coolers in automotive, medical implants and tools, functional parts of machinery, valves, and pipes.

It is essential to keep the filament dry, otherwise, its mechanical properties will significantly deteriorate. PEEK-CF is extremely hygroscopic and begins to absorb moisture from the air within 10 minutes, affecting the resulting quality and strength. Therefore, we recommend drying the filament for at least 4 hours at 130°C before printing and then using a dry box. Printing this material requires an enclosed and heated chamber with a minimum temperature of 85°C or higher.

Nozzle temperature: 380 - 440°C **Bed temperature:** 140 - 155°C **Print surface:** For printing, it is necessary to apply a special adhesive to the print bed (e.g., MAGIGOO HT) for the perfect first layer. We recommend applying the adhesive in multiple layers.

13. Regular Maintenance

Prusa Pro HT90 was engineered for maximum efficiency and reliability. However, it is still a mechanical device with components that require regular maintenance. Please see the information below to learn how to keep your HT90 in the optimal shape.

13.1. Double-Sided Spring Steel Print Sheets

To achieve the best adhesion of the print surface, it needs to be kept clean. The recommended option is high-quality isopropyl alcohol. Drop a small amount of the agent onto a clean paper towel and wipe the print surface. Best results will be achieved when the print sheet is cold. With a hot sheet, there is a risk of burns and also the alcohol will evaporate before it has a chance to clean anything.

The print surface does not need to be cleaned before every print, just be aware of not touching it with your fingers.

Consumable materials such as print sheets are not covered by our warranty unless they arrive damaged or incorrectly manufactured. Print sheets are consumables and the warranty only applies to defects that appear immediately after unpacking.

Double-Sided TEXTURED Print Sheet

- Surface resistant to damage and scratches
- Texture on the surface of the sheet is transferred to the bottom side of the printed object
- FLEX does not require a separation layer (e.g., glue)
- After the print sheet cools down, the print usually detaches itself
- PLA prints with small contact areas may require a brim
- Never clean with acetone

The textured powder-coated surface applied directly to metal allows us to create a print sheet that is highly resistant to damage. If a heated nozzle hits it, the metal is able to quickly dissipate heat. The textured powder coating also gives the bottom surface of the print a unique, interesting texture. The textured surface is able to mask most scratches and similar types of damage caused by various tools. One can only scratch the highest points of the texture, yet this type of damage will not be visible on the print.

Never clean the textured powder surface with acetone! This will cause micro-cracks in the PEI layer, which will eventually lead to a significant deterioration of the surface quality.

Improving the Adhesion

In certain special cases, such as printing a very tall object that touches the print sheet with a very small area, it may be necessary to improve the adhesion. PEI is fortunately a chemically very resistant polymer, so it is possible to apply various substances to improve adhesion without risking damage to the surface. This also applies to various materials whose adhesion to PEI would be very weak under normal circumstances. More information can be found on the website help.prusa3d.com/materials.

13.2. Changing the Print Head

The Prusa Pro HT90 printer is available with two print head configurations. By default, the printer comes with a head capable of printing materials up to 300°C (ASA, PETG, PC, PA). For printing industrial materials such as PEI, PEKK, and PEAK, a different type of print head with a temperature limit of 500°C is required (High-Temp Print Head, marked with a red stripe)

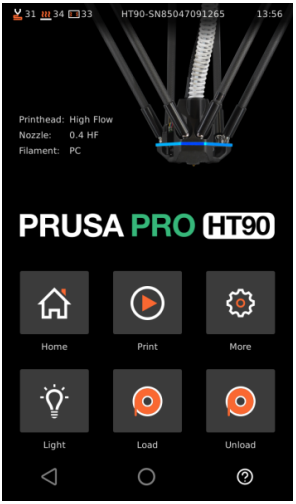
If necessary, changing the print heads is very simple:

1. First, unload the filament from the nozzle and cool the print head to below 50°C.
2. Extend the door to the maximum height.
3. At the top of the printer, behind the HEPA filter, there is a holder for the cable bundle and the air hose. It is fastened with one screw, which should be unscrewed using a 2.5mm hex key.
4. Pull the entire holder downwards.
5. Now, using the 2.5mm hex key, unscrew the screw from the air hose and cable bundle holder.
6. After unscrewing it, split the holder into two halves.
7. The final step is to detach the magnetic arms. Hold the print head with one hand and carefully disconnect the arm from the ball joints on the head with the other hand.
8. This will release the entire print head.
9. Attach the new print head by reversing the steps, following the order 8, 7, 6, 5, 4, and 3.

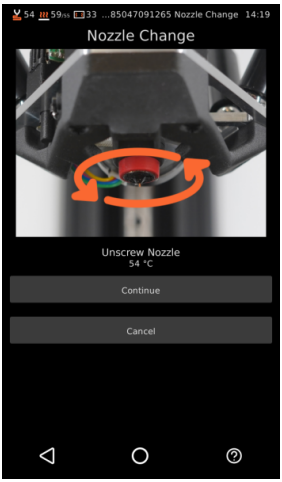
13.3. Nozzle Replacement

The Prusa PRO HT90 uses **quick-swap REV nozzles.** To change a nozzle, use the Wizard in the LCD Menu. From the Home screen, select More, then Nozzle change.

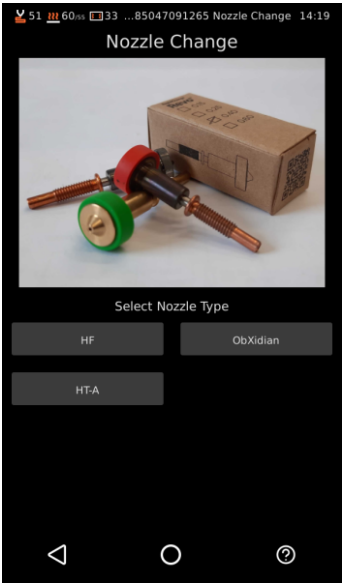
If the nozzle is too hot at this point, you will be presented with a warning. Select Cooldown and wait until the nozzle's temperature reaches safe levels.

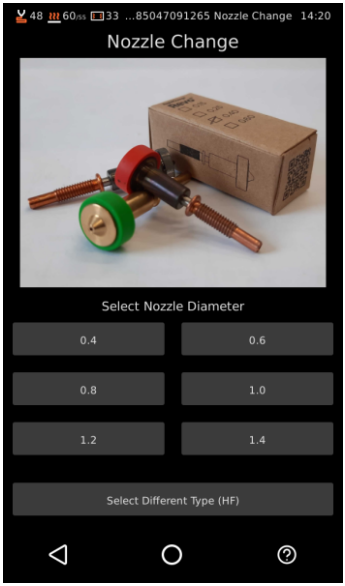


Once the nozzle's temperature is under 60 °C, you will be presented with a screen depicting the direction in which you should unscrew the nozzle. Please keep in mind that the nozzle's thread is longer (see the photo below).



Once the nozzle is out, select Continue on the screen. You will be presented with a menu where you can select which type of nozzle you wish to install. The standard type is the HF (High Flow) nozzle. The ObXidian is ideal for composite materials (e.g., containing carbon fiber) and the HT-A is designed for high-temperature abrasive materials, such as PEI CF, PEKK CF, and similar. Once you select the nozzle type, you will be also required to pick the correct diameter. Different diameters are marked with different colors.
See https://e3d-online.com/blogs/news/revo-nozzle-features for reference.





Finally, screw in the selected nozzle and confirm that the change is done by choosing the Save and Close button on the screen.

13.4. Filter Replacement

The top part of the printer's chamber contains a housing with a HEPA filter. This filter needs to be replaced or cleaned regularly.

1. First, allow the printer to cool down to room temperature to avoid the risk of burns from hot surfaces.
2. In the LCD Menu, select More - Move - Service position. This will move the print head to the service position, providing space for manipulation.
3. If the doors are not already extended all the way up, do so now.
4. Take the front part of the filter cover and gently pull it towards you. Next, remove the filter itself by pulling it down by the side loops. Insert the new filter by following these steps in reverse order.

13.5. Keeping the Printer Clean

After several hours of printing, various types of debris can start to accumulate in the printer chamber - filament pieces, dust, broken support remnants, etc. Always ensure that all parts of the printer are clean. You can use a brush, a small broom, or a vacuum cleaner to remove the debris. Do not clean the printer while it is printing. Additionally, clean the printer only after all parts have cooled down and the printer is turned off.

14. Advanced Hardware Troubleshooting

Due to the length of the articles, it is not possible to include detailed troubleshooting guides in this handbook. If the Prusa Pro HT90 encounters a problem, an error screen with a brief recommendation on how to proceed will be displayed. Detailed guides for troubleshooting component replacements and advanced hardware issues can be found online at help.prusa3d.com, or you can contact our technical support.

15. Print Troubleshooting

If you feel that your prints could look better or if there are noticeable print defects (layer shifting, blurred details, "ghosting," etc.), it is necessary to identify and resolve the cause of the problem. Our website help.prusa3d.com offers troubleshooting guides for 3D print quality issues, including images and specific advice for various printer models.