Please always refer to the www.prusa3d.com/drivers/ for an updated version of this 3D printing handbook (PDF download).

Failure to read the handbook may lead to personal injury, inferior results, or damage to the 3D printer. Always ensure that anyone who operates the 3D printer knows and understands the contents of the Manual/Handbook. We cannot control the conditions in which you assemble the Original Prusa MINI. For this, and other reasons, we do not assume responsibility, and expressly disclaim liability for loss, injuries, damage, or expense arising out of, or in any way connected with, the assembly, handling, storage, use or disposal of the product. The information in this handbook is provided without any warranty, expressed or implied, regarding its correctness.

Translated versions of the handbook will be available at:

Czech: www.prusa3d.cz/ovladace/
French: www.prusa3d.fr/drivers/
German: www.prusa3d.de/treiber/
Polish: prusa3d.pl/sterowniki/
Italian: www.prusa3d.it/driver/
Spanish: www.prusa3d.es/drivers-y-manuales/

### QUICK GUIDE TO THE FIRST PRINT

1. Read chapter **6 Your First Print** carefully
2. Place the assembled printer on a flat and stable surface
3. Start the printer, insert the enclosed USB drive and **update the firmware**
4. Follow the on-screen instructions to calibrate the printer
5. Print a sample object

---

**Important notice, tip, hint or information that helps you print with ease.**

Read carefully! This part of the text has the greatest importance - either for user safety or for proper printer service.

This symbol indicates text related to a printer kit only.

### How to contact Prusa Research tech support:

First, check the last chapters of this handbook for troubleshooting guides, or visit help.prusa3d.com for a complete list of common issues, print quality troubleshooting and other guides. If your problem is not listed there, or the solution does not work, please send an e-mail to support@prusa3d.com or use the Live chat at shop.prusa3d.com. Try to explain your problem as thoroughly as possible.
About the author

Josef Prusa (born Feb 23rd, 1990) became interested in the 3D printing phenomenon before joining Prague's University of Economics in 2009 - at first, it was a hobby, a new technology open to changes and improvements. The hobby soon became a passion and Josef grew into one of the leading developers of Adrien Bowyer's international, open-source, RepRap project. Today, you can see the Prusa design in different versions all around the world, it is one of the most popular printers and thanks to it, knowledge about the 3D printing technology significantly increased among the public.

All 3D printers by Josef Prusa are open-source and based on the RepRap philosophy - you can use your 3D printer to produce parts for building another 3D printer.

The lineup of Original Prusa devices is constantly expanded with new machines and upgrades. The main goal is to make technology more accessible and understandable to all users.

Josef Prusa also organizes workshops for the public, participates in professional conferences dedicated to the popularization of 3D printing. For example, he lectured at the TEDx conference in Prague and Vienna, at World Maker Faire in New York, Maker Faire in Rome or at the Open Hardware Summit hosted by MIT. Josef also teaches Arduino at Charles University and was also a lecturer at the Academy of Arts in Prague.

In his own words, he imagines 3D printers will be available in every home in the not-too-distant future. “If anything is needed, you can simply print it. In this field, you just push the boundaries every day... We're glad you're part of it with us!”
# Table of contents

- **About the author**  
  - 3
- **Table of contents**  
  - 4
- **Product details**  
  - 6
  - Warranty  
  - Licensing  
  - 6
- **1 Introduction**  
  - 7
- **2 Glossary**  
  - 8
- **3 Package contents - Accessories (bundled / optional)**  
  - 9
- **4 Handling / Transporting the printer**  
  - 9
- **5 Quick-Start Info**  
  - 10
  - 5.1 Disconnecting the power supply  
  - 11
  - 5.2 How to contact tech support  
  - 11
  - 5.3 Unpacking, assembly and first start  
  - 11
- **6 Your first print**  
  - 12
  - 6.1 Basic controls  
  - 12
  - 6.2 Updating the firmware  
  - 13
  - 6.3 Preparing flexible steel sheets  
  - 13
  - 6.4 Calibration Wizard  
  - 14
  - 6.5 Selftest  
  - 14
  - 6.6 Loading and unloading the filament  
  - 15
  - 6.7 First Layer Calibration  
  - 16
  - 6.8 Checklist  
  - 18
  - 6.9 Starting the first print  
  - 18
  - 6.10 Removing the print  
  - 18
  - 6.11 After-print checks and actions  
  - 19
- **7 Advanced features**  
  - 19
  - 7.1 Menu structure  
  - 19
  - 7.2 Mesh Bed Leveling explained  
  - 20
  - 7.3 Factory reset  
  - 21
  - 7.4 Network connections  
  - 21
- **8 Printing your own models**  
  - 21
  - 8.1 Obtaining a printable model  
  - 21
  - 8.2 Creating your own models  
  - 23
  - 8.3 What is a G-code file?  
  - 23
  - 8.4 PrusaSlicer  
  - 23
  - 8.5 Importing objects into PrusaSlicer  
  - 24
  - 8.6 Using supports  
  - 25
  - 8.7 Print speed vs Print quality  
  - 26
  - 8.8 Infill  
  - 27
  - 8.9 Brim  
  - 27
  - 8.10 Large object printing  
  - 28
  - 8.11 Print multi-colored objects  
  - 29
8.12 Slicing, exporting

9 Material guide
  9.1 PLA
  9.2 PET/PETG
  9.3 ASA/ABS
  9.4 Flex

10 Regular Maintenance
  10.1 Steel sheets
  10.2 Spring steel sheet with TEXTURED double-sided PEI
  10.3 Spring steel sheet with SMOOTH double-sided PEI
  10.4 Increasing the adhesion
  10.5 Bearings
  10.6 Fans
  10.7 Extruder drive gear
  10.8 PTFE tubes
  10.9 Filament sensor (optional accessory)
      10.9.1 Running out of filament
      10.9.2 False sensor readings and debugging
  10.10 Nozzle cleaning
  10.11 Replacing / changing the nozzle
  10.12 Flashing unsigned firmware

11 FAQ - common issues and how to solve them
  11.1 Skewed X-axis
  11.2 Printer can’t read the USB drive
  11.3 Loose X- and/or Y-axis belts
  11.4 Filament stuck in the PTFE tube
  11.5 X-Axis homing fails
  11.6 Heating errors
  11.7 Fan errors

12 Advanced hardware troubleshooting

13 Print quality troubleshooting
Product details

Title: Original Prusa MINI
Manufacturer: Prusa Research s.r.o., Partyzánská 188/7A, Prague, 170 00, Czech Republic
Contacts: phone +420 222 263 718, e-mail: info@prusa3d.com
EEE group: 3 (IT and/or telecommunication equipment), Device use: indoor only
Power supply: AC/DC SWITCHING ADAPTOR, INPUT 100-240 VAC, 50/60 Hz, 2.0 A / OUTPUT 24V, 6.67A, 160 W MAX, complies with Part 15 of the FCC Rules
Working temperature range: 18 °C - 38 °C, indoor use only
Working humidity: 85 % or less
Compatible filaments: 1.75 mm only
Package weight (gross / net): 7.6 kg / 4.85 kg
The serial number is located on the printer frame and also on the packaging.

Warranty

The pre-assembled Original Prusa MINI 3D printer is covered by a 24-month warranty for end customers in the EU and 12 months for businesses and non-EU customers. Wear and tear category components and consumables are excluded from this warranty.

The warranty period begins with the day when the customer receives the goods. Neither the Seller's liability for defective products nor the quality warranty applies to printers or parts damaged by handling, which is in conflict with the instructions and recommendations in the official manuals and guides, or improper handling caused by unofficial modifications, such as modifications to hardware or software.

Licensing

The Original Prusa MINI printer is a part of the RepRap project, the first open-source 3D printer project free to use under a GNU GPL v3 license (www.gnu.org/licenses/gpl-3.0.en.html). If you improve or alter any part of, a printer, and you are willing to sell it, then you have to publish the source code under the same license. All 3D-printed elements of the printer that can be improved upon can be found at www.prusa3d.com/prusa-i3-printable-parts.

Please be very cautious during any interaction with the printer. This printer is an electrical device with moving and high-temperature parts.

1. The device is for indoor use only. Do not expose the printer to rain or snow. Always keep the printer in a dry environment at a minimum distance of 30 cm from other objects. Make sure none of the vents/fan outlets are blocked.
2. Always place the printer on a stable place, where it cannot fall or tip over.
3. The printer supply is household power outlet 230VAC, 50 Hz or 110VAC / 60 Hz. Never connect the printer to a different power supply; it may cause malfunction or damage to the printer.
4. Place the power cord so you cannot stumble over it, step on it, or otherwise expose it to any potential damage. Also, make sure that the power cord is not mechanically or otherwise damaged. If so, stop using the damaged power cord immediately and replace it.
5. When you disconnect the power cord from the socket, pull the plug rather than the cord to reduce the risk of damage to the plug or to the AC outlet. The plug that goes into the printer has a safety locking mechanism: first pull the cover of the plug to unlock it, then disconnect the plug from the printer.
6. Never disassemble the printer’s power supply; it does not contain any parts that could be repaired by an unskilled worker. All repairs must be performed by a qualified technician.
7. Do not reach inside the printer while it is still in operation. An injury may be caused by its moving parts, heated elements or electrical current.
8. Prevent children from unsupervised access to the printer even when the printer is not printing.
9. Do not leave the printer unattended while it's still on!
1 Introduction

Thank you for purchasing the **Original Prusa MINI 3D printer** from Josef Prusa! Your purchase supports us in the further development of the MINI and other devices. Original Prusa MINI is our compact, smart and affordable 3D printer, which was designed as a 3D printing workhorse and a great starter printer. We hope you will enjoy using it! Please, read the handbook carefully, since all chapters contain valuable info regarding the operation of the equipment, getting-started instructions, 3D printing tips and everything about regular maintenance.

Please check the [prusa3d.com/drivers](http://prusa3d.com/drivers) page for the updated version of this 3D printing handbook (PDF download).

In case of any printer-related problem, do not hesitate to contact us at [info@prusa3d.com](mailto:info@prusa3d.com). We are glad to receive your valuable comments and tips. You can also visit our knowledge base at [help.prusa3d.com](http://help.prusa3d.com) and forums at [forum.prusaprinters.org](http://forum.prusaprinters.org).
1. **Spring steel sheet** - comes in two variants, smooth and textured. This is where the printed object appears. Please read the maintenance instructions carefully to ensure optimal conditions for 3D printing.

2. **Heatbed** - located directly under the spring steel sheet. Heatbed ensures good adhesion of the printed object. **Warning: do not touch the heatbed when it’s hot!**

3. **Y-Axis** - the general name for the **entire heatbed assembly** (smooth rods, belt, heatbed…).

4. **Knob** - main control of the Original Prusa MINI 3D printer: rotate left/right to scroll through options, press the knob to confirm your selection. There’s a reset switch directly next to the knob. Press it to perform a hard reset.

5. **LCD screen** - the display is used for the configuration of the 3D printer. Use the knob to scroll through the menus and confirm your selection with the press of the knob.

6. **Power switch** - when the printer is not in use, turn it off using the power switch.

7. **USB port** - compatible with USB 2.0/3.0 flash drives (FAT32 filesystem). Used for reading print files (G-Codes) from the inserted drive and flashing the firmware.

8. **Main PTFE tube** - leads the filament strand from the extruder into the print head / nozzle. Inspect it from time to time to make sure there is no debris inside that would prevent the filament strand from reaching the nozzle.

9. **Tube fitting** - on both ends of the main PTFE tube. In case filament gets stuck in the PTFE tube, use a spanner (included with the printer) to remove the fittings and the PTFE tube to get access to the filament strand.

10. **Print fan** - cools the printed object, improves print quality. Comes with RPM monitoring.

11. **Print head** - Lightweight print head consisting of the hotend (printing nozzle), M.I.N.D.A. probe and two fans. The heaviest parts of the extruder were moved onto the Z-axis tower to improve print quality.
12. **Nozzle** - 0.4mm nozzle, can be replaced with e.g. 0.25mm or 0.6mm E3D-compatible nozzles to achieve various effects.

13. **Z-Axis** - the entire vertical axis assembly (smooth rods, Z-axis motor…).

14. **Extruder / Extruder motor** - as opposed to 3D printers like MK2 or MK3S, the extruder motor is not moving along the X-axis. Instead, it’s fixed on the side and pushes filament through the PTFE tube towards the print head.

15. **X-Axis** - allows the movement of the belt-driven print head from left to right.

16. **Spool holder** - should be placed near the printer, so the filament strand can enter the PTFE tube from below. Filament must not bend at an acute angle.

### 3 Package contents - Accessories (bundled / optional)

Your Original Prusa MINI comes bundled with:

- **USB drive with firmware files**
- **Allen key**
- **Spanner**
- **Surface cleaning wipes** (infused with isopropyl alcohol)
- **Acupuncture needle**

These are the basic tools required for correct assembly and basic maintenance. However, we recommend getting some extra items to improve your 3D printing experience.

- **Side cutting pliers** - for filament cutting
- **Isopropyl alcohol, Windex, paper wipes** - for print sheet maintenance
- **Metal spatula** - for easier removal of prints stuck to the steel sheet

### 4 Handling / Transporting the printer

When transporting the Original Prusa MINI, use the recommended way of holding the device. You can either grab it by the Z-axis motor at the top, or by the Z-Axis assembly. See photos below for reference. **Never hold it by the cables or PTFE tubes!**
5 Quick-Start Info

Original Prusa MINI 3D Printer

✅ Place in a safe, dry and horizontally stable location (e.g. workbench)
✅ Place the power brick where nobody can trip over the power cord
✅ Leave at least 25 cm (1 ft.) around the printer

**Update the firmware by inserting the bundled USB drive and following the on-screen instructions**

❗ Do not place near running water or outside
❗ Do not touch / move the printer during operation
❗ Do not pull the power plug from the printer directly. It has a locking mechanism - pull the cover of the plug back to unlock it, then disconnect the plug

Smooth and Textured steel sheets

✅ Read maintenance instructions in chapter 10
✅ Regular maintenance
✅ Use isopropyl alcohol to clean (degrease) the sheet
✅ Calibrate Live Z when you switch sheets
✅ Wipe the sheet with IPA-infused towel before the first print

❗ Do not wash with running water
❗ Do not peel off the PEI sticker
❗ Do not use acetone to clean the textured sheet!

Spool holder and filament spools

✅ Place in a location where the spool can unwind without resistance
✅ Use correct temperatures for the selected material
✅ Store filaments in a dry location
✅ The filament should enter the extruder from below
✅ Filament end should have a sharp tip

❗ Do not print with tangled filament
❗ Do not print with incorrect temperature settings
❗ Some materials (ABS/ASA) can produce odors during printing - keep the room ventilated
5.1 Disconnecting the power supply

The **power supply plug** that goes in your printer has a locking mechanism - **do not pull it with force! Do not pull the cable!**

First, unlock the mechanism by pulling the cover of the plug away from the printer. Once the mechanism is unlocked, you can pull the plug out of the printer.

5.2 How to contact tech support

If you encounter an issue with your MINI 3D printer, please check the last pages of this handbook for **general troubleshooting guides or visit our knowledge base at help.prusa3d.com**. However, in case the suggested solutions don’t work, please contact our tech support via e-mail at `support@prusa3d.com` or via the Live chat at `shop.prusa3d.com` - the chat is in the lower right corner.

5.3 Unpacking, assembly and first start

The Original Prusa MINI 3D printer comes separated into three parts. **Please flip this handbook to get to the Assembly instructions and pre-flight check.** Then return here to continue with the setup process.

If you’re viewing this handbook online, please head to `manual.prusa3d.com` and open the online version of the assembly manual.

**Before you start using the device, you need to flash the firmware first - please insert the bundled USB drive that came with your 3D printer and follow the on-screen instructions to flash the firmware. You can also check [www.prusa3d.com/drivers](http://www.prusa3d.com/drivers)** for the latest firmware version.
6 Your first print

Preparing the printer for the first print takes about 30-50 minutes including assembly. Please pay attention to the following steps to ensure the machine will operate as intended.

In this chapter, you will learn how to:

- Control the printer
- Update the firmware
- Prepare the print surface for the first print
- Perform the initial calibration
- Load filament
- Start the first print

6.1 Basic controls

1 - You can configure and control the entire device with a single control element: a rotational knob, which can be pressed to confirm the selection.

2 - The reset button is placed to the left from the control knob. Pressing the reset button equates to quickly toggling the power switch. It is useful when the printer exhibits unexpected behavior or you see a failed print that requires to be stopped immediately.
6.2 Updating the firmware

The printer is shipped without firmware - the first step is flashing the firmware from a USB drive. You can simply take the USB drive that came with your printer, insert it into the USB slot on the side and start (or restart) the printer. However, we recommend to check www.prusa3d.com/drivers for the latest version to ensure optimal printing experience.

Follow these steps:

1. Download the correct firmware file from www.prusa3d.com/drivers (ZIP archive) and unpack it
2. Place the .BBF file on a FAT32-formatted USB drive - you can use the one bundled with your MINI
3. Insert the USB drive into the printer’s USB port
4. Restart the device using the restart button next to the Knob
5. The update process should start automatically
6. Wait until the process is complete

6.3 Preparing flexible steel sheets

Please follow the instructions for the sheet of your choice (see below) and prepare it for the first print.

The heatbed has embedded high curie temperature magnets inside, which hold removable spring steel sheets in place. There are two pins at the end of the heatbed that will align perfectly with cut out edges of the spring steel sheets. These pins aren’t meant to fit into the two
holes on the other side of sheet! Make sure the bed is clean and there isn’t any debris on it before you put on the steel sheet. Never print directly on the heatbed.

Unpack the IPA-infused cleaning wipe and clean the steel sheet before you start.

Heatbed and powder-coated steel sheet surface

To achieve the best adhesion of the printed object, it is important to keep the surface clean. Please see the information in the chapter 10 Regular Maintenance for more information about print surface preparation, recommended cleaning products and warranty info.

6.4 Calibration Wizard

Once the printer starts for the first time, the Wizard will show up. It will guide you through all the tests and calibrations you need to perform in order to start printing.

The Wizard can be also started manually from LCD menu - Calibration - Wizard. Do not forget to read chapter Flexible steel sheets before running the Wizard.

The Calibration Wizard consists of Selftest, Mesh Bed Leveling and First Layer Calibration.

6.5 Selftest

The Selftest is a diagnostic tool and consists of a series of tests. They are designed to reveal the most common issues, such as incorrect wiring. The progress and results of each step are displayed on the LCD. In case any issues are found, the selftest is interrupted and the cause of the error is shown to guide users in troubleshooting.
The selftest consists of:

- **Extruder and print fan test**
  - Using the RPM monitoring
- **X,Y, and Z-axis test**
  - Checks the length and free movement of each axis
  - This also verifies proper stepper motor wiring
- **Heatbed and hotend proper wiring**
  - This also verifies correct thermistor wiring

6.6 Loading and unloading the filament

Once the Selftest is finished, you can load a strand of filament. Every time you load a new filament, **make sure the tip is sharp** and there are no bumps or other shapes that would prevent the filament from entering the PTFE tube. Filament spools should be kept around the same level as the printer, so the filament strand can enter the PTFE tube without excessive bending.

**Loading filament**

1. Cut the end of the filament strand to create a sharp tip. Insert the filament into the extruder PTFE tube (right side of the device), or into the short PTFE tube of the filament sensor (optional accessory).
2. Choose **LCD Menu - Filament - Load filament** and press the button to confirm.
3. The **preheat menu** will be automatically displayed. Select the type of filament you wish to load and confirm the selection by pressing the control knob.
4. Wait for the nozzle to reach the **target preheat temperature**.
5. Press the control knob to start loading and **push the filament into the extruder** until the extruder drive gears grab the filament.
6. The filament is then loaded into the print head by the extruder gears automatically. The printer will ask you whether some plastic has been extruded from the nozzle. In case it wasn’t, choose “No” and the printer will attempt to extrude more plastic.
The printer keeps track of the currently loaded filament, even when powered off. You can check the type of the currently loaded filament in the bottom right corner of the LCD Menu.

For the next step, keep the filament loaded. If, for some reason, you need to change the filament before First Layer Calibration (or at any other time), follow these instructions:

**Unloading filament**
1. Choose LCD Menu - Filament - Unload filament.
2. The printer will automatically preheat to the correct temperature, the beeper will notify you when the preheating finishes.
3. Press the control knob to start unloading.
4. Once the extruder gears stop unloading the filament, pull the filament out from the PTFE tube manually.

### 6.7 First Layer Calibration

This process will calibrate the distance between the tip of the nozzle and the probe. You can launch the calibration at any time from LCD Menu - Calibration - First layer cal.

⚠️ Make sure the steel sheet is clean and there’s no grease on it, even fingerprints can cause an issue. Use the IPA-infused cleaning wipe that came with your printer to clean it before you start.

Once you start the First Layer Calibration, the printer will ask you whether filament is loaded. Once it is, the printer will probe the bed with Mesh Bed Leveling routine (see chapter 7.2 Mesh Bed Leveling for more information) and start printing a zig-zag pattern on the print surface. The nozzle will be at the height defined by the M.I.N.D.A probe setting. It must not by any means touch the printing surface.

![Diagram of nozzle height tuning](image)

*How to tune the nozzle height live during the test print*

Observe the line which is being extruded on the print surface. A new menu will automatically show up - use it to tune the nozzle height in real time by turning the knob. The aim is to adjust the nozzle height until the extruded plastic sticks nicely to the bed and you can see that it is being slightly squished.
Make sure the testing pattern looks like the one on the left or middle picture. The picture on the right displays both serious issues: nozzle too high (bent lines, shifted corners) and nozzle too low (plastic not extruded)

What to check:
- The top of the extruded plastic is slightly squished
- The corners of the zig-zag line are sharp and do not lift from the sheet
- The lines are straight and stay on the sheet firmly
- The ending box has an even surface, there are no waves or gaps

Below, you can see a close up of a print displaying quality issues caused by an incorrect nozzle height. The picture in the middle is the optimal quality.

The middle picture shows a correctly calibrated first layer
6.8 Checklist

If you followed the steps described in the previous chapter, you are now ready to start your first print. Before you start, let’s go through the final checklist:

1. MINI is fully assembled and placed in a suitable location
2. The firmware is updated to the latest version
3. All of the calibration procedures were successful
4. A clean print sheet cleaned with IPA-infused cleaning wipe is on the heatbed
5. Filament is loaded

6.9 Starting the first print

Once everything is ready, you can navigate to the PRINT menu (available only when a USB drive is inserted). Use the knob to select a sample object from the list and observe the printer. We recommend choosing the PRUSA logo, because it gives you a great overview of how well you tweaked the first layer during First Layer Calibration.

![Print Preview](image)

*You should aim for this level of quality with your first layer*

The nozzle will preheat to 170 °C independently on the selected filament - this is to prevent oozing of the filament during the initial phase, which is Mesh Bed Leveling. Once MBL is done, the printer will preheat the nozzle to the correct printing temperature and start with the initial purging line at the edge of the print bed. Then, it will start printing the skirt around the object and the object itself.

**Pay close attention to the quality of the first layer.** Extruded plastic should stick to the sheet nicely. If, for some reason, the nozzle is too high, either try to lower it with the Live Adjust Z function (available through the Tune menu during an active print job), or stop the print completely, clean the sheet and re-run First Layer Calibration from the Calibration menu.

6.10 Removing the print

Once the print finishes, wait for the heatbed to cool down. Depending on the material and steel sheet used, your print might automatically detach as the sheet cools down. If that’s not the case, lift the steel sheet and bend it both inwards and outwards, rotate it 90 degrees and
repeat the bending. **Don’t forget to remove all leftover pieces of filament**, such as the intro purge line, the skirt and the support base.

Try to **avoid touching the steel sheet surface with your fingers** - fingerprints can decrease the adhesion of the next print.

6.11 After-print checks and actions

Once the sample print is removed from the steel sheet, **inspect it closely and check for possible issues with quality**. Please note that objects printed using FDM/FFF methods are never perfectly smooth - **layers are usually visible**. There are ways to make 3D printed objects smoother, though. Check our blog at [blog.prusaprinters.org](http://blog.prusaprinters.org) to learn more about post-processing.

In case your print has **actual quality issues** (missing parts, shifted layers, under-extruded sections), please check our Print Quality Troubleshooting at the end of this handbook.

Make sure the steel sheet is clean and put it back onto the heatbed. You can leave the filament loaded in the nozzle. If you don’t plan to print another object, **turn the printer off using the switch on the side**.

7 Advanced features

7.1 Menu structure

- Print
- Preheat
  - PLA
  - PETG
  - ASA
  - FLEX
  - Cooldown
- Filament
Mesh bed leveling can be found in **LCD Menu - Calibration**. This procedure is performed before every print. It is also the same procedure that also happens at the start of First Layer Calibration (see next chapter)

The M.I.N.D.A probe goes through a **4x4 grid pattern spread across the sheet** (whether it is a powder-coated or smooth PEI does not matter) and **measures the distance to the sheet**. These points are interpolated and used to create a virtual mesh of the bed. If the bed is slightly warped, the probe will still precisely follow the surface during printing according to its measured mesh, improving the quality of the first layer.
7.3 Factory reset

A factory reset can be performed through **LCD Menu - Settings - Factory Reset**. This will reset all data to default values.

7.4 Network connections

Original Prusa MINI has an ethernet (RJ45) port onboard. You can use this port to connect your printer to your network.

Please note that network features may not be fully available in the default firmware. Keep checking our web [www.prusa3d.com](http://www.prusa3d.com) or social media profiles where we publish information about updated firmware versions.

Wi-Fi module (an optional accessory) is not supported in default firmware. Information about Wi-Fi module availability will be published at [www.prusa3d.com](http://www.prusa3d.com) or [blog.prusaprinters.org](http://blog.prusaprinters.org).

8 Printing your own models

Once the printer is fully calibrated and the sample models are looking good, you will probably want to **print your own model**. You have a number of options - read the following chapters to learn more.

8.1 Obtaining a printable model

The easiest way to get started with 3D printing is to **find models on the internet** - they are usually in .3mf, .stl or .obj formats. Fortunately, 3D printing has many fans, so there are sites
from which you can download a large variety of ready-made 3D models for free. These range from a simple playing dice to a detailed figure from a favorite movie, game or TV series. You can also download mechanical parts, RC model accessories, household items and much more. You can visit our own community website PrusaPrinters.org - it's our brand new community hub for all Prusa 3D printer owners! It has a ton of cool features and plenty of amazing models for you to print.

PrusaPrinters.org

3D models are very often free to download under the Creative Commons - Attribution - Non-Commercial (Models not to be used commercially, you must always include the name of the author) or for a small fee. We have selected the most interesting sites:

- www.prusaprinters.org
- www.thingiverse.com
- www.myminifactory.com
- pinshape.com
- www.youmagine.com
- www.shapeways.com
- www.gambody.com

However, models in .stl, .obj or similar formats cannot be 3D printed directly. First, they need to be “sliced” (converted) into a G-code file, which is then placed onto a USB drive. Connect the USB drive with a sliced project into the printer and select the model to print from the Print menu. Please see the chapters What is a G-code file? and PrusaSlicer for more information.
8.2 Creating your own models

To create a **3D model**, you need a dedicated program - a **3D editor**. There are a number of various 3D programs and your choice will usually depend on what type of model you wish to create.

The **easiest place to start** is **TinkerCad** ([www.tinkercad.com](http://www.tinkercad.com)) - it’s an online editor that runs in your browser’s window, so no installation is required. It’s free, easy to navigate, and you will find plenty of tutorials online. However, TinkerCad is mostly focused on creating **less-detailed and larger** (e.g. mechanical) parts, ideal for FFF/FDM printing - your MINI can take care of them easily. Another popular tool is **Autodesk Fusion 360** ([www.autodesk.com/products/fusion-360](http://www.autodesk.com/products/fusion-360)) for PC, Mac, and iPad. The website provides a quick guide along with detailed video tutorials so it is an ideal choice both for beginning enthusiasts and professionals.

8.3 What is a G-code file?

When you download 3D models from the internet or create your own models, you will need to convert the 3D model (.stl, .obj, .3mf and similar formats) into a set of instructions for the printer called **G-code**. G-code is a file format readable by 3D printers. The file contains information such as nozzle movement, the amount of filament to extrude, temperature settings or fan speeds.

There are dozens of slicers available, each with their own advantages and disadvantages. The three most commonly used slicers among the Prusa printer owners are:

- PrusaSlicer
- Cura
- Simplify3D

8.4 PrusaSlicer

As the name suggests, PrusaSlicer is our own in house developed slicer based on the open-source project **Slic3r**. It has many useful features such as:

- Ready to print, auto-updating profiles for over 3 dozen filaments
- Variable layer height
- Custom supports and modifier meshes
- Differential print settings

You can always download the latest stable version from [www.prusa3d.com/drivers](http://www.prusa3d.com/drivers). Development alpha/beta versions are available at [github.com/prusa3d/PrusaSlicer](https://github.com/prusa3d/PrusaSlicer).
1. The Add button loads models into PrusaSlicer
2. Delete and Delete All buttons remove the model(s) from PrusaSlicer
3. Opens the detailed settings of print, filament, and printer
4. Move, scale, rotate, Place on Face and cut tools
5. Slice and generate .gcode button
6. Quality / Speed setting of a print
7. Material selection
8. Printer selection
9. Right-click on model opens a context menu
10. Switch between 3D editor and layers preview
11. Model preview
12. Switch between Simple / Advanced / Expert mode

8.5 Importing objects into PrusaSlicer

Once you start PrusaSlicer, select the Original Prusa MINI in the Printer menu on the right side of the window. If it's not listed, you can add it either by selecting Add a new printer option in the same menu or by going to Configuration -> Configuration Wizard. Then, select the layer height, infill and the material you want to use. Please note that built-in filament profiles have pre-defined and tested settings. If you select a wrong profile, it may affect your print due to incompatible settings.
PrusaSlicer allows you to import **STL, OBJ, AMF, and 3MF** object files - these are the most common types of 3D files you can find on the internet. You can either drag-and-drop them in the *Plater* window, or you can use the *Add…* button in the top toolbar.

Next, use the tools in the left toolbar to **Move, Scale and Rotate** the model. If the object is blue, it means it’s too big for the printing platform. See the next chapter for more details. **Every object is different**, so there’s no default orientation that would work for every object. The rule of thumb here is to select the largest flat surface of the object at the base - the bigger the base, the better the adhesion. Select a perfectly flat side of the model for the best result.

### 8.6 Using supports

**Support material is a 3D-printed supporting structure** (similar to a scaffolding) used to print complex objects correctly - you can remove the support material after the print job is finished.

You can find many models that can be printed without supports - just place them on the print bed the right way and you can start printing. However, **not all objects can be printed without supports**.

If you print an object with a gradient lower than 45° (a steep side), the material overhang will prevent the object from being printed correctly. You can even find objects that would require the printer to start printing mid-air - which is impossible. This is where **supports need to be used**. PrusaSlicer has this function built-in.

**Automatically generated supports in PrusaSlicer**

You can select from three options:

- **Support on build plate only** - generate supports only around the object
- **For support enforcers only** - generate supports only where enforcers are placed
- **Everywhere** - generate supports everywhere
The default values work in a majority of scenarios. However, if you need to **tweak where the supports should be generated**, follow these steps: Go to the **Print Settings** tab and click the **Support Material** option in the left column.

- Check the **Generate support material** box.
- **Overhang threshold** lets you set the minimal angle for printing the support material. Setting this item to zero lets the printer detect problematic parts automatically and print support where it is needed.
- **Enforce support** option is used mostly with small models or models with a small base to prevent the object from breaking or tearing out from the bed.

Areas above supports usually have a less-than-ideal surface quality. Before you decide to use supports, try to move/rotate the object to find an orientation that will require less or even no supports.

### 8.7 Print speed vs Print quality

Printing a small object takes a few minutes, but printing larger models are more time consuming - there are prints that will take dozens of hours to finish. **There are various elements affecting the print time.** The first way to affect the printing speed is to **change layer height** in PrusaSlicer - the upper right window shows the Print settings option. The default setting is 0.15 mm **QUALITY**, but you can speed up the printer by choosing the 0.20 mm (SPEED) option - or even 0.25 mm DRAFT quality. **Raising the layer height will result in less detailed models**
with more prominent layers. If you prefer quality over speed, choose a 0.10 mm (DETAIL) option. Printing time will increase noticeably, however, the models will look much better. Increasing the printing speed may result in a less-detailed model.

Our most used profiles **0.15 mm** and **0.20 mm** come in two different versions.

- **Quality** - slower perimeters and infill, gives you better surface quality
- **Speed** - faster perimeters and infill without much sacrifice of surface quality

---

### Print quality vs print time

The speed can also be changed while printing. Go to **LCD Menu - Tune - Speed**. By turning the control knob clockwise you can increase the print speed up to **150 %**. Turning the knob counterclockwise decreases the print speed - the minimum speed is **50%**. Watch the results of increased or decreased speed on the printed model and adjust the speed accordingly. Keep in mind that acceleration is unaffected by this value, so the change in speed is not directly proportional to the change in print time.

#### 8.8 Infill

Infill is another parameter that affects the printed object - mainly the speed of printing, structural strength and even the look of the printed object.

Objects printed with the FFF/FDM method are not solid. Instead, you can find a certain 3D pattern inside these objects. The patterns can range from a simple grid or honeycomb to more complex shapes. The purpose of the infill remains the same: give the object a certain level of structural strength. Many models can be printed with 10-15% infill, but if you require the model to be stronger (e.g. a structural part), select a denser infill.

#### 8.9 Brim

A brim is used to increase adhesion to the print bed and decrease the possibility of warping. This is done by printing extra outlines of the first layer and it's especially useful when the footprint of the first layer is very small. You can turn it on in PrusaSlicer by ticking a
checkbox in the right panel. When the print is finished, it’s usually very easy to remove brim with hand. Alternatively, you can use a deburring tool or a scalpel to remove the brim.

8.10 Large object printing

Original Prusa MINI has a smaller bed compared to the Original Prusa i3 MK3S, however, with some clever tricks, you can print massive objects with the MINI as well. Do not let the smaller print bed stop you - check our blog at prusaprinters.org to learn how to assemble large models from several smaller parts.

A special case is when you want to print objects larger than the heatbed. The first option is to resize the object to a printable size. Select the **Scale tool** from the left menu (or press the **S** key). Then use the 3D gizmo to change the scale of the object. If you want to scale the model uniformly, drag the scale gizmo by one of its corners. You can also change the size of a model along one of the axes by dragging the corresponding gizmo handles. Alternatively, you can use the object manipulation window in the bottom right corner to type in an exact scale value.

![Changing the scale of a printed object](image)

If you need to print an object that does not fit the printer at its original scale, you have to cut the object into smaller pieces. Select the **Cut tool** from the left menu (or press the **C** key). Position the cut plane to the desired position or enter a precise height into the Cut dialog window. You can decide whether you want to keep the upper, lower or both object parts.
8.11 Print multi-colored objects

There is a simple way to create layer-based multi-colored 3D prints directly in PrusaSlicer - see the instructions below.

1. Switch to the layer preview using the button in the lower-left corner
2. Using the slider on the right side, select a layer where the color change should occur
3. Click on the orange plus icon
4. A preview is instantly displayed. You can remove the color change by clicking on the grey cross button that is now displayed instead of the orange plus button
5. Export the G-code and you're ready to print!
Setting up color change in PrusaSlicer

8.12 Slicing, exporting

One of the most important parts of the slicing process is the inspection of the sliced object in the Preview window. Use the slider on the right side of the window to go through all the layers of the sliced object. This will help you to uncover possible problematic spots - e.g. when the bottom of the object doesn’t sit directly on the bed or when there are supports missing and some parts of the object start mid-air.

Before you export the model as a G-code and place it onto a USB drive, always make sure to inspect the object in the preview mode. It’s the best way to prevent possible printing issues.
9 Material guide

The Original Prusa MINI can print with many popular materials. Materials vary in both mechanical and optical properties as well as in the ease of use. If you’re new to 3D printing, your first filament should definitely be PLA. Only when you are limited by some of PLA’s disadvantages, it’s time to try other filaments like PETG or ASA.

9.1 PLA

PLA is the most commonly used material for 3D printing. It is biodegradable, easy to print with, and PLA prints are very hard. It’s the perfect choice for printing large objects due to low thermal expansion (prints do not warp on the heatbed) and for printing small detailed models. It is the only material that is well suited for printing 50-micron layers (Ultradetail resolution) and it can be also used to produce great-looking miniatures.

### Advantages

- Easy to print
- Detailed prints of small models
- Trouble-free printing of large objects
- Hard, low flexibility
- Almost odorless
- Affordable
- Wide color selection

### Disadvantages

- Brittle
- Low-temperature resistance (60 °C)
- Difficult to post-process
- Not suitable for outdoor use

### Typical use

- prototypes
- toys
- figures

- jewelry (tiny and detailed models)
- architecture models

### Tips and tricks

When post-processing PLA, use wet sanding to achieve a perfectly smooth surface. Without water, you'll quickly start heating the plastic by friction - the object can melt locally. PLA is only soluble in chemicals like chloroform or hot benzene. The preferred option for connecting multiple parts is superglue or acetone.

- **Nozzle temperature**: 215 °C
- **Bed temperature**: 50 - 60 °C
- **Heatbed**: Make sure the surface is clean, as described in 10.1 Flexible steel sheet surface preparation chapter
9.2 PET/PETG

PETG is another commonly used material for 3D printing. It is a great choice for printing **mechanical components**. Compared to PLA, it has **higher temperature resistance**, is more **ductile** and therefore **less brittle**. Due to its low thermal expansion it holds well on the heatbed and **does not warp**. Printing with it is almost as easy as with PLA. But unlike PLA, it can offer better mechanical properties. Parts for our printers are printed from PETG!

<table>
<thead>
<tr>
<th>Advantages</th>
<th>Disadvantages</th>
</tr>
</thead>
<tbody>
<tr>
<td>High-temperature resistance</td>
<td>Not suitable for printing small models</td>
</tr>
<tr>
<td>Easy to print</td>
<td>Possible stringing</td>
</tr>
<tr>
<td>Low thermal expansion</td>
<td>Bridging is problematic</td>
</tr>
<tr>
<td>Ductility and strength</td>
<td>Strong adhesion to the printbed</td>
</tr>
<tr>
<td>Easy sanding</td>
<td>Cannot be smoothed with acetone</td>
</tr>
<tr>
<td>Almost odorless</td>
<td>Supports can be difficult to remove</td>
</tr>
<tr>
<td>Glossy surface</td>
<td></td>
</tr>
<tr>
<td>Perfect layer adhesion</td>
<td></td>
</tr>
<tr>
<td>Does not absorb water</td>
<td></td>
</tr>
<tr>
<td>Recyclable</td>
<td></td>
</tr>
</tbody>
</table>

**Typical use**
- mechanical components
- holders and cases
- watertight prints (pots)

**Tips and tricks**

Unlike PLA or ABS, PETG tends to ooze a bit and may leave **strings of plastic** on your print. You can fight this by increasing retraction and playing with hotend temperature. But if you use our presets in PrusaSlicer, we already did that for you and the amount of stringing is minimal. If you witness a tiny bit of stringing anyway, you can get rid of it by quickly blasting your finished prints with a heat gun. PETG is often referred to simply as PET. Generally, they’re almost the same material (you can even get pure PET filament, but it’s difficult to print with).

- **Nozzle temperature**: 240 °C
- **Bed temperature**: 80 - 100 °C
- **Heatbed**: Make sure the surface is clean, as described in 10.1 Flexible steel sheet surface preparation chapter (**extra important with PETG**, risk of damage to the Smooth PEI sheet if treated incorrectly)
9.3 ASA/ABS

ASA and ABS are very similar materials. ASA is many ways even better than ABS and can be considered a successor to ABS. ASA is UV stable compared to ABS (less yellowing) and shrinks somewhat less when printed. The only advantage of ABS is the easier surface treatment with acetone. Going forward, we will only talk about ASA, but the same goes for ABS.

ASA is a strong and versatile material. A higher melting point than PLA gives ASA excellent heat resistance, your prints will not show signs of deformation up to around 100 °C. Unfortunately, ASA has a very high thermal expansion compared to PLA, which complicates printing, especially for larger models. Even with a heated bed set to 100 °C, the print can begin to warp and peel off the bed. The material also produces an unpleasant odor during printing.

### Advantages
- High impact and wear resistance
- Very good temperature resistance
- Suitable for outdoor use - UV stable
- Soluble in acetone - easy to glue together
- Can be smoothed with acetone vapors

### Disadvantages
- Difficult to print
- Tendency to warp
- Worse detail resolution
- Unpleasant odor during printing

### Typical use
- covers and protective cases
- prototypes
- replacement parts
- toys and figures

### Tips and tricks
Printing with ASA/ABS is much easier when done inside an enclosure. The heated bed will quickly heat the enclosure and reduce the thermal shock on the extruded filament. As a result, both warping and layer separation is decreased significantly. Acetone makes it easy to glue multiple prints together. All you have to do is rub the contact surfaces lightly with acetone and press the parts together. In addition, prints can be smoothed with acetone vapor to give a perfectly glossy finish. Be careful when handling acetone!

- **Nozzle temperature**: 245 - 265 °C
- **Bed temperature**: 90 - 100 °C. (a larger object requires a higher temperature)
- **Heatbed**: Make sure the surface is clean, as described in 10.1 Flexible steel sheet surface preparation chapter
9.4 Flex

Flex is a very strong and elastic material. In many cases, classic hard plastics may not be suitable for printing a specific model. Whether you are printing a phone cover, an action camera case, or an RC car wheel, it is better to use a flexible material.

Before you print from Flex, clean the nozzle from any previous material. Start by preheating to the highest temperature you’ve recently printed at. Then load PLA into the extruder to push out any previous material. When inserting Flex, loosen the pressure on the idler as much as possible.

**Advantages**
- Flexibility and elasticity
- Minimal warping
- Good layer adhesion
- Abrasion resistance

**Disadvantages**
- It requires extra steps to load the filament
- More demanding to prepare and print
- Must be printed slowly
- Higher price
- Must be stored in a dry environment

- **Nozzle temperature:** 230 - 260 °C
- **Bed temperature:** 45 - 65 °C (a larger object requires a higher temperature)
- **Heatbed:** Make sure the surface is clean, as described in 10.1 Flexible steel sheet surface preparation chapter. **WARNING:** Some very soft materials can adhere very strongly to the smooth PEI sheet and require the use of a glue stick as a separator to avoid damaging the PEI surface. This is not necessary with the textured powder-coated PEI.
10 Regular Maintenance

We have designed the MINI as a reliable 3D printing workhorse, however, it’s still a machine with moving parts, which means it requires maintenance from time to time. Following the instructions below will ensure that your 3D printer will remain in good shape for a long time.

10.1 Steel sheets

If the printed objects have trouble adhering to the steel sheet surface, you need to wipe the sheet clean. Choose the right cleaning product for your sheet (see below), pour a small amount of it onto an unscented paper towel and wipe the print surface. The bed should be cleaned while cold to achieve the best results - cleaning the bed when hot can lead to potential injury, either from the heatbed or from the nozzle. When cleaning at higher temperatures, the alcohol will evaporate before it can clean anything. More details can be found in the Your First Print chapter at the beginning of this handbook.

The surface does not have to be cleaned before every print! It is just important not to touch the steel sheet with your hands or dirty tools.

Recommended cleaning products for different types of sheets are listed below:

**Smooth PEI sheet**
- ✅ Isopropyl alcohol 90%+ (IPA) - best degreaser
- ✅ Windex - degreasing effect is worse compared to IPA
- ✅ Warm water + few drops with dish soap (in case IPA/Windex don’t remove sugar residue on the sheet)
- ✅ Acetone - once in a while to rejuvenate the sheet
- ![Use Windex when printing with PETG
- ![Use glue stick when printing FLEX
- ![IPA + PETG will cause the print to adhere very strongly to the sheet. Removing it could be extremely difficult

**Textured powder-coated sheet**
- ✅ Isopropyl alcohol 90%+ (IPA) - best degreaser
- ✅ Windex - degreasing effect is worse compared to IPA
- ![Never use acetone

Calibration values might be slightly different for different steel print sheets because the coating thickness varies. If you just switched the steel sheets, **visually inspect the quality of the first layer** and adjust the nozzle height accordingly with **Live Adjust Z** when switching between different types of steel sheets.
Consumable parts, such as PEI sheets (smooth, textured, etc.) are **not covered by our warranty** because the coatings are designed to diminish over time unless failure has occurred due to a defect in materials or workmanship. Cosmetic damage, including but not limited to scratches, dents, cracks or other cosmetic damage is also not covered by the warranty. Only sheets that are defective on arrival are covered by warranty.

The surface does not have to be cleaned before every print! It is just important to **not touch** the steel sheet with your hands or dirty tools.

All original print surfaces by Prusa Research are coated from both sides.

---

**Smooth PEI sheet, Textured powder-coated PEI and the effect on the first layer**

10.2 Spring steel sheet with TEXTURED double-sided PEI

- ✔ Scratch-proof surface, damage-resistant
- ✔ Transfers texture onto the first layers of your prints
- ✔ More forgiving Live Adjust Z setting
- ✔ No need for Windex with PETG
- ✔ No need for glue stick with FLEX
- ✔ Prints usually detach automatically once the sheet cools down
- ✔ The first layer can be more squished (see Live Z calibration chapter) compared to smooth sheets
  
- ❌ PLA prints with a small contact area may need a brim
- ❌ PLA prints with a huge footprint (across the entire sheet) may warp
- ❌ Need to change Live Adjust Z if you also own Smooth PEI sheet
- ❌ **Never clean with acetone**
The powder coating applied directly onto metal helped us create a build plate that is highly damage-resistant. If a heated nozzle crashes into it, metal can dissipate the heat. Powder coating also gives the surface a distinct textured look that will be visible on your prints.

The textured surface hides most of the mechanical damage caused by various tools. Only the top of the small bumps can be scratched, however, these scratches will not affect the bottom of the model.

Never clean the textured powder-coated sheet with acetone! Doing so will create microfractures in the PEI texture, which will cause the surface to deteriorate over time.

This surface is used on our print farm, look at the printed parts you have on your printer for reference.

### 10.3 Spring steel sheet with SMOOTH double-sided PEI

- Perfect for PLA
- Great adhesion with almost all materials
- Smooth bottom of prints
- Tiny details will stick great
- Rejuvenate with acetone from time to time
- Do not clean with IPA when printing with PETG. Release agent might be necessary.
- You must apply glue stick when printing Flex
- Need to change Live Adjust Z if you also own the textured PEI sheet

The industrial glue which holds the PEI sheet on the heatbed itself softens when temperatures greater than 110 °C are used. If higher temperatures are used, the glue can migrate under the PEI and create slight bumps on the surface.

If you notice small bubbles forming under the PEI sheet simply flip the sheet over and print on the other side. These bubbles do not affect the print quality and will disappear after a few days or weeks.

### 10.4 Increasing the adhesion

On some special occasions, such as printing a tall object with a very small contact area (with the print surface), you might need to increase the adhesion. Fortunately, PEI is a very chemically resistant polymer and you can temporarily apply various adhesion solutions
without damaging it. This also applies to certain materials that would not stick to PEI under normal circumstances. Check the material guide for material-specific instructions.

Before applying anything onto the bed, consider using the **Brim option** in PrusaSlicer which increases the surface area of the first layer.

**10.5 Bearings**

After a couple hundred hours of printing, the smooth rods should be cleaned with a paper towel. You can buy a lubricant (any good super lube will do) and apply it onto the smooth rods (XYZ axes). Then, move the X-axis by hand left and right a couple of times and use the printer’s menu to move the Z-axis up and down. Move the heatbed back and forth on the Y-axis. For our detailed maintenance guide, please head to [manual.prusa3d.com](http://manual.prusa3d.com)

If you feel that one of the axes doesn’t move smoothly anymore, the bearings can be taken out and greased on the inside (they need to be removed from axis because the plastic lip will prevent most of the grease from getting inside).

**10.6 Fans**

Both fans measure current RPM (Revolutions Per Minute), meaning they can detect when the fan suddenly slows down, e.g. due to a piece of filament getting stuck in it. If you get a fan error, check that it can rotate freely; clear any debris that may be stuck in it.

Both fans should be checked and cleaned every couple hundred hours. Computer cleaner spray (compressed air) will get the dust away and tweezers can be used for little plastic strands. Do not blow compressed air on running fans.

**10.7 Extruder drive gear**

The extruder gears do **not** need any lubrication. Over time, the extruder gears can suffer from a build-up of filament shavings in the grooves, which can cause under-extrusion. Use compressed air to blow the debris out or use a small brass brush to clean the grooves, a regular toothpick will do the job as well. Check and clean from the access window on the top of the extruder assembly. Clean what you can, then rotate the gear and repeat.
10.8 PTFE tubes

The MINI has **four PTFE tubes that may need a basic level of attention** from time to time. PTFE tubes are used to lead filament strand from the spool, through the extruder, all the way to the nozzle. Over time, filament debris can accumulate in the PTFE tubes over time.

Cleaning (or replacing) the PTFE tubes is very easy - use a can with compressed air to clean them.

1. The **first tube** leads to the extruder and it can be pulled out with your hand
2. The **second tube** leads from the extruder to the print head. Use a spanner (size 10) to loosen the nuts - this is also useful if the filament failed to load/unload and got stuck in this tube
3. Remove the main PTFE tube (as described in Step 2) and also undo the fittings in the extruder or in the print head to access **PTFE tube three and four**. These only need to be inspected if the filament fails to pass through them.
10.9 Filament sensor (optional accessory)

You can buy a **mechanical IR sensor as an optional accessory** for the Original Prusa MINI. The sensor is installed at the end of the PTFE tube where you insert the filament into the printer, and it needs to be connected to the mainboard. It uses a simple mechanism to detect whether a strand of filament is inserted or not. Thanks to the filament sensor, the printer can **pause the print in case you run out of filament**. Once you insert a new filament strand, you can resume the print.

**Installation instructions** can be found at [manual.prusa3d.com](http://manual.prusa3d.com).

10.9.1 Running out of filament

In case you have the **optional filament sensor**, running out of filament **will no longer cause a print failure**. If you run out of filament, the printer will automatically pause the print, unload the remaining few centimeters in the heatbreak, and move the X-carriage away from the print. You will be prompted to replace the spool and insert a new filament strand. **Use pliers to remove the filament extruded during the loading process.** Once this is done, you can continue in the current print.

If you don’t have the filament sensor, it’s possible that once you run out of filament, the **remaining strand will be left in the main PTFE tube** and the extruder won’t be able to pull it out. Use a spanner to unscrew the nut on the main PTFE tube and disconnect it from the extruder. Then, manually, pull the filament strand out - don’t forget to preheat the nozzle! See chapter PTFE tubes for exact instructions on how to remove the PTFE tube.
10.9.2 False sensor readings and debugging

You can encounter false runout readings reported by the sensor. The first step is to check if the filament sensor is working properly in **LCD menu - Support - Sensor info**. When you *insert the filament* into the extruder, the IR state should be “1”. When you *unload and remove* the filament from extruder, the state should change to “0”.

Possible causes might be:

**Wiring problem**
If the sensor is not changing state, please check whether the connectors are properly seated on both sides of the sensor cable. In case of incorrect wiring, you will see the IR sensor state as “0” (even when you insert a filament stand).

**Incorrectly seated IR sensor**
The filament sensor should be seated as depicted in the picture above. For detailed information, please see the assembly manual.

**Dust on the sensor and how to clean it**
This is not a common cause, but as a last measure, try to unscrew the filament sensor cover and take out the filament sensor carefully. Clean the sensor in the marked area as depicted in the picture below - the easiest way is to use a can of compressed air from a safe distance.

Before you mount the sensor back, connect the sensor cable (note the correct orientation) and start the printer. Open the **LCD menu - Support - Sensor info** and try to push the lever manually back and forth to simulate inserting/removing a filament. If the IR value has changed from “0” to “1”, the filament sensor is working properly, which means the problem is not in the filament sensor itself. Please refer to the assembly manual and make sure that everything is assembled properly (e.g. filament sensor level, steel ball, PTFE tubes etc.).

**IR sensor is defective**
If you have tried all mentioned solutions without success, there is a possibility that the IR sensor is defective. In this case, please contact our support.

**Clogged / jammed extruder**
Material clogged in the extruder can cause problems with the printing or with the loading of a new filament.

On the right side of the extruder (when viewed from the front), there’s a screw right under the PTFE tube. You can use this screw to increase or decrease the idler’s pressure. If you remove the screw completely, you can open the idler door on the left side of the extruder - make sure the print head is on the opposite side of the X-axis.
Opening the idler will enable you to clean both gears easily and/or remove any filament leftovers caught in the extruder gears. It's advised to inspect the gears from time to time and remove any debris caught in the extruder.

Open idler - you can see the strand of loaded filament inside
10.10 Nozzle cleaning

Do not touch the nozzle during these procedures - it’s very hot and it may cause burns!

To make cleaning easier, move the print head all the way up in the LCD menu - Settings - Move axis - Move Z axis. You will gain better access to the hotend.

The filament is being extruded just a little

If the filament doesn’t go through the extruder smoothly and only a small volume is coming out, first check if the extruder fan is working properly and that the temperature is set correctly (PLA 210 °C; ASA 260 °C, PETG 240 °C). Also, check that the filament was correctly loaded into the extruder.

If that is not the case. Follow the instructions below:

1. Heat the nozzle according to the filament you want to print from. Load the filament and push an acupuncture needle (bundled with the printer) or a thin wire (0.3-0.35 mm) into the nozzle from below - between 1 and 2 cm deep. Wearing gloves is recommended - melted material may pour out unexpectedly.
2. Choose Load filament option from the LCD menu and check if the filament is extruded properly.
3. Push the acupuncture needle or wire into the nozzle again and repeat these steps several times. When the filament is extruded properly, the nozzle is clear.

The filament is not pushed out of the nozzle

If you can see the filament passing through the main PTFE tube, yet none of the filament is going through the nozzle, then most likely, your hotend got clogged. In order to fully clear it, please follow these instructions:

1. Heat up the nozzle to 250 °C for PLA or 270°C for ABS jams.
2. Wait 3-5 minutes and then go to LCD Menu - Load filament. If you cleared the clog and the filament went through, simply lower the temperature to normal and re-do load filament again.
3. If the filament loads successfully, you can resume printing.

10.11 Replacing / changing the nozzle

If you are replacing the Olsson Ruby nozzle, please visit this website for instructions, otherwise, you might damage it! support.3dverkstan.se/article/66-the-olsson-ruby-instructions-for-use .
CAUTION: Heated parts can cause severe burns! Be extra careful around the hotend thermistor leads. You can break them off easily.

1) Preheat the nozzle to 280 °C (LCD Menu - Settings - Temperature - Nozzle).
   Heating the nozzle is essential for this process!

2) Unload any loaded filament

3) Gain better access to the nozzle by moving the extruder axis as high as possible: Go to LCD Menu - Settings - Move Axis - Move Z. Alternatively, you can just hold down the knob for a few seconds. Rotate the knob to set the height.

4) Hold the heater block with a 16mm spanner (or an adjustable wrench)

5) Use pliers, or preferably a 7mm socket to unscrew the nozzle. Be careful, the nozzle is still hot! Put the original nozzle in a safe location and carefully insert the new nozzle and tighten it gently - do not use excessive force!

6) Insert a filament and you are ready to print.

⚠️ The nozzle is hot during this whole process and can cause burns!
Be careful around the hotend thermistor leads, you can break them easily!
You can bend the heatbreak easily - do not use excessive force!

Run **First layer calibration** after changing the nozzle!

### 10.12 Flashing unsigned firmware

**We take safety very seriously.** Before every release, our firmware goes through intense testing to verify that all safety features work properly. The printer watches the RPM of both fans and the temperature of all thermistors. If any of the readings fall out of line, **heating is stopped to prevent any possible damage** to the printer and its surroundings. Furthermore, the Original Prusa MINI can connect to your home or company network and that’s another focus point of our developers regarding safety. Unfortunately, **we cannot verify that these safety features are present in the community-made firmware.**

Our goal is not to discourage our community from developing custom-made firmware - on the contrary. On the other hand, flashing firmware files from unknown sources may have serious consequences (e.g. if the heater failsafe doesn’t work correctly).

**Official firmware is always signed with a private key** and the printer verifies this signature before an update. In order to **flash your own (or a community-made) unsigned firmware,** you’ll have to **break the appendix on the Buddy board.** Breaking the appendix also enables developer tools such as ST-LINK and DFU.

⚠️ **Breaking the appendix on the Buddy board is IRREVERSIBLE and voids warranty of your printer’s electronics.** If you break the appendix, we disclaim responsibility for any possible damage done to the printer and/or its surroundings (e.g. in case of a fire).

![Unbroken (left) and broken appendix (right)](image)
11 FAQ - common issues and how to solve them

11.1 Skewed X-axis

If Mesh Bed Leveling fails, there are two possible causes of this: a faulty M.I.N.D.A. sensor, or a skewed X-axis. The latter is usually the result of incorrect assembly. Loosen the three screws that connect the whole Z-axis assembly to the Y-axis (heatbed). Re-attach the Z-axis according to the assembly manual. Pay special attention to the correct alignment of both parts.

Once the 3D printer is reassembled, move the Z-axis (nozzle) close to the steel sheet without touching it. Then, carefully move the print head from left to right by hand and observe, whether the nozzle keeps the same distance from the sheet all the time.

Slight deviations are allowed, since the printer can compensate for them thanks to the Mesh Bed Leveling routine.

A faulty M.I.N.D.A. sensor can be recognized easily. When you start Mesh Bed Leveling from the Calibration menu, you can see a red light at the top of the sensor. If it turns off during a certain movement or it doesn’t light up at all, then the sensor or its cable can be defective. Please contact our tech support.

11.2 Printer can’t read the USB drive

If the printer can’t read the USB drive, first try restarting the printer. In case you see an error say “Error mounting USB drive”, your USB drive probably has an incompatible filesystem (e.g. exFAT).

There are two common situations after inserting the USB drive:

- The Print menu is inaccessible after inserting a USB drive
  - First, restart the device
  - Use a USB drive with a single partition and FAT32 file system
  - Try a different USB drive
  - If you tried various USB drives and they are still not recognized, your motherboard might be defective. Contact our tech support.

- The USB drive is recognized, but no files are shown in the list
  - Make sure you’re using a compatible G-code
  - Make sure the file was written onto the drive fully and correctly (use the safe remove function in Windows before you remove the USB drive)
  - Try a different flash drive and G-code file
  - Try renaming the file to something simple, e.g. print.gcode
11.3 Loose X- and/or Y-axis belts

Check if both belts are properly tightened, loose belts would cause a printer malfunction and prevent proper printing. The easiest way to check is printing a round object - if any of the belts are not tightened properly the result is an irregular shape instead of a perfect circle. Y-axis belt is located under the heatbed, X-axis belt moves the print head.

11.4 Filament stuck in the PTFE tube

If a filament strand gets stuck anywhere in the printer, it’s generally very easy to remove it. Follow the steps described in the Clogged / jammed extruder chapter - disconnect the main PTFE tube and inspect where the filament got stuck and proceed accordingly.

11.5 X-Axis homing fails

This is most likely caused by the cables leading from the print head to the extruder getting in the way. When the print head is about to hit the right side of the printer (the Z-axis tower), pay close attention whether there are no cables blocking the way.

11.6 Heating errors

If your printer stops with a red screen and error related to heating, please check the wiring of the heaters and thermistors. See manual.prusa3d.com for more details.

11.7 Fan errors

If your printer stops operating and displays an error related to fans, please check the wiring of both print head fans (hotend fan and print fan). See manual.prusa3d.com for more details.

12 Advanced hardware troubleshooting

Advanced hardware troubleshooting and part replacement guides are available online through manual.prusa3d.com.

13 Print quality troubleshooting

If you feel your prints are not fully up-to-specs, or they display some serious quality issues (shifted layers, ghosting, under-extruded parts), you may need to run some troubleshooting. Head over to our website www.prusa3d.com and visit the Support section, where you can find detailed troubleshooting guides along with illustrations and printer-specific instructions.
Print and share!

Do not forget to tag your prints with #prusamini while sharing so we can find, pin and showcase them with our

http://www.prusa3d.com/original-prusa-i3-prints/

Happy Printing :)