

3D PRINTING HANDBOOK

USER MANUAL FOR 3D PRINTERS:

- ORIGINAL PRUSA i3 MK2 KIT 1.75MM
- ORIGINAL PRUSA i3 MK2 1.75MM





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

QUICK GUIDE TO THE FIRST PRINT

1. Place the printer on a flat and stable surface (page 11)
2. Read the safety instructions carefully (page 7)
3. Download and install the drivers (page 27)
4. Prepare the print surface (page 11), run selftest (page 13) and calibrate the printer (page 14)
5. Insert SD into the printer and print your first model (page 20, page 29)

About the author

Josef Prusa (born Feb 23rd, 1990) became interested in the 3D printing phenomenon before joining the 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 public.

Jo's work on self-replicating printers (you can print the other printer parts with your printer) are still ongoing and currently there is Prusa i3 - the third iteration of the original 3D printer. It is constantly updated with the latest innovations and you've just purchased its latest version. In addition to printer hardware upgrades, the main goal is to make the technology more accessible and understandable for 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 a 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
2 Product details	6
3 Introduction	6
3.1 Glossary	6
3.2 Disclaimer	7
3.3 Safety instructions	7
3.4 Licenses	7
4 Original Prusa i3 MK2 printer	8
5 Original Prusa i3 MK2 printer kit	9
6 First steps	10
6.1 Printer unpacking and proper handling	10
6.2 Printer assembly	11
6.3 Setup before printing	11
6.3.1 PEI print surface preparation	11
6.3.2 Increasing the adhesion	12
6.3.3 Plugging the AC power	13
6.3.4 Selftest	13
6.3.5 Preparing the Z Calibration	14
6.3.5.1 Checking the Y-Axis (kit only)	14
6.3.5.2 Probe alignment (kit only)	14
6.3.5.3 Checking the Z-axis	16
6.3.6 Loading the filament into the extruder	17
6.3.7 Z Calibration	18
6.3.7.1 Finetuning the Z adjustment	19
6.3.7.2 Securing of the probe (kit only)	19
7 Printing	20
7.1 Printer Control	21
7.1.1 LCD screen	21
7.1.2 Print statistics	21
7.1.2 Silent vs. Hi-power mode	22
7.1.3 Factory reset	22
7.1.4 LCD layout	22
7.1.5 Print speed versus print quality	24
7.1.6 USB cable and Pronterface	24
7.2 Printer addons	26
7.2.1 Different nozzles	26
7.2.1.1 Hardened steel nozzle	27
7.2.1.2 0.25mm nozzle	27
8 Printer drivers	27
9 Printing your own models	27
9.1 Where you can get the 3D models?	27
9.2 In what program you can create your own models?	28

9.3 Slic3r	28
9.4 Bundled 3D models	29
9.5 Print in color with ColorPrint	30
9.6 Printing of non-standard models	32
9.6.1 Printing with support material	32
9.6.2 Large object printing	33
10 Materials	34
10.1 ABS	34
10.2 PLA	34
10.3 PET	34
10.4 HIPS	34
10.5 PP	34
10.6 Nylon (Taulman Bridge)	34
10.7 Flex	35
11 FAQ - Printer maintenance and print issues	36
11.1 Print surface preparation	36
11.2 Clogged / jammed extruder	36
11.3 Nozzle cleaning	37
11.4 Replacing / changing the nozzle	37
11.5 Axis fluency	38
11.5 Printing problems	39
11.5.1 Layers break and split when printing from ABS material	39
11.5.2 Models contain either too much or not enough of the filament	39
11.6 Problems with finished models	39
11.6.1 Model breaks and/or is easily damaged	39
12 FAQ - common issues when assembling the printer kit	40
12.1 Nozzle/print surface gap is greater in the middle than at the corners	40
12.2 Printer stops printing soon after start	41
12.3 Printer can't read SD card	41
12.4 Loose X- and/or Y-axis belts	42
12.5 Detached cables to the heatbed	43
12.6 Loose heatbed zipties	43



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



Read carefully! This part of text has uppermost importance - either for user safety of for a proper printer service.



This symbol indicates text related to printer kit only.

2 Product details

Title: Original Prusa i3 MK2 / Original Prusa i3 MK2 (kit), Filament: 1.75 mm

Manufacturer: Prusa Research s.r.o., 621/8a Prvniho pluku, Prague, 186 00, Czech Republic

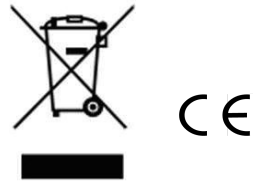
Contacts: phone +420 222 263 718, e-mail: info@prusa3d.com

EEE group: 3 (IT and/or telecommunication equipment), Device use: indoor only

Power supply: 90-135 VAC, 2 A / 180-264 VAC, 1 A (50-60 Hz)

Working temperature range: 18 °C (PLA)-38 °C, indoor use only

Working humidity: 85 % or less



Kit weight (brutto / netto): 9.8 kg / 6.3 kg, assembled printer weight (brutto / netto): 12 kg / 6.3 kg. Serial number is located on the printer control unit and also on the packaging.

3 Introduction

Thank you for purchasing our original 3D printer **Original Prusa i3 MK2** from Josef Prusa either as an assembled printer or a printer kit - as your purchase supports us with its further development. Read the handbook carefully, please, all chapters contain valuable info for the correct service of the printer.

Please check the <http://prusa3d.com/drivers> page for 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. We are glad to receive all your valuable comments and tips. We strongly suggest you to visit our official forum at forum.prusa3d.com, where you can find solutions to common issues, tips, advice and hints in addition to actual information about the Original Prusa i3 printer's development.

3.1 Glossary

Bed, Heatbed, Printbed - Commonly used term for printing pad - a heated area of 3D printer where 3D objects are printed.

Extruder - Printing head or extruder is a part of a printer consisting of a nozzle, hobbed pulley, idler and a nozzle fan.

Filament - Term for plastic provided on spool is called "filament", it's used throughout this handbook as well as in the LCD menu on the printer.

Heater, Hotend - other name for a printing nozzle.

1.75 - 3D printers use two different diameters (thickness) of a filament (thickness): 2.85 mm (commonly called as 3 mm) and 1.75 mm. 1.75mm version is more used worldwide though there is no difference in printing quality.

3.2 Disclaimer

Failure to read the Manual 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. We can not control the conditions in which you assemble the Original Prusa i3. 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 Manual is provided without any warranty, express or implied, regarding its correctness.

3.3 Safety instructions



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


1. 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.
2. Always place the printer on a stable place, where it can not fall or tip over.
3. The printer supply is household power outlet 230 VAC, 50 Hz or 110 VAC / 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 can't stumble on it, or step on it or otherwise expose to any damage. Make sure that the power cord is not mechanically or otherwise damaged. Stop using damaged cable 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 plug or AC outlet.
6. Never disassemble the printer power supply, it does not contain any parts that could be repaired by an unskilled worker. All repairs must be provided by a qualified technician.
7. Do not touch the nozzle or heat bed when the printer is printing or is warming up. Note that the temperature of the nozzle is 210-300 °C (410-572 °F); heatbed temperature can reach over 100 °C (212 °F). Temperatures above 40 °C (104 °F) can cause harm to human body.
8. Do not reach inside the printer while it is still in operation. An injury may be caused by its moving parts.
9. Prevent children from unsupervised access to the printer even when the printer is not printing.
10. Do not leave the printer unattended while it's still on!
11. Plastic is being melted during printing which produces odors. Set up the printer some place well ventilated.

3.4 Licenses

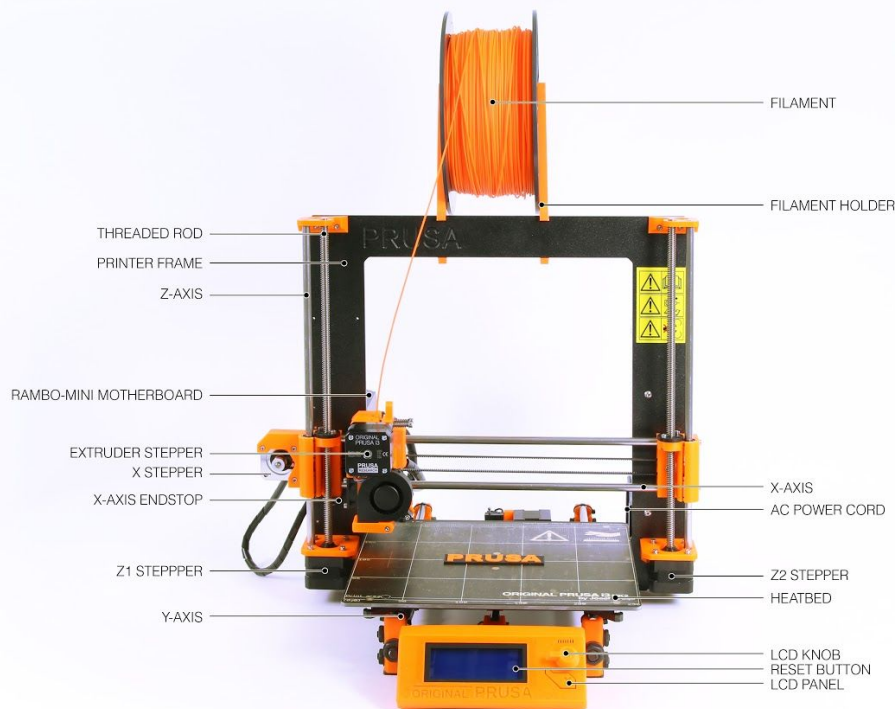
Original Prusa i3 MK2 printer is a part of the RepRap project, the first open source 3D printer project free to use under a GNU GPL v3 license (www.gnu.org/licenses/gpl-3.0.en.html). If you improve or alter any part of a printer and you are willing to sell, then you have to publish the source codes under the same license. All 3D-printed elements of the printer that can be improved upon can be found at <http://www.prusa3d.com/prusa-i3-printable-parts/>.

4 Original Prusa i3 MK2 printer

Unlike the printer kit, it's completely assembled and almost ready to print. After plugging in and running the necessary calibration you, can print a 3D object in the matter of minutes after unpacking the printer. Keep in mind you can use our support email when you purchased the assembled printer. Do not hesitate to write us if you need any advice or help. We will gladly help with any specific prints.

 3D printers use two different diameters of a filament (you can find more in chapter *Materials*): 2.85 mm and 1.75 mm. 1.75mm version is used more worldwide, though there is no difference in printing quality. Filament is provided on a spool where you can find the basic information - filament maker, material (ABS, PLA, etc.) and filament diameter. 2.85 mm filament is commonly called as 3 mm.

This printer supports only a 1.75 mm filament. Please check the filament diameter to be 1.75mm before inserting into the extruder. Do not try to insert wider filament it could damage the extruder.



Pict. 1 - Original Prusa i3 MK2 printer description

5 Original Prusa i3 MK2 printer kit



Original Prusa i3 MK2 kit is pictured in pict. 2. Detailed information and assembly description can be found in chapter [6.2 Printer assembly](#). We offer the support for users who purchased the printer kit through our official forum. If you need help do not hesitate to visit our forum at forum.prusa3d.com. You can find the answers for your problem there. If not, please just post your question directly there.

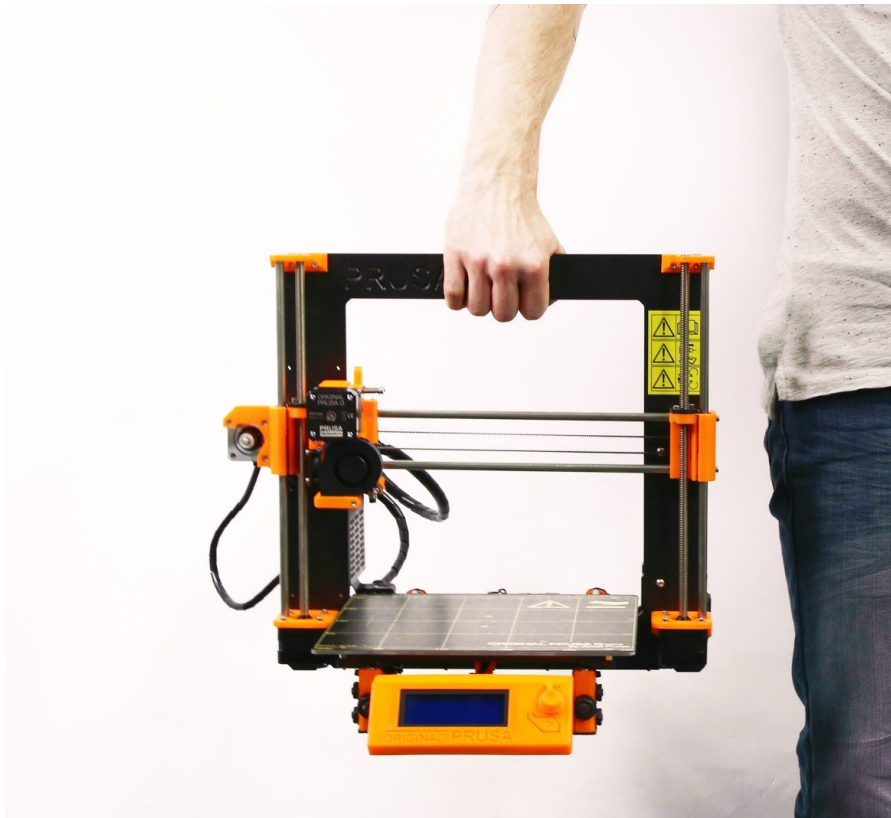


Pict. 2 - Original Prusa i3 MK2 printer kit unboxed

6 First steps

6.1 Printer unpacking and proper handling

Holding the upper frame, take the printer and pull it out from the box. Be careful when handling the printer not to damage the electronics and thus the proper printer functionality. Anytime you move the printer, always hold the upper frame with hotbed upright pointing away from you as pictured in pict. 3.



Pict. 3 - Proper handling of a printer

6.2 Printer assembly



With Original Prusa i3 MK2 printer kit we suggest to follow the guidelines and assemble the kit according to the the online manual at manual.prusa3d.com. (Online manual is available in several languages on the website). The construction of the printer should not take more than one working day. After a successful completion continue to the chapter [6.3 Setup before printing](#).

6.3 Setup before printing

- Place the printer to a horizontally stable position, best place is a workbench where there is no risk of draft.
- Attach the filament holders to the upper frame.
- Attach **Filament** to the holders. Make sure the filament spool doesn't jam and can move freely.



Filament is a common term for the **plastic rod** - material provided on a spool from which 3D objects are printed.

6.3.1 PEI print surface preparation

To achieve the best adhesion on the new surface, it is important to keep the surface clean. Cleaning of the surface is very easy. The best option is **Isopropyl alcohol** available in drugstores which is the best for ABS, PLA and others, expect for PETG where the adhesion may be too strong. Second option is to use a gentle spray of a **window cleaner** (Windex or similar) on a paper towel and wipe the surface. The bed can be cleaned even while hot, just be careful not to touch the surface or the nozzle. Alternatively, you can clean the bed with **warm water and a few drops of a dish soap** on a paper towel. **Denatured alcohol** is yet another option.



The surface does not have to be cleaned before every print! It is just important to **not touch** the print bed with **your hands** or **dirty tools**. **Clean your tools with the same solution** as you would the bed and you will be able to start your next print right away.

You can leave small marks on the print surface with your nozzle or tools, they will typically be shinier than the rest. It does not affect the functionality or adhesion. However, if you want to have same surface look on the whole printbed you can resurface it. The easiest way is to take a hard side of dry kitchen sponge and wipe the affected area with circular motion gently few times.

6.3.2 Increasing the adhesion

In some special occasions, like 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 temporary apply other adhesion solutions without damaging it.

This also applies to materials which would not stick to PEI otherwise, like Nylon etc.

Before applying anything to the bed, consider using **Brim option** in Slic3r which increases the surface area of the first layer.

For PLA and Nylon blends a simple glue stick does the trick. Glue can be later easily removed by window cleaner or dish soap water.

For ABS prints, ABS juice can be used and later cleaned with pure acetone. Be very gentle when applying the juice and do so while the bed is cold. Prints will attach very strongly.



Prepared juice can be also purchased in our e-shop. Unfortunately, UPS service does not allow to deliver any acetone-based products due to shipping constraints. In that case you get only the bottle and ABS from our e-shop and you have to source the acetone locally.



Pic. 4 - ABS Juice can be used to increase the adhesion

6.3.3 Plugging the AC power

- Plug in the AC power cord, check to make sure the proper setting for AC voltage is selected (110V/220V) and turn on the switch.

6.3.4 Selftest

The purpose of the selftest routine is to check most common errors when assembling and connecting electronics and to help indicate any possible errors after assembly. You can run the **Selftest** from **Settings** menu on LCD panel. This should not be necessary on the assembled printers as those are pretested.

Initiating this routine performs a series of tests. The progress and results of each step are displayed on the LCD. In case of errors found, the selftest is interrupted and the reason for error is shown to guide users in troubleshooting.

Test consists of

- Heatbed** and **hotend** proper wiring
- XYZ motors** proper wiring and functionality
- XYZ endstop** mechanical settings and proper wiring.

Error messages and resolution :

Heater/Thermistor - Not connected :

Check proper wiring of hot end power cables and thermistor cables. Ensure that both are properly connected to the Rambo electronics, and that they are not swapped.

Bed/Heater - Wiring error :

Check that heatbed and hotend power cables are not swapped or thermistor cables from both hotend and heatbed are not swapped in the Rambo electronics.

Endstops - Wiring error - {XYZ} :

Check the proper cabling of endstops. Routine indicates axis on which endstop reported malfunction or is not properly responding. Check the proper connection in the Rambo electronics.

Motor - {XYZ} - Endstop {XYZ} :

Check that motor and endstop on indicated axis are properly connected to the Rambo electronics and not swapped with motor or endstop of different axis. Axis causing the problems is indicated on the LCD panel.

Endstop not hit - Motor {XZY} :


Check mechanical settings that endstop can be reached when axis is in maximal position.

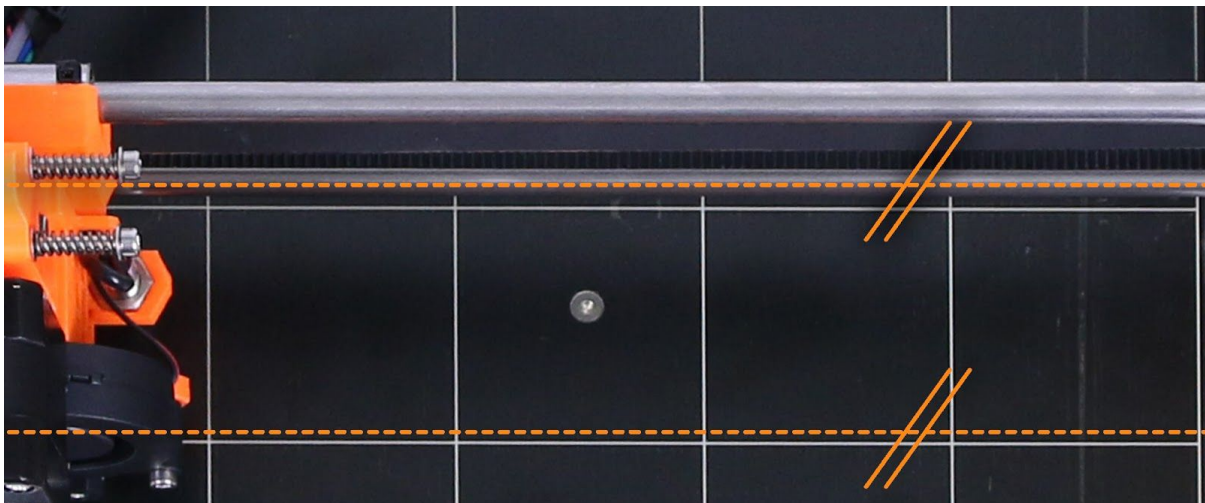
6.3.5 Preparing the Z Calibration

The Original Prusa i3 MK2 comes with a full mesh bed leveling feature, however for this to work we need to first calibrate the distance between tip of the nozzle and P.I.N.D.A (Prusa INDuction Autoleveling) probe.

The process is fairly straightforward, so let's get to it.


6.3.5.1 Checking the Y-Axis (kit only)

 For autocalibration to work properly, it is extremely important for the Y axis to be perpendicular to the X axis. This can be easily checked by looking at the printer from the top and visually aligning the X-axis rods with lines on the heatbed. You can see it on the next picture. If the Y axis is misaligned, you can easily adjust the position of the Y axis inside the frame by loosening the **M10 nuts** on the Y axis and securing them at the newly adjusted position.



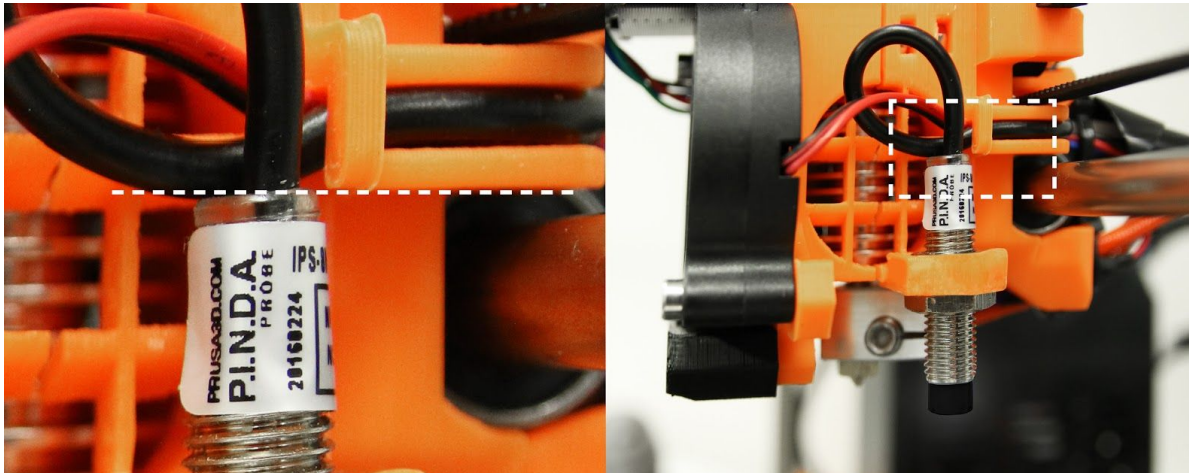
Pic. 5 - X-axis rods must be visually aligned with lines on the heatbed.

6.3.5.2 Probe alignment (kit only)

 The first step is to mechanically set the position of the probe. The top of the probe should be aligned with the bottom of a printed part as you can see in the picture number 6. The position can be easily changed even on an assembled printer by loosening the bottom nut few turns, moving the whole probe and adjusting the top nut accordingly.



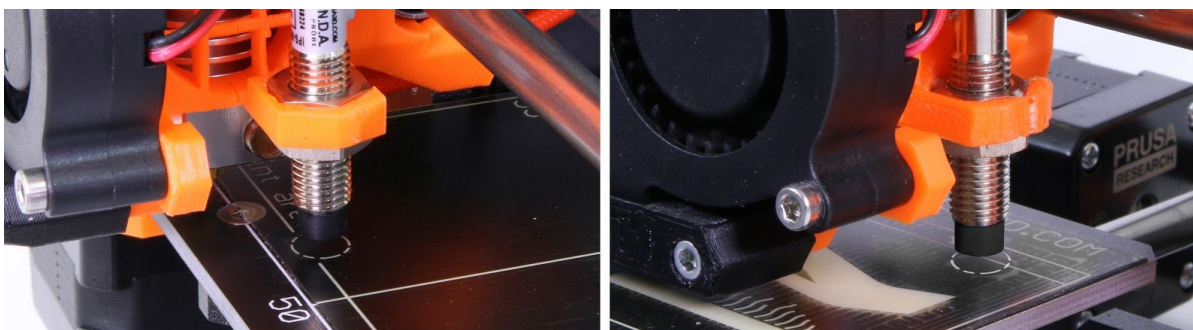
**Secure the bottom nut afterwards or it can get loose during printing later!
Also the probe should be perpendicular to the print bed.**



Pic. 6 - The top of the probe should be aligned with the bottom of a printed part

When you run the **Calibrate Z** feature from the **Settings** menu on the LCD panel, the printer will run the calibration routine. Closely watch how the probe approaches the 9 calibration points. 1st and the last points are marked with white dashed circles for easy recognition. The probe must be inside those circles to successfully calibrate.

- ⚠ The nozzle must not touch the print surface during this cold calibration. **If it bends the print surface OR the probe is not inside the circles, it is wrong and you have to readjust the P.I.N.D.A probe!** Only then you should continue with the calibration process. If you're having trouble, feel free to ask us for help. If the problem is not taken care of during the cold calibration, the nozzle will **permanently damage** the print surface when hot.

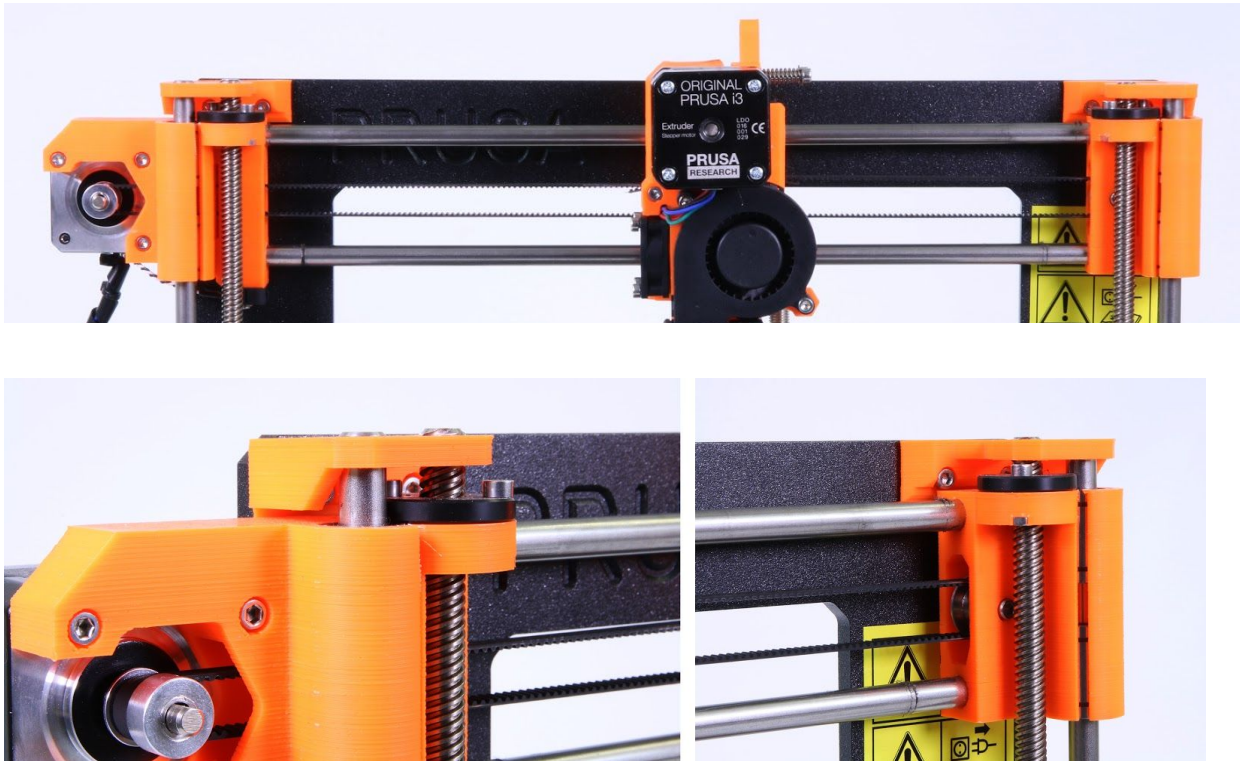


Pic. 7 - The probe must be inside the circles to successfully calibrate.

If the probe is misaligned, you can easily adjust the position of the Y axis inside the frame by loosening the **M10 nuts** on the Y axis and securing them at the newly adjusted position.

6.3.5.3 Checking the Z-axis

It is extremely important for both X-END parts to be at the same height. This can be achieved by raising the Z axis until it hits the top from the LCD menu. Go to **Settings -> Move axis -> Move Z** and rotate the knob until the axis reaches the top. Don't worry, the motors do not have enough power to damage anything, but it will make the Z axis nice and straight.

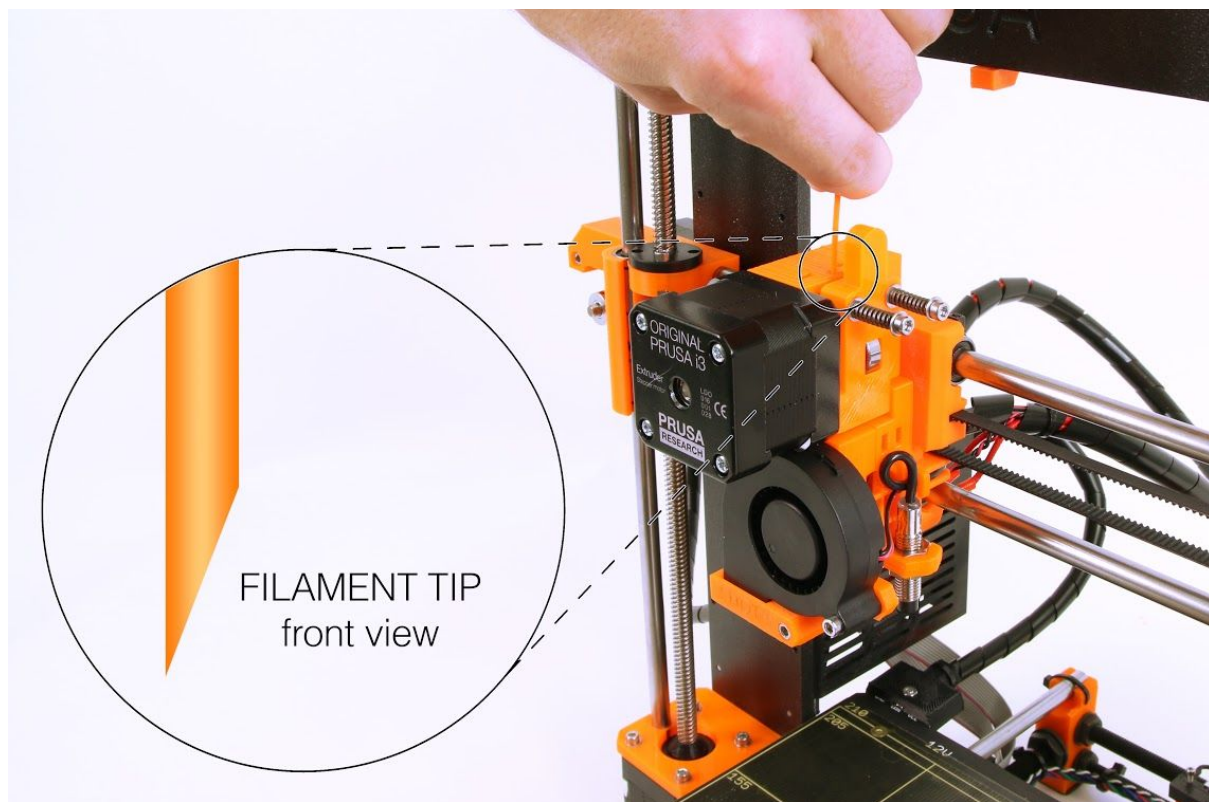


Pic. 8 - It is extremely important for both X-END parts to be at the same height

If there is any misalignment, it will cause the printer to touch the print surface with the nozzle typically on the 3 calibration points on the right side.

6.3.6 Loading the filament into the extruder

- Press the LCD-knob to enter the main menu on the LCD. Rotate the button to choose **Preheat** option and confirm by pressing the LCD-knob. Next you choose the material you will print from. Choose a material then confirm with LCD-knob. The nozzle and heatbed will heat to the requested temperature.
- You need to preheat the nozzle before inserting the filament (and the bed too if you like to print right away). The temperature depends on the material used. Detailed information about nozzle and bed temperatures are described at chapter [10 Materials](#).
- Press the LCD-knob on the LCD panel to enter the main menu. Insert the filament to the extruder, choose the **Load filament** option in the menu and press the button to confirm. Filament is then loaded to the extruder by the extruder stepper automatically. You should cut the top of the filament as shown in the picture below (pict.9 - detail).



Pict. 9 - Loading the filament to the extruder

- Check if the filament is flowing from the nozzle.
- If you change the filament for a new one do not forget to completely remove the old filament before the printing.

6.3.7 Z Calibration

*NOTE: The calibration process has been made easier and is now called V2 Calibration. If the **V2Calibration.gcode** is not present on your SD card you can easily obtain it from our support or on our <http://www.prusa3d.com/drivers/> page.*

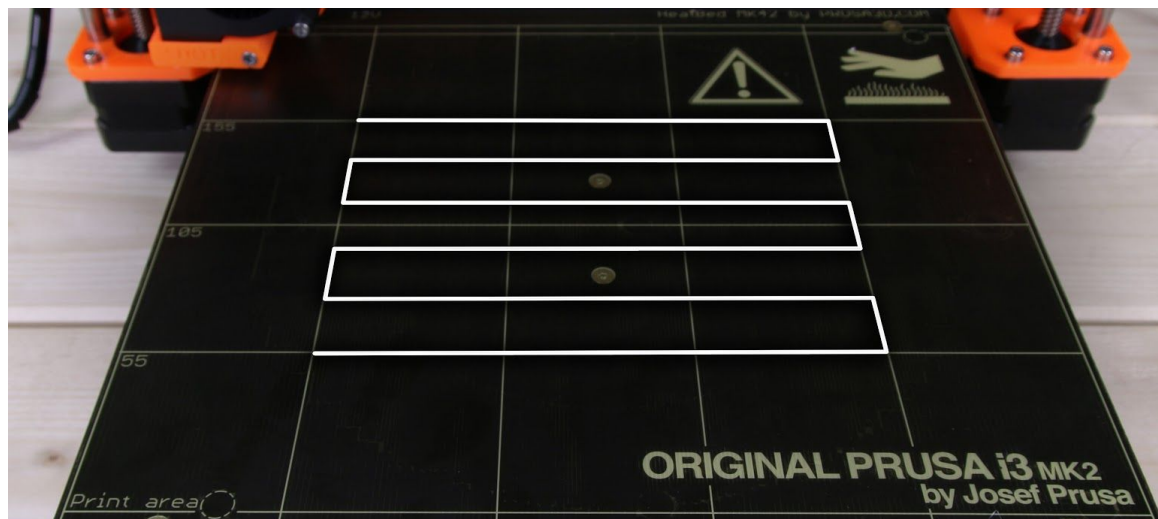


Check if your print surface is clean! You can find instructions how to clean it in the chapter [6.3.1 PEI print surface preparation](#). Don't forget to complete [6.3.5 Preparing the Z Calibration](#) chapter or **you can permanently damage the print surface!**

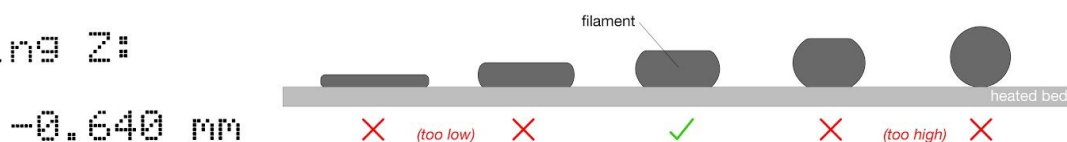
Now we will finally calibrate the distance between tip of the nozzle and the probe. Preheat the nozzle for PLA. On the LCD menu, go to **Print from SD** and run **V2calibration.gcode** file from the **bundled SD card**.

```
*Main          †
  V2Calibration.gcode
```

The printer will probe the bed and start printing zig zag pattern on the print surface. The nozzle will be at the height based on the P.I.N.D.A probe setting, it must not by any means touch the print surface.



Adjusting Z:

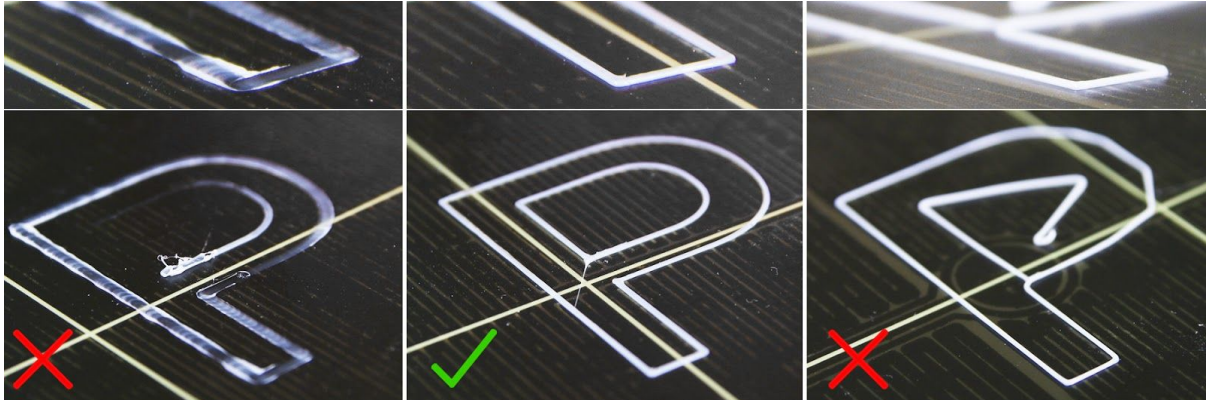


Pict. 10 - How to tune the nozzle height live during the test print

Observe the line which is being extruded on the print surface. Go to the LCD menu and choose the **Live adjust Z** option. A new menu will show up where you can tune the nozzle height live during the test print. The point is to lower the nozzle until the extruded plastic sticks nicely to the bed and you can see it is being slightly squished.

6.3.7.1 Finetuning the Z adjustment

After finishing the calibration gcode, it is a good idea to print a simple object. The Prusa gcode from the supplied SD card is a great example. The **Live adjust Z** function works during every print, so you can finetune at any point. You can see the properly tuned first layer on the images below.



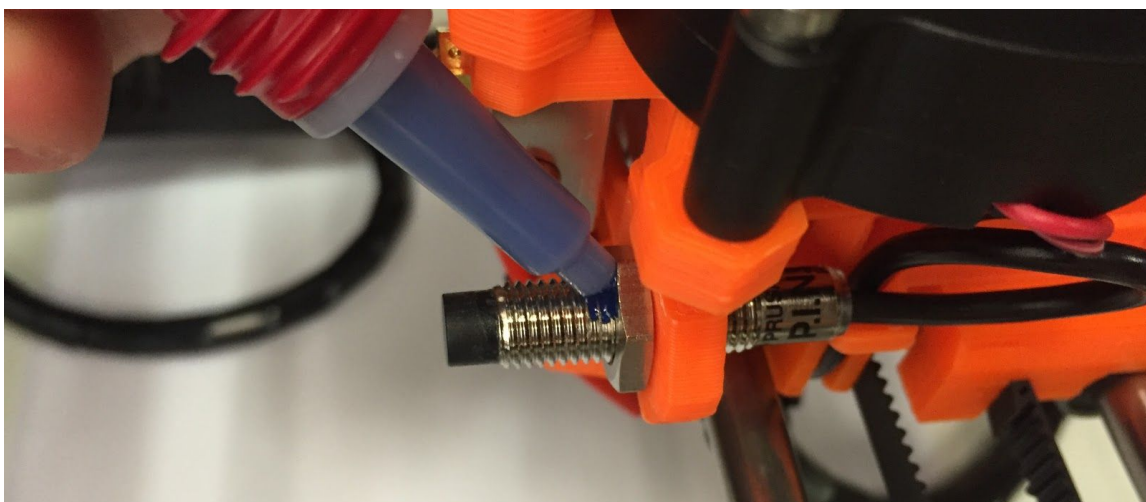
Pict. 11 - The properly tuned first layer

Now you are done!

6.3.7.2 Securing of the probe (kit only)



After couple of prints ideally secure the bottom nut on the probe with a drop of **Loctite 243** (Blue Threadlocker) to prevent loosening over time.



Pict. 12 - secure the bottom nut on the probe with a drop of Loctite 243

7 Printing

- Make sure that the nozzle and the bed are heated to the desired temperature. If you forget to preheat the printing nozzle and the bed before printing, the printer will automatically check the temperatures of the nozzle and the bed; printing will start when desired temperature is reached - it can take several minutes. However, we recommend preheating the printer beforehand as described in the chapter [6.3.6 Loading the filament into the extruder](#).



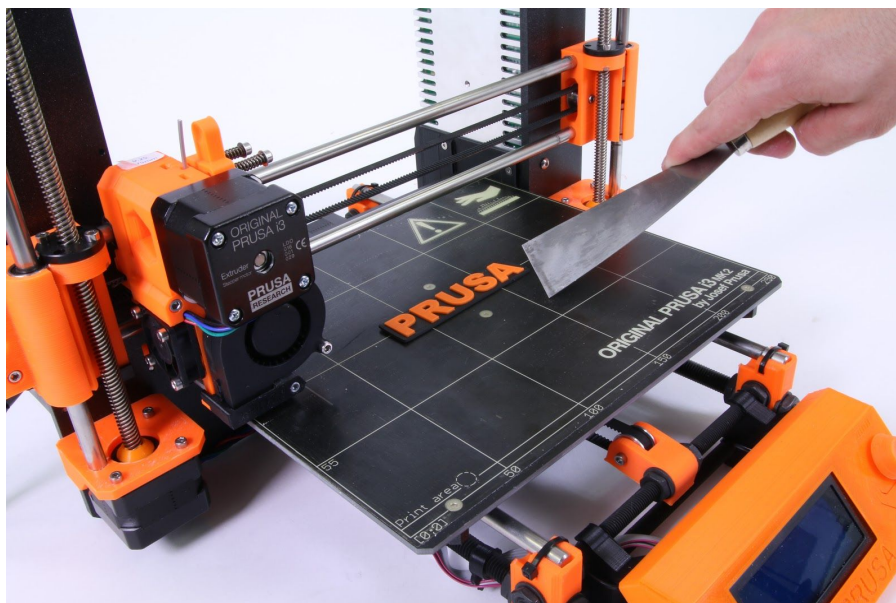
Do not let the preheated printer idle. When a printer is preheated and non-printing material in an extruder degrades over time - it may cause the nozzle to jam up.

- **Watch the first few printed layers to be sure filament has attached to the bed properly (5 to 10 minutes).**
- Press the LCD-knob and choose the **Print from SD** option from menu, press to confirm and pick the desired model **model_name.gcode**. Printer will start printing the object.



The **filename (.gcode) must not contain any special characters** otherwise the printer is not able to display the file on the LCD. If you remove the SD while printing, printer will pause automatically. When you reinsert the SD, press the LCD-knob and choose “Continue” option. Confirm and printing will resume.

- When printing is finished let the nozzle and heatbed cool down before removing the printed object. Always handle the printed objects when temperature of the bed and nozzle drop to the room temperature, when the bed is hot objects are very hard to remove. Pull the bed towards you and remove the object gently.
- If you experience any troubles removing the object (especially the small ones) you can use a flat tool like a spatula or Stanley blade.



Pict. 13 - Removing the model from PEI print surface with spatula

7.1 Printer Control

There are two ways controlling the printer. You can use the LCD panel integrated with the printer or you can connect your computer with USB cable. We suggest the **LCD panel** because of its speed and reliability, and moreover you do not rely on a computer.

7.1.1 LCD screen

- Main screen is an **information screen** displaying the most important details. These are the temperature of the nozzle and the heatbed (1, 2), printing time (3) and the actual Z-axis position (5).



The image shows a monochrome LCD screen with the following text and callouts:

- ① → 182/0° (Nozzle temperature)
- ② → 49/0° (Heatbed temperature)
- ③ → 50---% (Progress of printing)
- ④ → Prusa i3 MK2 ready. (Status bar)
- Z 0 (Z-axis position)
- ⑤ → *100% (Printing speed)
- ⑥ → 0--:-- (Elapsed printing time)
- ⑦ → (Unused callout)

Pict. 14 - LCD layout

1. Nozzle temperature (actual / desired temperature)
2. Heatbed temperature (actual / desired temperature)
3. Progress of printing in % - shown only during the printing
4. Status bar (Prusa i3 MK ready / Heating / model_name.gcode, etc.)
5. Z-axis position
6. Printing speed
7. Elapsed printing time - shown only when printing

7.1.2 Print statistics

The printer tracks printing statistics. When you access this option during a print, you will see statistics for the running print. If you do so while the printer is idle, you will see the lifetime statistics. Both filament usage and print time is being tracked.

```
Total filament :  
                    5.94 m  
Total print time :  
    0d : 1h :58 m
```

7.1.2 Silent vs. Hi-power mode

The printer offers two settings for motor power consumption. Silent uses less current and makes the printer quieter, but less powerful. Hi-power is great for very large (over 200 gram) prints and for freshly assembled kits before you fine tune everything. If you experience lost steps (shifted layers), use Hi-power mode.

7.1.3 Factory reset

Printer can be reset to factory settings, for example if you select wrong language and can't locate the language settings menu.

1. Printer must be powered on.
2. Press and hold the rotary selector
3. Press and release the reset switch marked with X directly below the rotary selector
4. The printer will indicate a successful reset by beeping. You can then release the rotary selector and wait for the printer to boot up.

7.1.4 LCD layout



Items not mentioned below are not used for the common print setup - you should not change any of the unmentioned items unless you are absolutely sure what you are doing.

- Info screen
- Live adjust Z (during the printing process only)
- Tune (during the printing process only)
 - Speed
 - Nozzle
 - Bed
 - Fan speed
 - Flow
 - Change filament
 - Mode
- Pause print (during the printing process only)
- Stop print (during the printing process only)
- Preheat
 - ABS - 255/100
 - PLA - 210/50
 - PET - 240/90
 - HIPS - 220/100
 - PP - 254/100

- FLEX - 230/50
 - Cooldown
- Print from SD
- Load filament
- Unload filament
- Settings
 - Temperature
 - Nozzle
 - Bed
 - Fan speed
 - Move axis
 - Move X
 - Move Y
 - Move Z
 - Extruder
 - Calibrate Z
 - Disable steppers
 - Auto home
 - Mode
 - Live adjust Z
 - Select language
 - English
 - Cestina
 - Italiano
 - Espanol
 - Polski
 - Selftest
- Statistics
- Support
 - prusa3d.com
 - forum.prusa3d.com
 - howto.prusa3d.com

7.1.5 Print speed versus print quality

Printing a small object takes a few minutes, but printing larger models is time consuming - there are prints taking tens of hours. The overall printing time can be changed by different ways. First way to alter the printing speed is changing layer height in Slic3r - upper right windows shows Print settings option. Default setting is 0.20 mm (NORMAL), you can speed up the printer by choosing the 0.35 mm (FAST) option. Raising speed will result the model to be less detailed with visible layer borders. If you prefer quality over speed, choose 0.10 mm (DETAIL) option. Printing time will double but the model gets the extra detail. Again, higher printing speed results in less detailed model.

Speed can be changed also while printing. LCD shows the FR 100 % item - it's actual print speed (feed rate). By turning the LCD-knob clockwise you can increase the print speed up to 999 %. However, we do not advise to increase the speed over 200 %. Watch the results of increased speed on the printed model and adjust the speed eventually.



When increasing the speed always check the model is cooled properly - especially when printing small object from ABS increased speed causes the distortion (sometimes called "warping") of the model. You can prevent this issue by printing more similar objects together - layer printing interval is long enough to prevent this issue.

If the model shows lower quality than desired you can decrease the printing speed - turn the LCD-knob counterclockwise. Minimum usable printing speed is around 20 % of nominal speed.

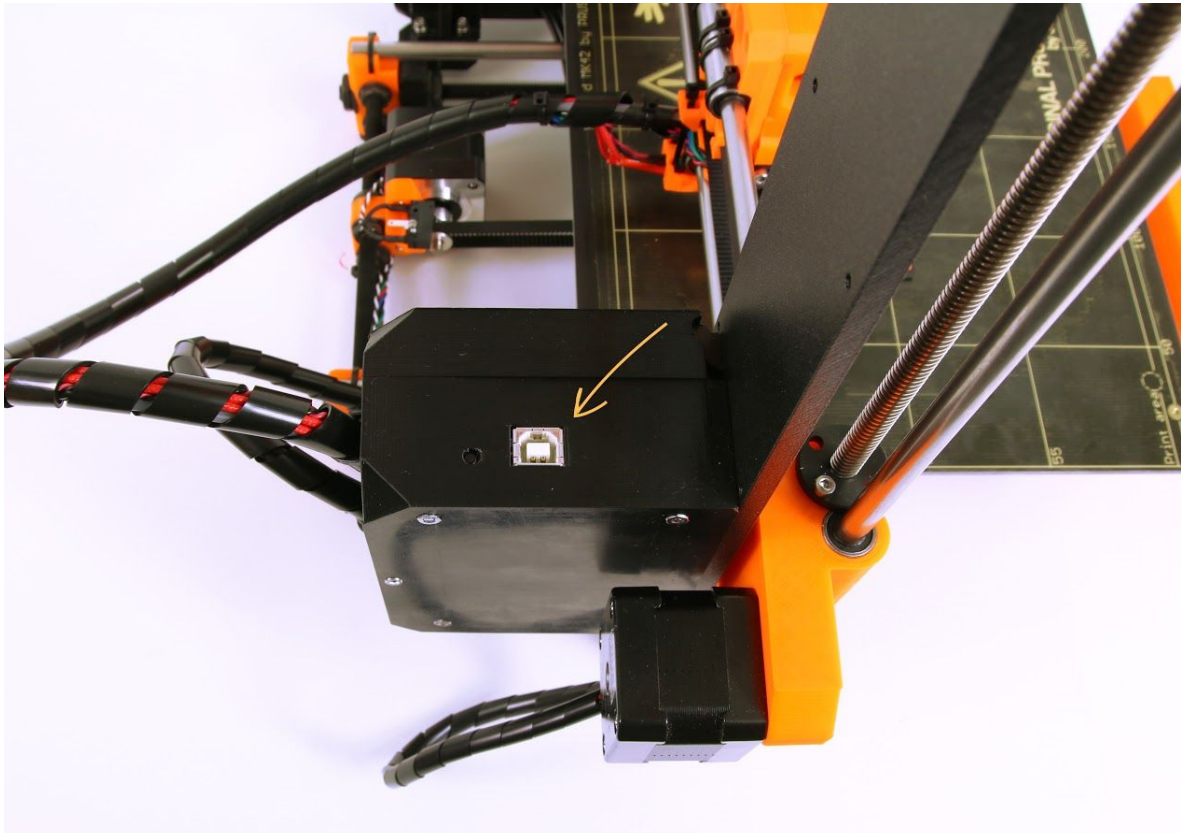
7.1.6 USB cable and Pronterface



We strongly recommend to use LCD panel while printing on Prusa i3 MK2 - Pronterface doesn't support all functions of a new firmware (e.g. filament change while printing).

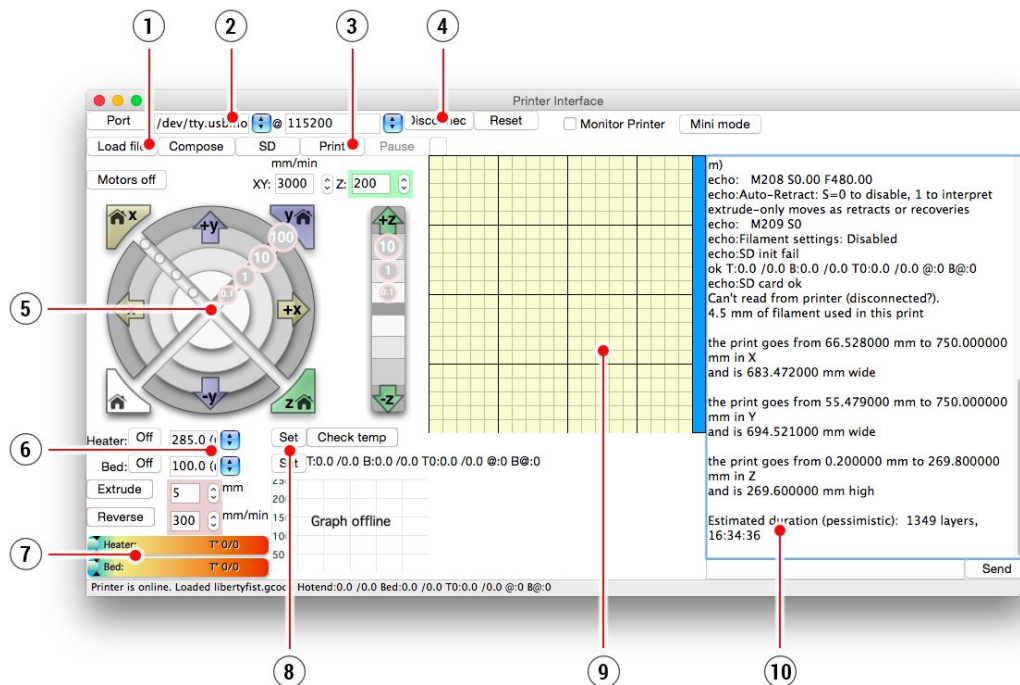
Keep in mind that when printing from the Pronterface the **computer must be connected to the printer during the whole printing process** - computer must be prevented from sleep, hibernation or shutting down. Disconnecting the computer during the print ends the printing without the option to finish the object.

- Connect the printer to the computer with the USB cable.



Pict. 16 - You can find USB port here

- Choose connection port in Pronterface (download available with the printer drivers, see the chapter [8 Printer drivers](#)): Mac users use `/usbmodem` port, PC Windows ports are COM1, COM2, etc.; the correct port is displayed in device manager, Linux users connect the printer using the virtual serial port. When the printer is connected click the **Connect** button. Right column shows the connection information.
- Next step is loading the model with **Load model** button and choosing the **model_name.gcode** (no special symbols in file name).
- You can control the movement of all printer axes at the control area.
- Next you can preheat the printer and prepare it for the printing. Set the temperatures for the nozzle (heater) and heatbed (bed) and click **Set** button. Printer starts heating immediately. **Always check that the temperatures set in Pronterface are correct according to our material guide!**
- You can check the actual temperatures of nozzle and bed in Pronterface.
- When model is loaded right column shows the estimated print duration: **Estimated duration (pessimistic)**



Pict. 17 - Pronterface

1. **Load file** button is used to load the desired model.. Model must be in *.**gcode** file format.
2. Choose the port printer is connected to computer. (mostly /**usbmodem** for Mac, COM1, COM2, etc for Windows PC).
3. **Print** button starts the printing process.
4. **Disconnect** button disconnects the printer from the computer.
5. Printer controls. Here you can manipulate the printer axes.
6. Setting the nozzle and bed temperatures.
7. Thermometer.
8. Confirming the set temperatures, heating start.
9. 2D print process preview.
10. Info panel. Estimated print time, axis position and other info is displayed after loading the model.

7.2 Printer addons

7.2.1 Different nozzles

E3D, a UK based company, supplies hotends for the Original Prusa i3 MK2 has whole ecosystem of upgrades and addons. We support some of them.

You can check out how to change the nozzle in section [11.4 Replacing / changing the nozzle.](#)

7.2.1.1 Hardened steel nozzle

Hardened steel nozzles are a must for highly abrasive materials. Regular brass nozzles will degrade very quickly and lose their properties.

Most of the abrasive materials are composites, plastics with something mixed in. Some examples are ColorFabb XT CF20, ColorFabb Bronzefill, ColorFabb Brassfill and some glow in the dark filaments. Always ask your filament vendor if you are not sure. Slight disadvantage is that some standard materials like ABS isn't possible to print as fast as with regular nozzle.

7.2.1.2 0.25mm nozzle

To get finer detail on 0.1mm or 0.05mm print settings, you can use 0.25mm nozzle. But use it for only very small objects, only couple centimeters big. The print time can be considerably longer compared to 0.4mm. Ideal use is jewelery.

8 Printer drivers

Latest drivers and information can be found at <http://www.prusa3d.com/drivers/>.

Driver package contains following settings and programs:

Slic3r - preparing the 3D models to .gcode format for printing.

Pronterface - printing from a computer (in case you don't want to print from SD)

NetFabb - repairing the corrupted or unprintable models

Settings - optimized print settings for Slic3r, Cura, Simplify3D and KISSlicer

Drivers for Prusa i3 printer - Windows a Mac drivers

Test objects

9 Printing your own models

9.1 Where you can get the 3D models?

The best way to get started with your own 3D printing is to find already created models on internet - they should be in the **.stl** or **.obj** format . Fortunately there are lot of fans and there are sites from which you can download a wealth of ready-made 3D models - from a simple shaver holder to a detailed aircraft engine model.

3D models are generally 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 with high-quality models:

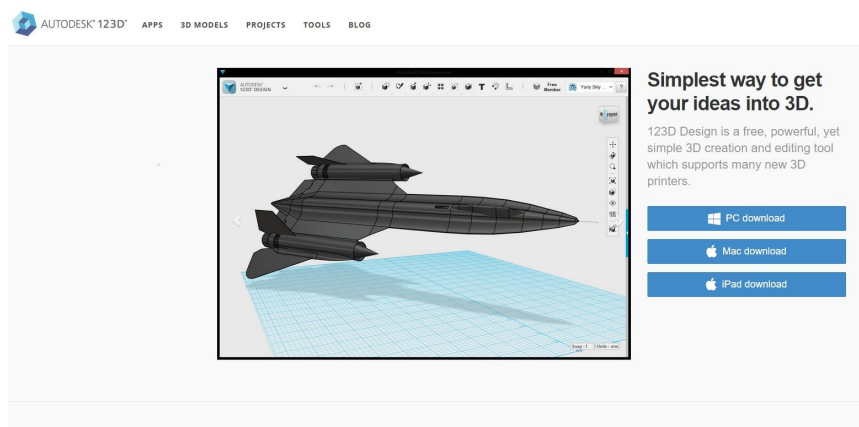
1. <http://www.thingiverse.com/>
2. <https://pinshape.com/>
3. <https://www.youmagine.com/>
4. <http://www.shapeways.com/>
5. <http://www.123dapp.com/>
6. <http://cubify.com/>

9.2 In what program you can create your own models?

To create a 3D model yourself, you need a dedicated program. The easiest way to quickly create a model is TinkerCad (www.tinkercad.com) - an online editor (no installation needed) - you create your 3D model directly in the browser window. It is free, is easy to operate and you will find even basic video tutorials, so after a few minutes nothing prevents you to create your first 3D object.

Other popular tool for creating models is 123D Design (<http://www.123dapp.com/design>) for PC, Mac and iPad. Website provides a quick guide along with detailed video tutorials so it's a very good choice for novice enthusiasts.

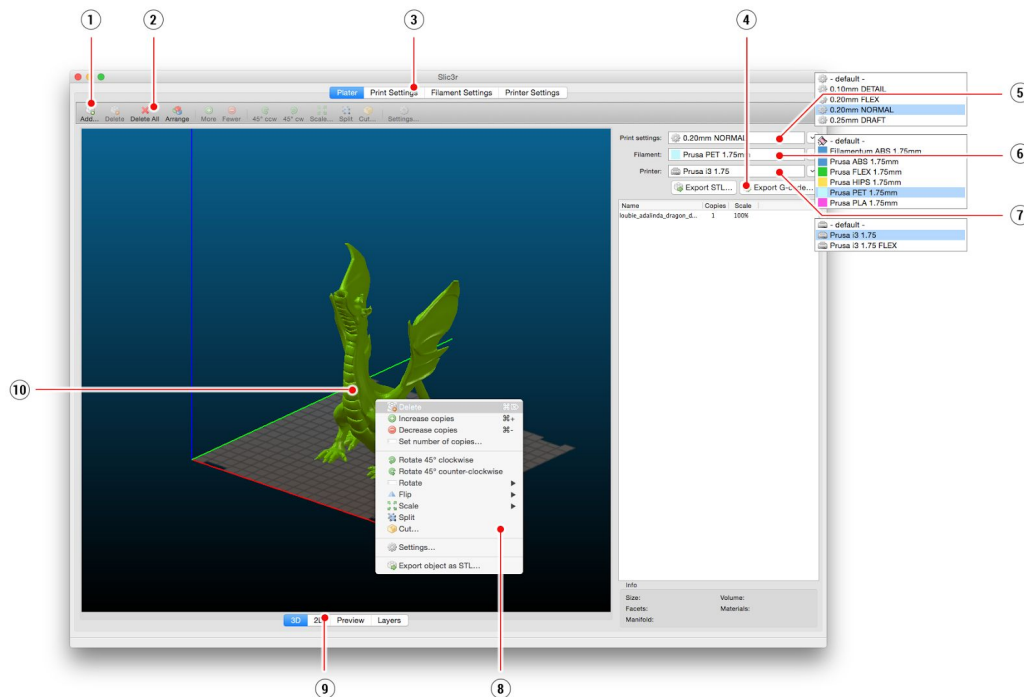
There is a great deal of 3D programs - free or paid - your choice depends more on your personal taste and preferences. The following is a list of other programs used for making 3D models: OpenScad, DesignSpark Mechanical, Fusion 360°, Blender, Maya, 3DS Max, Autocad and many more...



9.3 Slic3r

A 3D printer can print almost anything. Whether you've downloaded 3D models from the Internet or created your own models, you will need to **convert the .obj or .stl format into a .gcode file**. Gcode is a file format readable by a 3D printer. The file contains information for nozzle movement and the amount of filament to extrude. The right tool for this task - and for many more - is the Slic3r program.

You set the printing material, print quality and the print speed in Slic3r. You can manipulate the object here, varying the placement on the printbed, resize it, etc.



Pict. 19 - Slic3r interface

1. **Add** button loads models into Slic3r.
2. **Delete** and **Delete All** buttons remove the model(s) from Slic3r.
3. Opens the detailed settings of print, filament and printer.
4. When the model is ready for print this button generates the **.gcode** file.
5. Quality / Speed setting of a print
6. Material selection
7. Printer selection
8. Right-click on model opens the menu with rotate, resize and other options
9. Type of model preview
10. Model preview

9.4 Bundled 3D models

We asked couple known 3D designers and prepared some printable object for you to print. They are ideal for the first prints on your new printer. STL and GCODE files are available after installing the drivers package in "3D Objects" folder or bundled on your SD card. You can check them out at <http://www.prusa3d.com/printable-3d-models/>.




Pict. 20 - 50 microns treefrog is commonly used as a 3D printing benchmark.

9.5 Print in color with ColorPrint

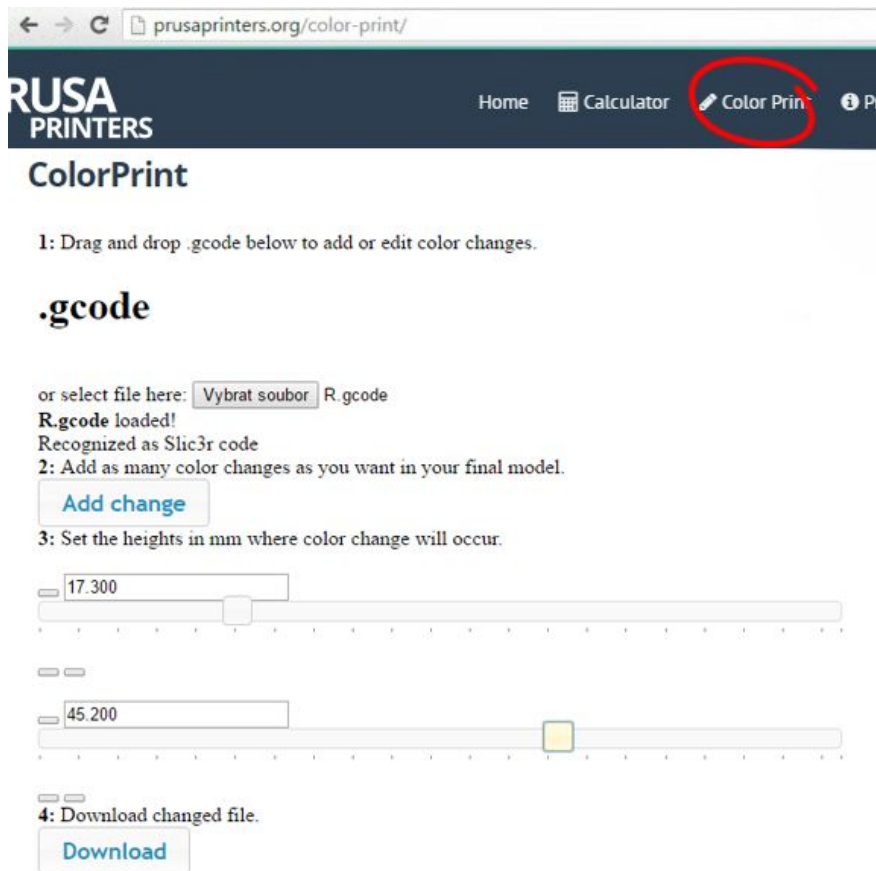
There is a simple way on how to create layer based multicolored 3D prints with our simple ColorPrint app.



Pict. 21- Multicolored object printed with ColorPrint

 ColorPrint will not work correctly if your print settings **contain hop or z-lift**. The app detects layers in gcode by tracking increase in Z height and the hop/zlift does that multiple times in the same layer. Turn off this setting before using ColorPrint. The preconfigured Slic3r in our drivers offers the **Original Prusa i3 MK2 ColorPrint** option in printer selection.

- First of all you need to prepare regular **gcode** with common print and filament settings. Save the file.
- Then go to **www.prusaprinters.org** and choose Color Print in the header menu.
- Drag the gcode to frame and click on **Add change** button.
- Find the **height** of the layer where you want to make the color change. This can be easily found in Slic3r under tab “Layers.” The scale along right side displays the height of individual layers. Set this number to the box. Number of these changes is unlimited.
- When you are done with your modification, download the file. This file is **ready** to be printed!



Insert the filament which you want to start with into your printer and start printing the file.

When the color change is triggered from the gcode the printer will follow simple procedure:

- Stop moving and retract
- Raise the Z by 2 mm and move quickly outside the printbed
- Unload the current filament
- You will get asked to insert the new filament. When you do so and continue, filament will be pulled into the hotend and LCD will display **“Changed correctly?”** with three options:

1. **“Yes”** Everything went ok and printing can continue. Check if the new color is clear without any remains of the previous filament - if yes, choose this option to continue printing with a new color.
2. **“Filament not loaded”** If the new filament was not loaded properly, choose this option and the printer will start the automatic filament load again. When the filament is loaded properly, you can choose the “Yes” option and the printing will continue with a new color.
3. **“Color not clear”** Filament was loaded but the color is still mixed with the previous filament. Press the button with this option and the printer will extrude more filament from the nozzle. When the color is pure without any remains of the previous filament you can choose the “Yes” option and the printing will continue with a new color.

After confirming, printer returns to the original position and continues to print.

i Other options for **multicolored print** is to use the filament change option. Choose the **Tune** and then **Change filament** option during the print. Printer will pause the printing process, unload the filament and signals you to insert the new filament. The procedure is the same as above.

i You should always use the same material or combine materials with similar print temperatures and settings.

9.6 Printing of non-standard models

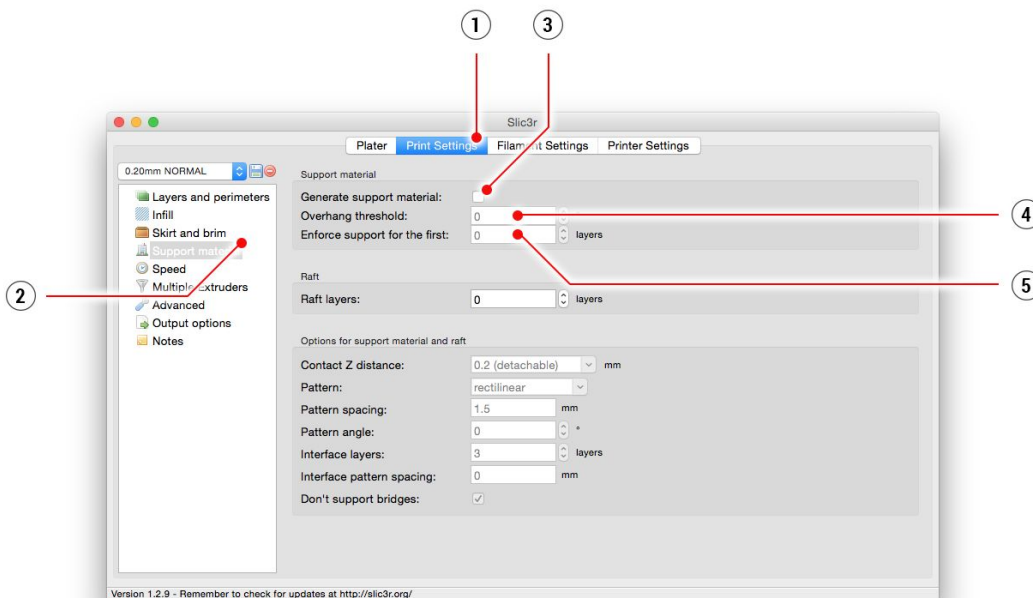
Slic3r helps you while printing the non-standard models as models with overhangs and/or models larger than a printing bed.

9.6.1 Printing with support material

When you print models you can find special cases different from standard printing. The first case is printing with support material.

If you print an object with a gradient lower than 45° the material overhang would be preventing the object to be printed correctly. Slic3r allows you to print such objects thanks to the 'Printing with support' function. Support material is an extra structure printed as scaffolding for the object - you can remove the support material after the printing is finished. Choose the *Print Settings* tab **(1)** and click the *Support Material* option **(2)** in left column. First you have to check the *Generate support material* box **(3)**. Next item - *Overhang threshold* **(4)** 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's needed.

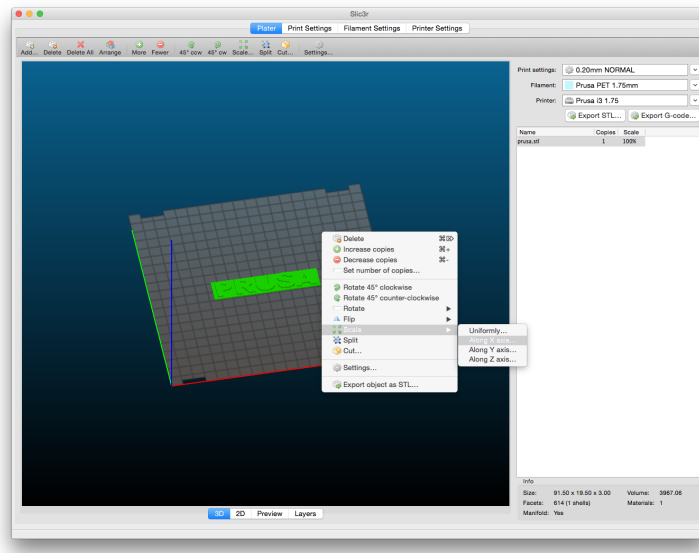
Enforce support option **(5)** is used mostly with small models or models with small base to prevent the object from breaking or tearing out from the bed.



Pict. 23 - Print with support menu

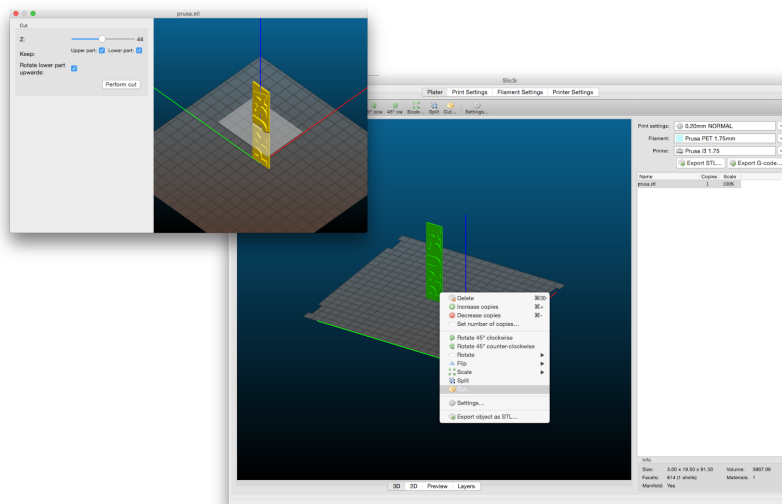
9.6.2 Large object printing

Another special printing case is when printing objects larger than the heatbed. First option is to resize the object to a printable size. Right-click on an object in Slic3r opens a menu with the *Scale...* option, then you choose *Uniformly*, if you want to scale down the model evenly; or you can alter the size of a model along the one of the axes: *Along X, Y, Z axis...*



Pict. 24 - Size change of a printed object

If you need to print an object that doesn't fit the printer, you have to cut the object using Slic3r. Right-click and choose the *Cut...* option in menu. You can cut the object horizontally - if you need to perform a cut in a different axis, use the *Flip...* option in the same menu.



Pict. 25 - Cutting the object with the Cut option

10 Materials

Temperatures and the heatbed treatment before print according to a specific material.

10.1 ABS

Material suitable for common, robust objects.

- **Nozzle temperature:** 255 °C
- **Bed temperature:** 100 °C. You can set the bed temperature between 80 to 110 °C depending the size of an object (larger object means higher temperature)
- **Heatbed:** Make sure the surface is clean as described in [6.3.1 PEI print surface preparation](#) chapter

10.2 PLA

Material suitable for detailed models. PLA is made of renewable sources, mostly from the cornflour and cellulose.

- **Nozzle temperature:** 210 °C
- **Bed temperature:** 50 - 60 °C
- **Heatbed:** Make sure the surface is clean, as described in [6.3.1 PEI print surface preparation](#) chapter

10.3 PET

Material suitable for large objects thanks to its minimal thermal expansion. Universal material suited also for printable mechanical components.

- **Nozzle temperature:** 240 °C
- **Bed temperature:** 80 - 100 °C
- **Heatbed:** Make sure the surface is clean, as described in [6.3.1 PEI print surface preparation](#) chapter

10.4 HIPS

Universal, stable material with good heat resistance suited for printable mechanical components.

- **Nozzle temperature:** 220 °C
- **Bed temperature:** 100 °C. You can set the bed temperature between 80 to 110 °C depending the size of an object (larger object means higher temperature)
- **Heatbed:** Make sure the surface is clean, as described in [6.3.1 PEI print surface preparation](#) chapter

10.5 PP

Polypropylene is a flexible and resistant material suitable for printing of the precise objects requiring the flexibility, firmness and persistence.

- **Nozzle temperature:** 254 °C
- **Bed temperature:** 95 - 100 °C.
- **Heatbed:** The best results are obtained with common scotch tape - just attach the tape directly to the print surface and clean it after the print is finished.

10.6 Nylon (Taulman Bridge)

Nylon is very tough material suitable for mechanical parts.

- **Nozzle temperature:** 240 °C
- **Bed temperature:** 80 - 90 °C.
- **Heatbed:** Use one coat of gluestick. Clean as described after the print.

10.7 Flex

Flex is a very strong and flexible material suitable especially for the flexible prototypes, covers, etc. **Important:** Before you start printing from Flex, clean the nozzle from the previous material - preheat the nozzle and load PLA to remove any other previous material. When loading Flex loosen the extruder (idler) screws. Keep in mind that when printing from Flex the automatic filament exchange function may not work properly.

- **Nozzle temperature:** 230 °C
- **Bed temperature:** 50 °C. You can set the bed temperature up to 65 °C depending the size of an object. (larger object means higher temperature)
- **Heatbed:** Make sure the surface is clean as described in [6.3.1 PEI print surface preparation](#) chapter



11 FAQ - Printer maintenance and print issues

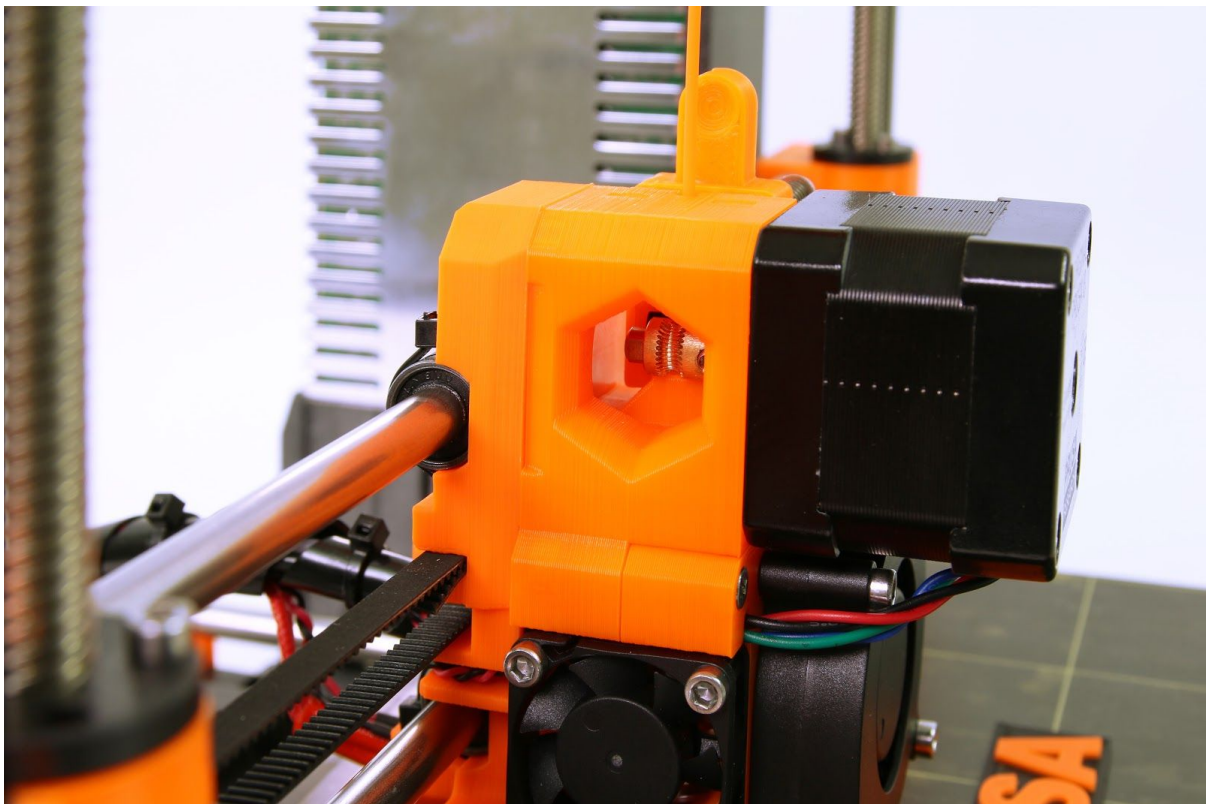
11.1 Print surface preparation

Print surface preparation is described in [6.3.1 PEI print surface preparation](#) chapter.

11.2 Clogged / jammed extruder

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

- Heat the nozzle, remove the filament from the extruder and cut the rod cca 10 cm above the damaged part.
- The next step is to clean the extruder. There is a service hole on the left side of the extruder where you can access the hobbed pulley (pict. 28).
- Clean the hobbed pulley, then heat the nozzle before reloading the filament.
- If a problem persists you will have to clean the nozzle.



Pict. 28 - Cleaning the extruder - you can see the hobbed pulley through the service hole

11.3 Nozzle cleaning

Use a wire brush to clean the nozzle from outside. Heat the nozzle before you do so.

If filament is not extruded from a nozzle (or in very small volume), first check the extruder fan is working properly and that the temperature is set correctly (PLA 210 °C; ABS 255 °C, HIPS 220 °C, PET 240 °C). Also check that the filament was correctly loaded into the extruder. If the filament pours out at least a little, check the direction. If it swirls and goes up to the hotend you need to clean the nozzle.

First you have to move the extruder to the rightmost position, out of the way of the heatbed, to reach the nozzle from below.

Heat the nozzle according to filament you want to print from, load the filament and put a thin wire or a needle (0,3-0,35 mm) into the nozzle from below - between 1 and 2 cm deep.

Choose **Load filament** option from the LCD menu and check if filament is extruded properly. Put the thin wire to the nozzle again and repeat these steps a few times more. When the filament is extruded properly, the nozzle is clear.

11.4 Replacing / changing the nozzle

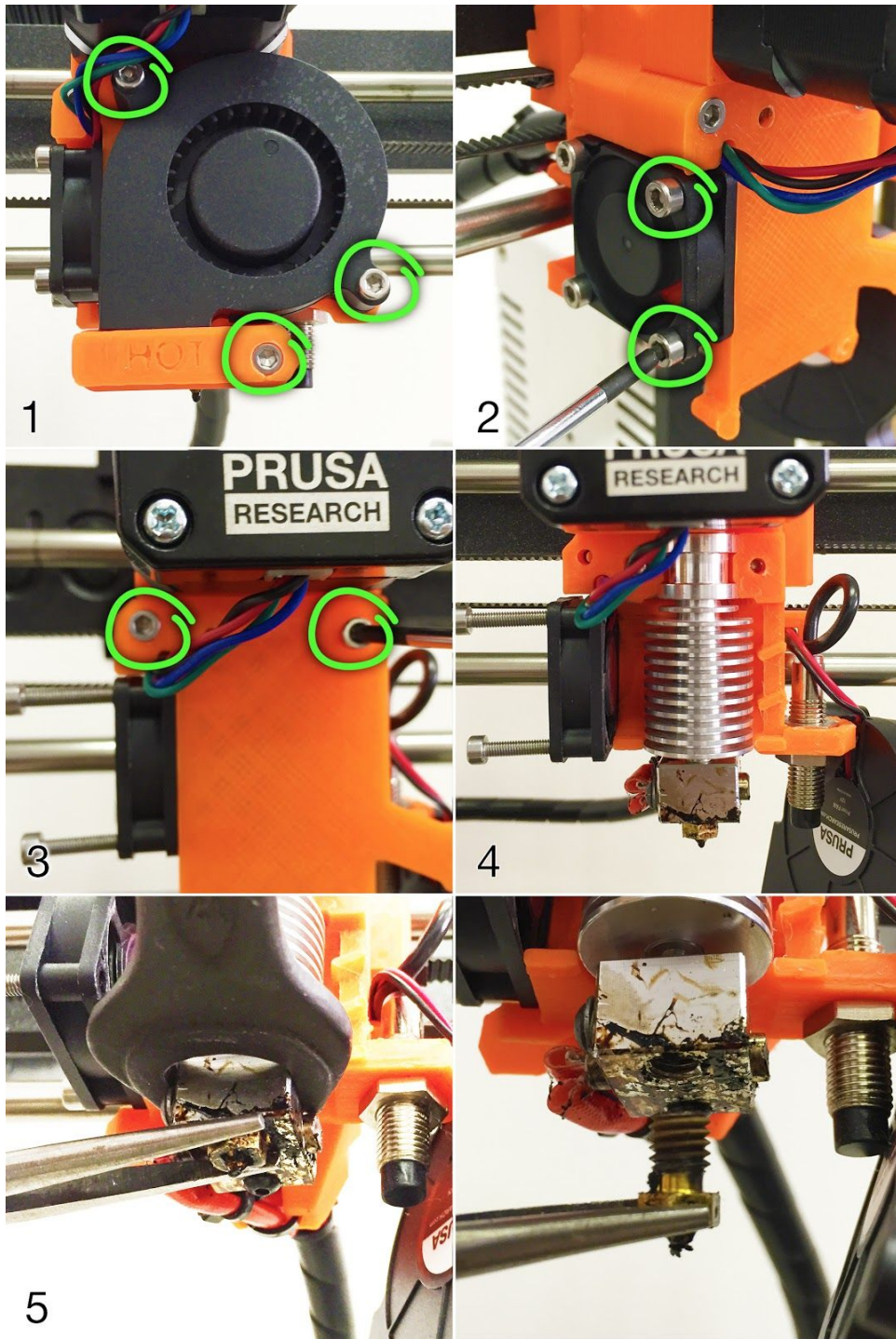
Preheat the nozzle (LCD menu -> Settings -> Temperature -> Nozzle) and set the temperature to at least 200°C. Heating the nozzle is crucial for removing the old nozzle and putting in the new one.

- 1) Move the extruder body upwards to get to the nozzle end (LCD menu -> Settings -> Move axis -> Move Z -> Set the height by rotating the LCD knob and then confirm).
- 2) Unscrew the screw on **fan mouthpiece** and the two screws on the **print fan** and remove both parts (**Pict. 29, part 1**).
- 3) Remove the two front screws on the **nozzle fan** (**Pict. 29, part 2**).
- 4) Unscrew the two screw holding the **extruder cover** (**Pict. 29, part 3**). Even though the nozzle itself is accessible directly, we recommend to take the extruder cover to get access to the heating element.
- 5) Now the whole nozzle body is accessible (**Pict. 29, part 4**).
- 6) Hold the heating element with a spanner (size 17) and unscrew the nozzle (**Pict. 29, part 5**). **Be careful, the nozzle is still hot!**

When the new nozzle is inserted, tighten it while the nozzle is preheated. While tightening do not forget to hold the heating element with the spanner. Re-assemble the extruder, insert the filament and you are ready to print.



Be careful, the nozzle is hot during this whole process and can cause burns!
Be careful around the hotend thermistor leads, you can break them easily.
Be careful and don't apply force to the nozzle or heater block, you can bend the heatbreak easily.



Pict. 29 - Nozzle change

11.5 Axis fluency

- If one of the axes doesn't move fluidly you can apply a small amount of oil (e.g. all purpose machine oil). This will lower the friction.

11.5 Printing problems

11.5.1 Layers break and split when printing from ABS material

ABS material has a higher thermal expansion than other materials. We suggest other materials as PET, HIPS or PLA when you print larger models.

11.5.2 Models contain either too much or not enough of the filament

You can manage the filament flow during the print. Use the LCD-knob and choose **Tune - Flow - xx%** where you can adjust the filament flow. Pronterface users can enter the value M221 Sxx into the command line.



When you change the filament flow next print will use the same settings unless you change it again in menu or you reset the printer or unplug it from the power source.

11.6 Problems with finished models

11.6.1 Model breaks and/or is easily damaged

A typical feature of larger models printed from ABS. If you have set the temperature properly, the printer is away from drafts and object design is right, the printed object should not break. The easiest way to avoid breaking or overall model fragility is to choose a different material. The strongest are PET, HIPS and PLA; while PLA has low heat resistance, PET is the firmest and has the lowest thermal expansion.

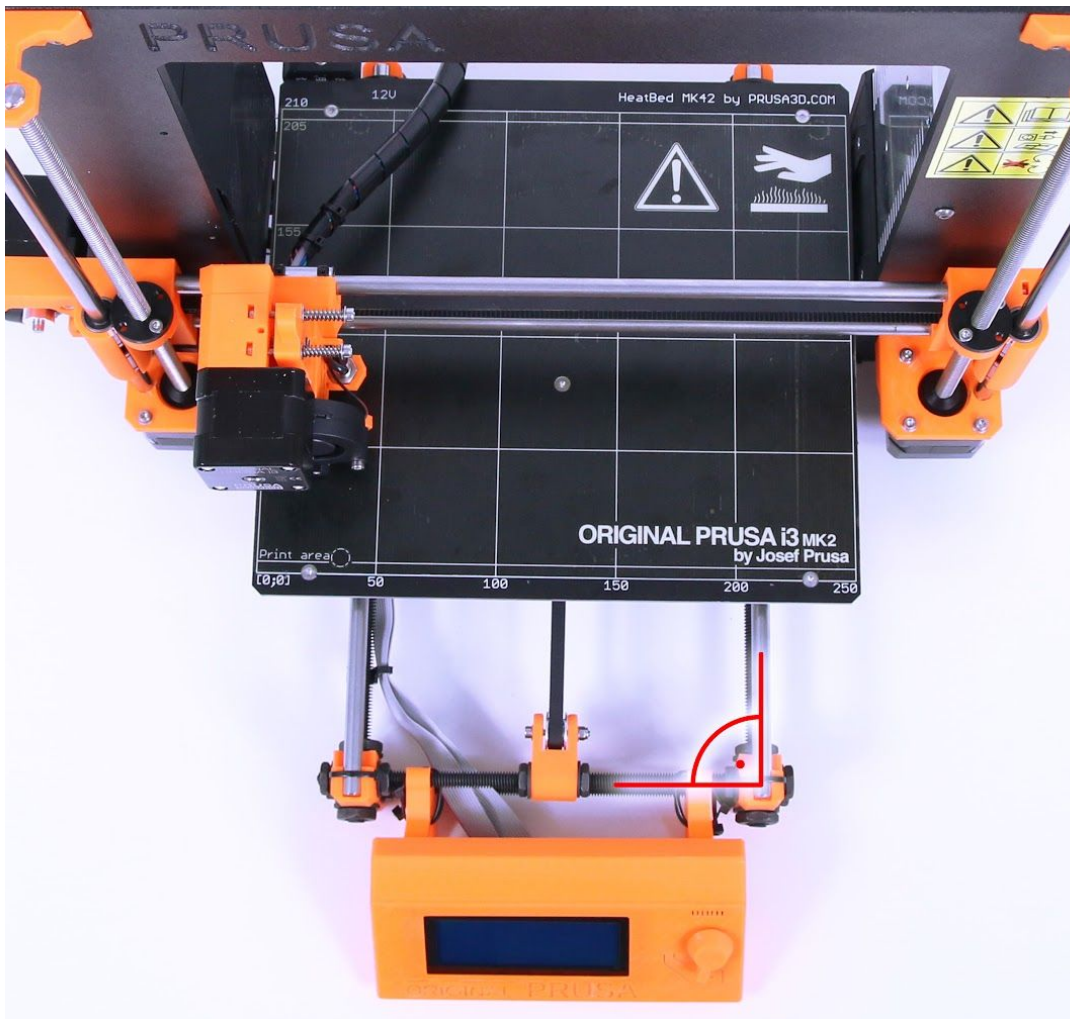


12 FAQ - common issues when assembling the printer kit

12.1 Nozzle/print surface gap is greater in the middle than at the corners

The reason of this issue isn't the bent printer surface or bed but a distorted Y-axis. We suggest to remove the whole Y-axis out of the printer. Follow these steps:

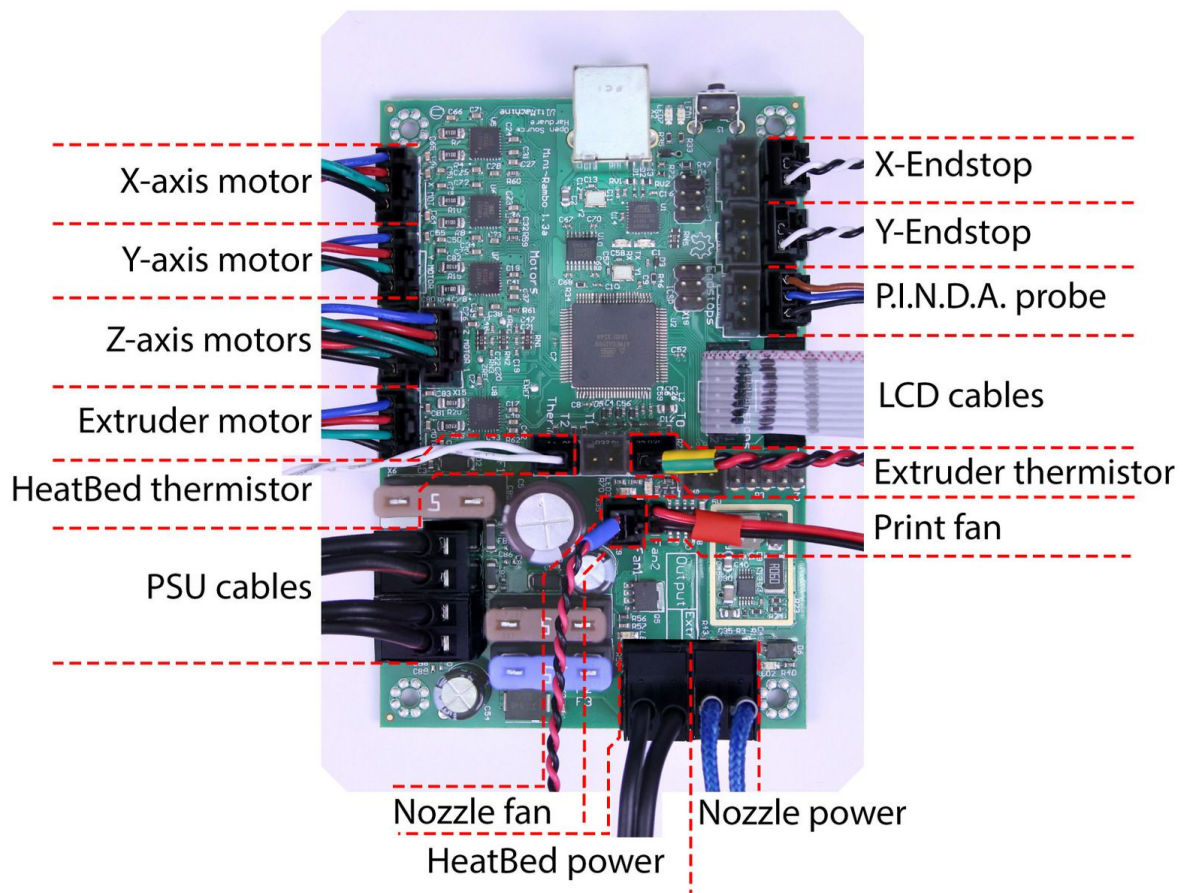
- Align the Y-axis so that each Y-corner is sitting on the table - none should be off the table.
- Align the tightening of each Y-axis threaded rod so that each Y-corner is perpendicular (facing upright) to the table surface.
- Align the tightening of each Y-axis threaded rod so that each M8 threaded rod is at a right angle to the M10 threaded rods - the Y-axis must form a perfect rectangle when viewed from above (pict. 30).



Pict. 30 - Right angle between Y-axis and M8 / M10 rods

12.2 Printer stops printing soon after start

Extruder is likely overheated. Make sure the nozzle fan is working properly. If not, please inspect its connection according to the assembly manual.



Pict. 31 - Proper wiring of the connectors

12.3 Printer can't read SD card

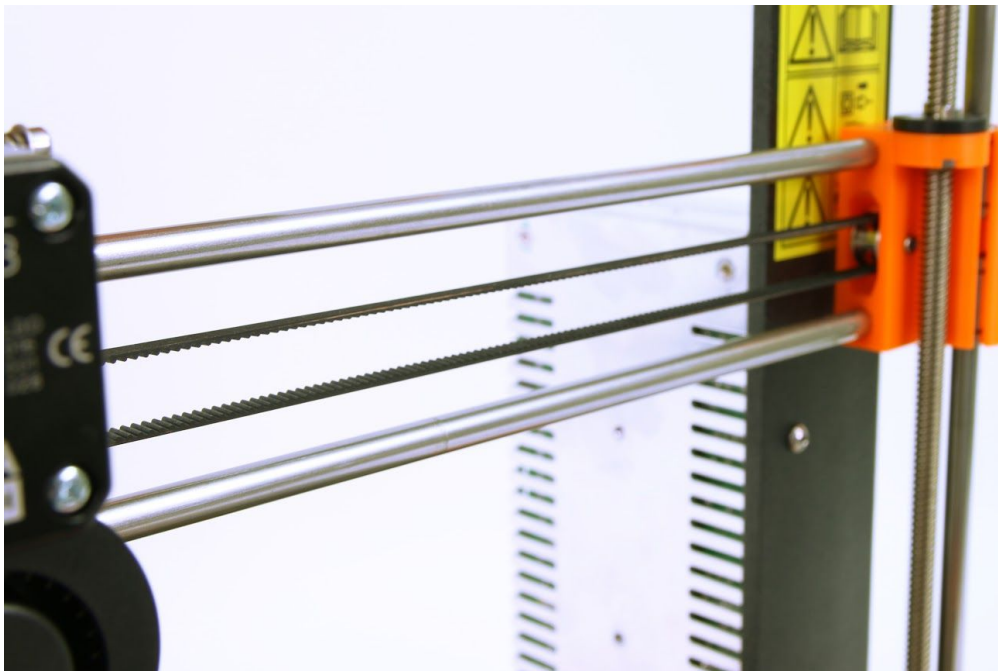
First, make sure that the **file name** on the SD **does not contain special characters** - otherwise the file could not be displayed on LCD. If there is no error in the file name, check the EXT2 wiring (from electronics to LCD). If the cable is connected properly, try to swap the cables.

12.4 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 extruder. See the pictures with properly tightened belts.



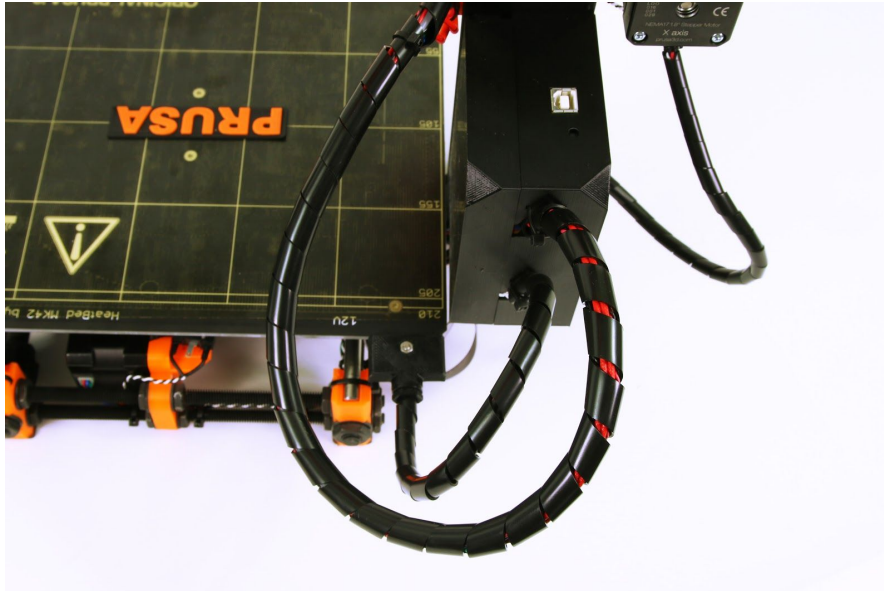
Pict. 32 - A properly tightened Y-axis belt under the heatbed



Pict. 33 - A properly tightened X-axis belt

12.5 Detached cables to the heatbed

Do not forget to use a spiral wrap on heatbed cables and attach the cables properly so they won't restrict movement during printing.



Pict. 34 - Cables to be wrapped in a spiral wrap

12.6 Loose heatbed zipties

Make sure that the zipties under the heatbed are properly tightened. There are three zipties in total, two on the right side, one on the left.



Pict. 35 - Properly tightened zipties under the heatbed

