# 3D PRINTING HANDBOOK

## USER MANUAL FOR 3D PRINTERS:

- ORIGINAL PRUSA i3 MK2S KIT 1.75MM
- ORIGINAL PRUSA i3 MK2S 1.75MM


PRUSA RESEARCH by JOSEF PRUSA

**PRUSA RESEARCH S.R.O.**
Partyzánská 188/7A
170 00 Praha
www.prusa3d.com
info@prusa3d.com

**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



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

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

This symbol indicates text related to a printer kit only.

# 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 the 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 to all users.

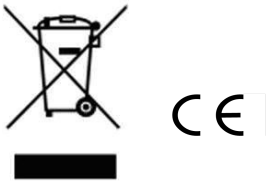
Josef Prusa also organizes workshops for the public, participates in professional conferences dedicated to the popularization of 3D printing. For example, he lectured at the TEDx conference in Prague and Vienna, at World Maker Faire in New York, Maker Faire in Rome or at the Open Hardware Summit hosted 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 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

# 2 Product details

Title: Original Prusa i3 MK2S / Original Prusa i3 MK2S (kit), Filament: 1.75 mm
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: 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 frame and also on the packaging.

# 3 Introduction

Thank you for purchasing our original 3D printer **Original Prusa i3 MK2S** 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. **Original Prusa i3 MK2S** is a successor to Original Prusa i3 MK2 with small hardware tweaks for easier assembly and improved reliability.

**Please check the 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*.We are glad to receive all your valuable comments and tips. We strongly suggest you 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** - A commonly used term for printing pad - a heated area of the 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 a spool is called "filament", it's used throughout this handbook as well as in the LCD menu on the printer.
**Heater, Hotend** - another 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, expressed 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. 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.
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

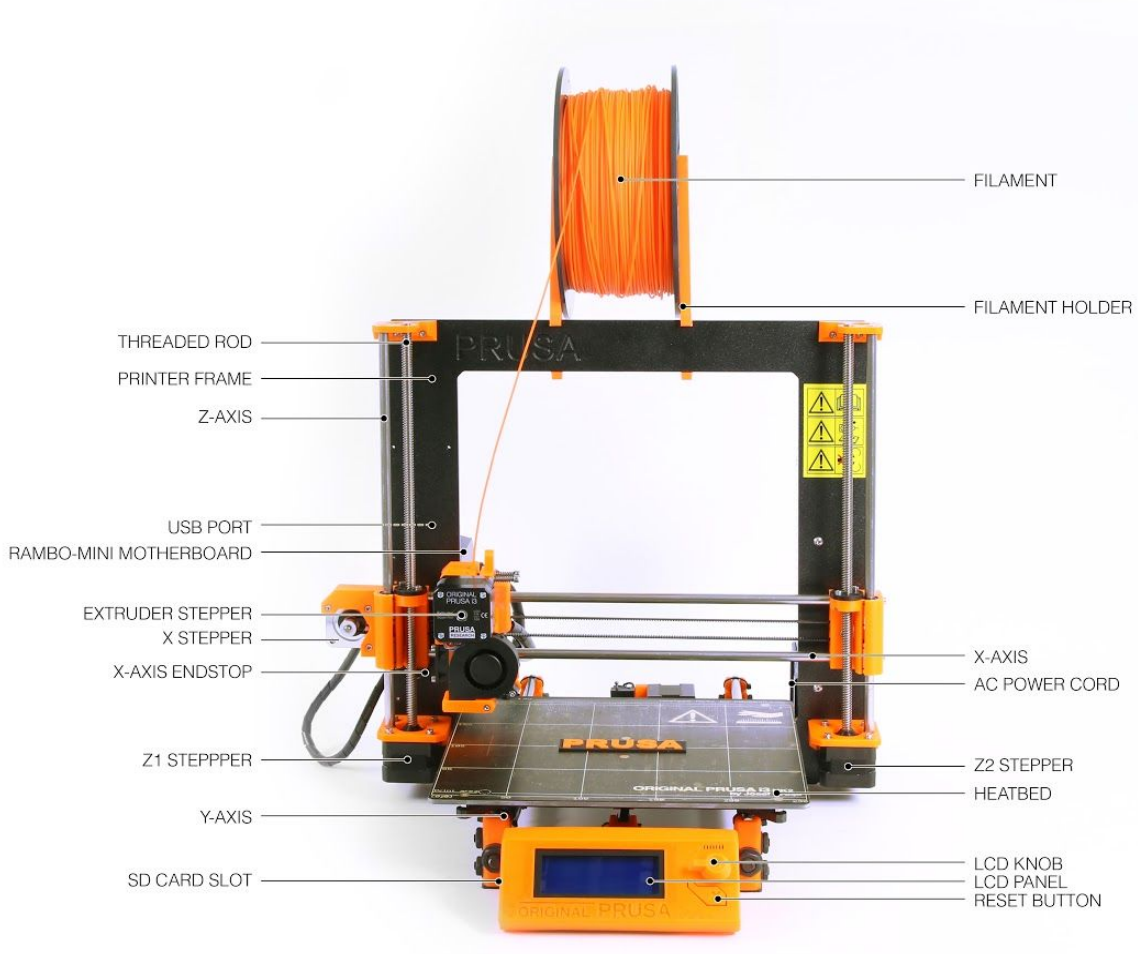
Original Prusa i3 MK2S printer is a part of the RepRap project, the first open source 3D printer project free to use under a GNU GPL v3 license (www.gnu.org/licenses/gpl-3.0.en.html). If you improve or alter any part of a printer and you are willing to sell, then you have to publish the source codes 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 MK2S 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 a 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. The 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 MK2S printer description*

# 5 Original Prusa i3 MK2S printer kit

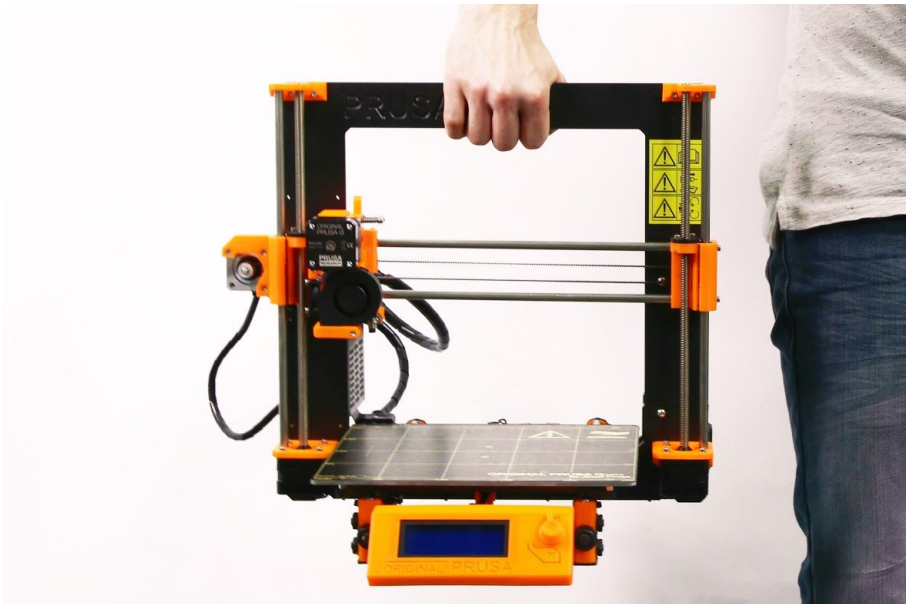
Original Prusa i3 MK2S kit is pictured in pict. 2. Detailed information and assembly description can be found in chapter 6.2 Printer assembly. We offer the support for users who purchased the printer kit through our official forum. If you need help do not hesitate to visit our forum at forum.prusa3d.com. You can find the answers to your problem there. If not, please just post your question directly there.


*Pict. 2 - Original Prusa i3 MK2S 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. When unpacking **the fully assembled version,** remove the top foam from the box and gently lift the printer up. Parts of the printer are secured by more foam which needs to be removed. Some parts are additionally secured with the white zip-ties, cut those off too.



*Pict. 3 - Proper handling of a printer*

Both the assembled version and the kit version come with a few things you might need during the printer use.

- **USB Cable** - used for uploading a new firmware or alternatively printing from the computer.
- **Acupuncture needle** - used for cleaning the nozzle when stuck. See the chapter 11.4 Nozzle cleaning for more information.
- **Glue stick** - Used for better Nylon adhesion or as a separator for Flex materials. See the chapter 11 Materials for more information.
- **Test protocol** - All the components of every printer are tested. The electronic parts are even connected as in a final assembly and battery of tests is ran. Only when all tests pass the electronics get a serial number and protocol + S/N stickers are printed. Test protocol shows all the test results of your printer components.

## 6.2 Printer assembly

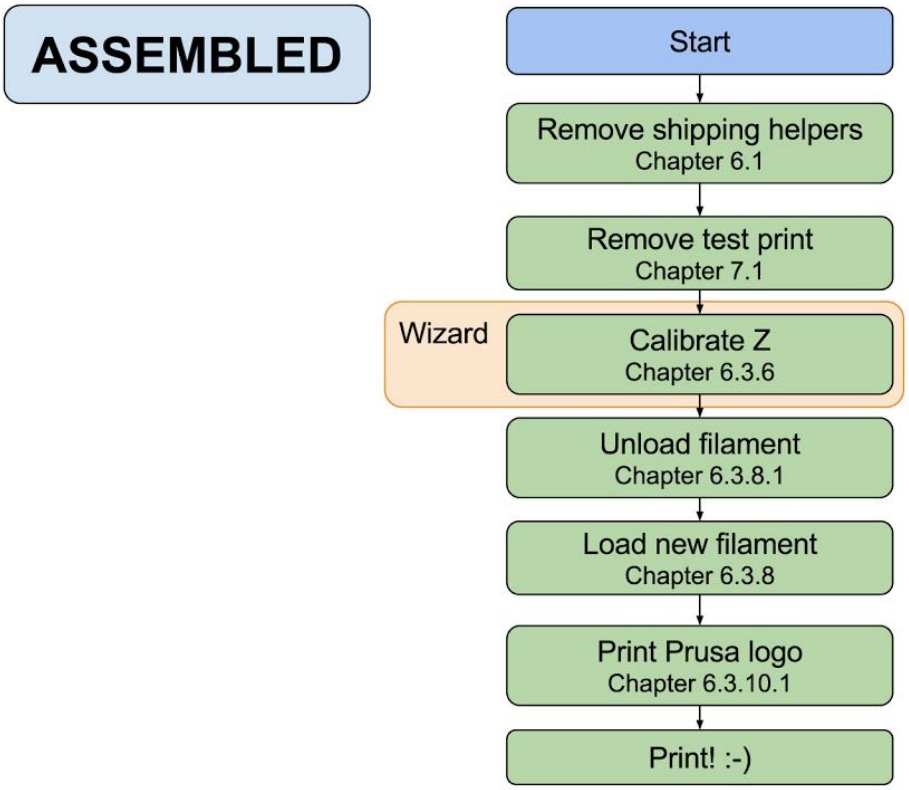
With Original Prusa i3 MK2S printer kit we suggest to follow the guidelines and assemble the kit according to the the online manual at manual.prusa3d.com. (Online manual is available in several languages on the website). The construction of the printer should not take more than one working day. After a successful completion continue to the chapter 6.3 Setup before printing.

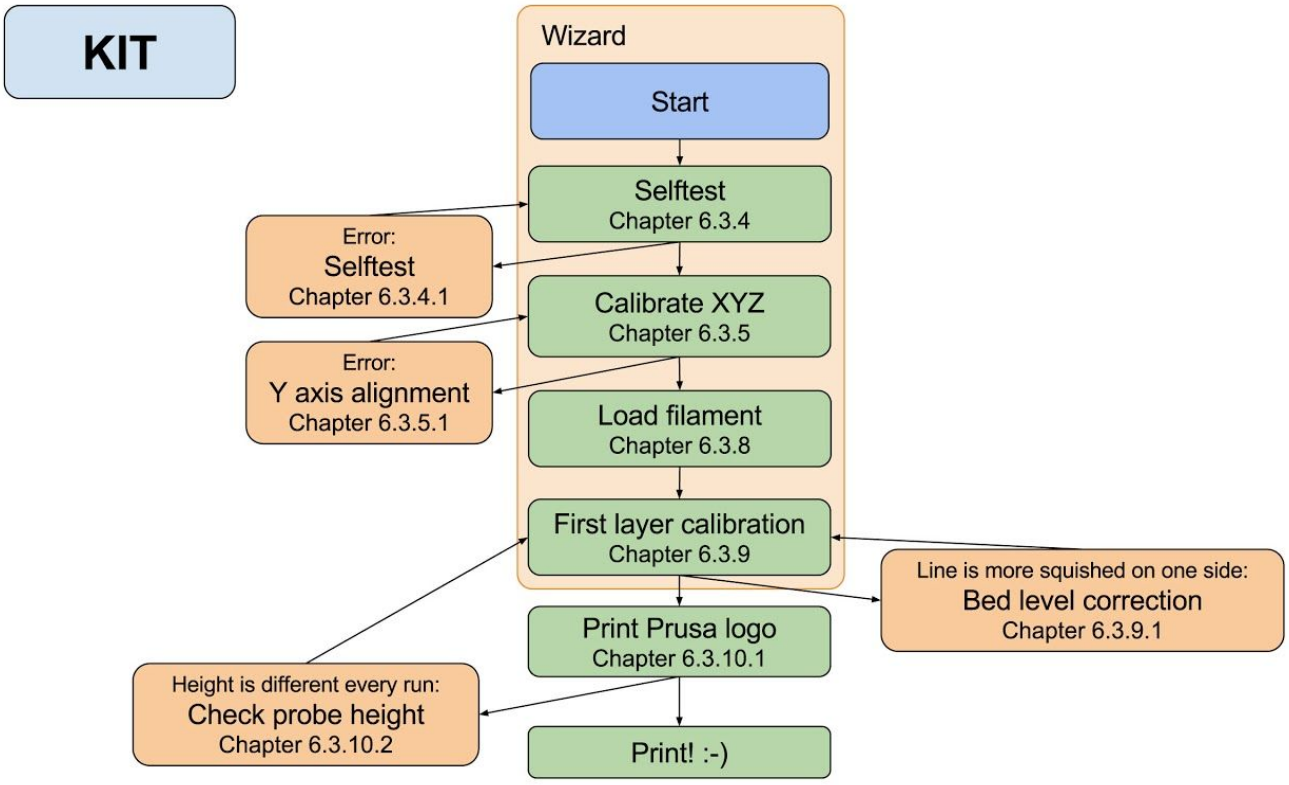
## 6.3 Setup before printing

- 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.
- 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.
- Check out the firmware version (in the Support menu via LCD panel) and please upgrade to the latest one from our website www.prusa3d.com/drivers.

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

### 6.3.1 Calibration flow and wizard






**Wizard is available since firmware version 3.1.0 onward.**

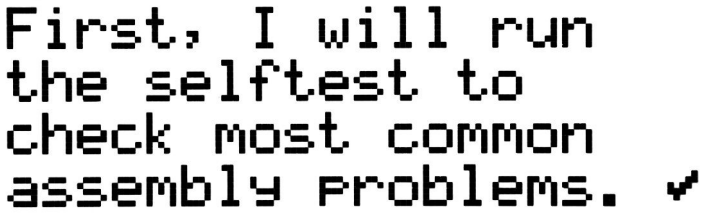
With your first start-up of your freshly assembled printer, it will guide you through all the tests and calibrations you need to do to get started printing.

Wizard can be also started manually from LCD menu **Calibration -> Wizard**. Do not forget to read chapter 6.3.2 PEI print surface preparation before running the Wizard.

It follows the calibration flow and helps you with following steps:

- **Selftest** - Chapter 6.3.4
- **Calibrate XYZ** - Chapter 6.3.5
- **Loading the filament** - Chapter 6.3.8
- **First layer calibration** - Chapter 6.3.9

It is not mandatory to use it, and you can cancel the wizard at the beginning. Then you should just manually follow the calibration flow as on older firmware revisions.




*Pict. 4 - Wizard setup*

There are few special occasions where you will need to redo the calibration or part of it.

- **Firmware update** - Complete guide is in the chapter 12.8 Updating printer firmware. 6.3.9. First layer calibration needs to be rerun otherwise the printer will show an error message.
- **Replacing the PEI/Ultem sheet** - When PEI is changed (guide is at manual.prusa3d.com), whole heated bed is removed and reassembled. This might change geometry of the printer and the whole **KIT calibration flow** should be followed even on the pre assembled printer.
- **Readjusting the P.I.N.D.A. probe** - Run 6.3.6 Calibrate Z to store new reference Z height values.

It is important to disconnect the printer USB from any computer or OctoPrint running on Raspberry Pi for the whole calibration. Printer will not respond to any request from the host and communication will timeout, when host resets the connection, the printer restarts and might end up in weird state requiring 7.2.4 Factory Reset.

### 6.3.2 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 (except for PETG where the adhesion may be too strong. See the chapter 11.3 PET for instructions). Pour a little amount on unscented paper towel and wipe the print surface. The bed should be cleaned while cold for the best results but it can also be cleaned when already preheated for PLA, just be careful not to touch the bed surface or the nozzle. When cleaning at higher temperatures the alcohol will evaporate before it can clean anything. 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.

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.

### 6.3.3 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 or PrusaControl 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.

### 6.3.4 Selftest (kit only)

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 **Calibration** 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.

The selftest is just a diagnostic tool, the printer will still attempt to print even after the test fails. If you are absolutely certain that the affected part is correct, you may continue with the print process.

Test consists of

- **Extruder and print fan** test
- **Heatbed** and **hotend** proper wiring
- **XYZ motors** proper wiring and functionality
- **XYZ endstop** mechanical settings and proper wiring.
- **Loose belt pulley** test.

#### 6.3.4.1 Selftest error messages and resolution (kit only)

*Fan - Wiring error:*

Check proper wiring of print and hotend fan cables. Ensure that both are properly connected to the Rambo electronics, and that they are not swapped.

*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 minimal position.

*Loose pulley - {XY}:*

The belt pulley is loose and slips on the motor shaft. It is important to tighten the first grub screw on the flat piece of shaft, then continue with the second grub screw.

### 6.3.5 Calibrate XYZ (kit only)

The Original Prusa i3 MK2S 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 (**P**rusa **IND**uction **A**utoleveling) probe.

The process is fairly straightforward, so let's get to it. We also suggest to check out our Guide for New User first at www.prusa3d.com/buildvideomk2 with calibration tips.

The purpose of the X/Y/Z calibration routine is to measure the skew of the X/Y/Z axes and to find the position of the 9 calibration points on the print bed for the proper bed leveling. You can run the **XYZ calibration** from **Calibration** menu on LCD panel. This should not be necessary on the assembled printers as those are factory calibrated.

Place a sheet of a regular office paper (for example the checklist shipped with every order) and hold it under the nozzle during the first round (first 4 points being checked) of calibration. If the nozzle catches on the paper during the process, power off the printer and lower the P.I.N.D.A. probe slightly. See the P.I.N.D.A. probe response diagram in **6.3.10.2 Check probe height**. The paper will not affect the calibration process. The nozzle must not touch the print surface or deflect the bed by any means. If everything went correctly, continue with the calibration process.

Initiating this routine performs a series of measurements in three rounds: In the first round, 4 sensor points on the print bed are searched for carefully so as not to touch the print bed by the nozzle. In the second round, all 9 sensor points are found. In the last round the height above the 9 sensor points is measured and stored into a non-volatile memory for reference, this finished the Z axis calibration.

The progress and results of each step are displayed on the LCD. In case of errors found, the XYZ calibration is interrupted and the reason for error is shown to guide in troubleshooting.

At the start of the XYZ calibration procedure the printer prompts you by a following message: ***"Calibrating X/Y. Move Z carriage up to the end stoppers. Click when done."***

After that, the printer asks you to confirm this step: ***"Are left and right Z carriages all up?"***

Please make sure you really move the Z carriage up to the end stoppers until you hear a rattling sound as the Z stepper motors skip steps. This procedure ensures, that 1) the X axis is perfectly horizontal, 2) the print nozzle is in a known distance from the print bed. In case the Z carriage did **not** touch the end stoppers, the printer could not possibly know the height of the print nozzle above the print bed and it could, therefore, crash into the print bed during the first round of the X/Y calibration procedure.

The XYZ calibration procedure also prompts you to ***"Please clean the nozzle for calibration. Click when done."***

If this advice is not followed and there is a plastic debris on the print nozzle, then the debris may touch the print bed or even push the print bed away from the PINDA probe, so the PINDA probe will not trigger properly and the calibration will fail.

After the calibration is passed, the values can be reviewed for tweaking later. When you get your axes **perpendicular** or **slightly skewed**, nothing needs to be tweaked as printer will perform with the best accuracy. Learn more in chapter 8.3 View XYZ calibration details (Optional) under 8 Advanced calibration.

#### 6.3.5.1 Calibrate XYZ error messages and resolution (kit only)

1) ***XYZ calibration failed. Bed calibration point was not found.***

   Calibration routine did not find a bed sensor point. The printer stops close to the bed point, which it failed to detect. Please verify, that the printer is assembled correctly, that all axes move freely, the pulleys do not slip and the print nozzle is clean. If everything looks good, re-run the X/Y calibration and verify with a sheet of paper between the nozzle and the print bed that the print nozzle does not touch the print bed during the calibration routine. If you feel a friction of the nozzle against the sheet of paper and the nozzle is clean, you need to screw the PINDA probe slightly lower and re-run the X/Y calibration.

2) ***XYZ calibration failed. Please consult the manual.***

   The calibration points were found in positions far from what should be expected for a properly assembled printer. Please follow the instructions of case 1).

3) ***XYZ calibration ok. X/Y axes are perpendicular. Congratulations!***

   Congratulations, you built your printer precisely, your X/Y axes are perpendicular.

4) ***XYZ calibration all right. X/Y axes are slightly skewed. Good job!***

   Good job, the X/Y axes are not precisely perpendicular, but still quite all right. The firmware will correct for the X/Y skew during normal printing, so boxes will be printed with right angles.

5) ***XYZ calibration all right. A skew will be corrected automatically.***

   You may consider to re-align the X/Y axes (as described in the chapter **6.3.5.2 Y axis alignment**). Still the firmware will correct the skew during normal printing and as long as the X and Y axes move freely, the printer will print correctly.

6) ***XYZ calibration failed. Left front calibration point not reachable.***

   Even if the printer moves the print bed to the end Y end stop, the PINDA probe cannot reach the left front bed calibration point. Move the left Y threaded rod in the Z frame away from you, so the PINDA probe reaches the left front bed calibration point reliably. **You can find how to fix this in the next chapter 6.3.5.2 Y axis alignment.**

7) ***XYZ calibration failed. Right front calibration point not reachable.***

   Even if the printer moves the print bed to the end Y end stop, the PINDA probe

cannot reach the right front bed calibration point. Move the right Y threaded rod in the Z frame away from you, so the PINDA probe reaches the right front bed calibration point reliably. **You can find out how to fix this in the next chapter 6.3.5.2 Y axis alignment**.

8) ***XYZ calibration failed. Front calibration points not reachable.***

   Even if the printer moves the table to the end Y end stop, the PINDA probe cannot reach the front row of the bed calibration points. Move both left / right Y threaded rods in the Z frame away from you. **You can find out how to fix this in the next chapter 6.3.5.2 Y axis alignment.**

9) ***XYZ calibration compromised. Left front calibration point not reachable.***
   ***XYZ calibration compromised. Right front calibration point not reachable.***
   ***XYZ calibration compromised. Front calibration points not reachable.***

   Printer will likely work, but the bed leveling may be compromised and the skew of the X/Y axes may not be fully corrected. It is recommended to adjust the positions of the Y threaded rods in the Z frame as in cases 6) to 8). **You can find out how to fix this in the next chapter 6.3.5.2 Y axis alignment.**

During the mesh bed leveling procedure, the following errors may be reported on the display.

1) ***Bed leveling failed. Sensor disconnected or cable broken. Waiting for reset.***

   Verify, whether the PINDA probe cable is plugged into the RAMBo board correctly. If it is the case, the PINDA probe is broken and it needs to be replaced.

2) ***Bed leveling failed. Sensor didn't trigger. Debris on nozzle? Waiting for reset.***
   This is a safety check to avoid the nozzle to crash into the print bed if the PINDA sensor stops working or something goes wrong with the printer mechanics (for example, a pulley slips). This safety check may be triggered as well, if the printer has been moved to an uneven surface. Before doing anything else, make the Z axis level by going all the way up and try again.

   At the end of the X/Y calibration, the printer measures the reference height above each of the 9 bed sensor points and stores the reference heights into a non-volatile memory. During the normal bed leveling, it is expected that the PINDA probe triggers not further than 1 mm from the reference value, therefore the nozzle is not allowed to move more than 1 mm below the reference value during the bed calibration.

   If you moved the printer, you may need to re-run the Z calibration to sample new reference Z height values reflecting the twist and bend of the table surface the printer is sitting on. If that does not help, please verify, that the PINDA probe is aligned with the sensor points on the print bed during the bed Z calibration. The alignment shall be ensured by the automatic X/Y calibration routine. If the PINDA probe is no more

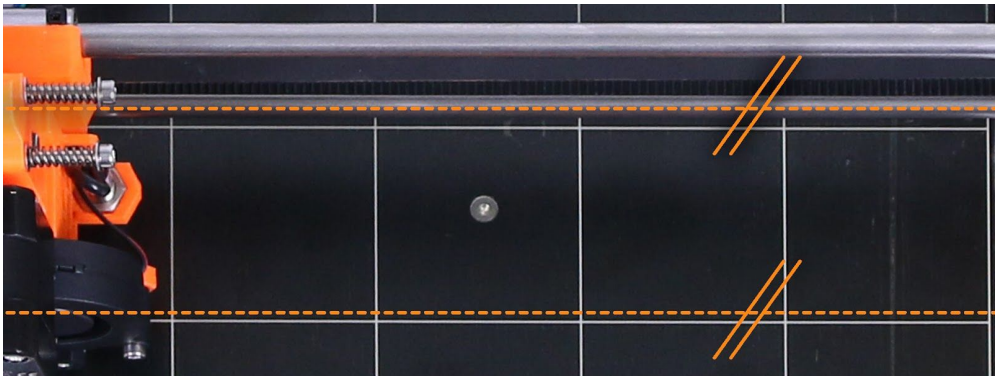
aligned during the Z calibration over time, it is possible, that a pulley is slipping or something on the machine frame got loose.

3) ***Bed leveling failed. Sensor triggered too high. Waiting for reset.***

Similar to case 2). This time the PINDA sensor triggered more than 1 mm above the reference height. Before doing anything else, make the Z axis level by going all the way up and try again.

#### 6.3.5.2 Y axis alignment (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. 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. You can see how in the **Assembly Manual 7.PSU/Step 20** (Check if everything is correct ) or in P.I.N.D.A. Probe Misaligned topic at help.prusa3d.com.



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

If the Calibrate XYZ gives "***XYZ calibration compromised. Front calibration points not reachable.***" error, you don't necessarily need to adjust the M10 nuts and position of the axis in the frame, but you can adjust the belt holder position. **Y-belt-holder with a slot allows to make a 1 mm adjustment to both the front and the back.** Loosen the Y belt holder screws, push the belt holder part towards the Y motor and retighten the screws.

### 6.3.6 Calibrate Z

**Calibrate Z** is located in **Calibration** menu. It should be performed whenever you move the printer to different location. It saves the heights of all 9 calibration points in non-volatile memory. Stored information is used every time mesh bed leveling is called during a print. When the measured values are vastly different to the stored value, print is canceled as it is a good indicator something is wrong. Calibrate Z is a part of Calibrate XYZ routine so there's no need to run it after successful Calibrate XYZ.

It is a good practice to run this procedure every time you travel or printer is shipped as the geometry might change slightly and cause an error.

At the start of the Z calibration procedure the printer prompts you by a following message: ***"Calibrating Z. Move Z carriage up to the end stoppers. Click when done."***

After that, the printer asks you to confirm this step: ***"Are left and right Z carriages all up?"***

Please make sure you really move the Z carriage up to the end stoppers until you hear a rattling sound as the Z stepper motors skip steps. This procedure ensures, that 1) the X axis is perfectly horizontal, 2) the print nozzle is in a known distance from the print bed. In case the Z carriage did **not** touch the end stoppers, the printer could not possibly know the height of the print nozzle above the print bed and it could therefore crash into the print bed during the Z calibration procedure.

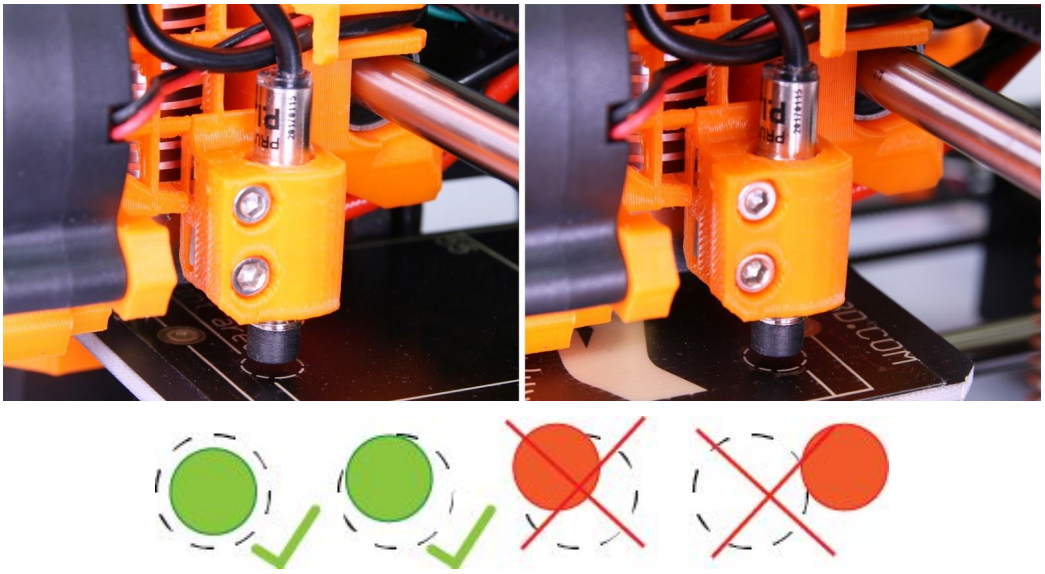
The Z calibration procedure also prompts you to ***"Please clean the nozzle for calibration. Click when done."***

If this advice is not followed and there is a plastic debris on the print nozzle, then the debris may touch the print bed or even push the print bed away from the PINDA probe, so the PINDA probe will not trigger properly and the calibration will fail.

### 6.3.7 Mesh bed leveling

Mesh bed leveling can be found in **Calibration** menu. It is the same procedure which is performed before every print. You can use it to check the P.I.N.D.A. probe alignment with the calibration points however it is not necessary during the calibration process as Mesh bed leveling is a part of Calibrate XYZ and Calibrate Z routines.

**Before running this command clean the nozzle tip from any debris and run it while the nozzle is cold. If the probe is misaligned and the nozzle is preheated, the nozzle can make permanent impressions in the print surface.**



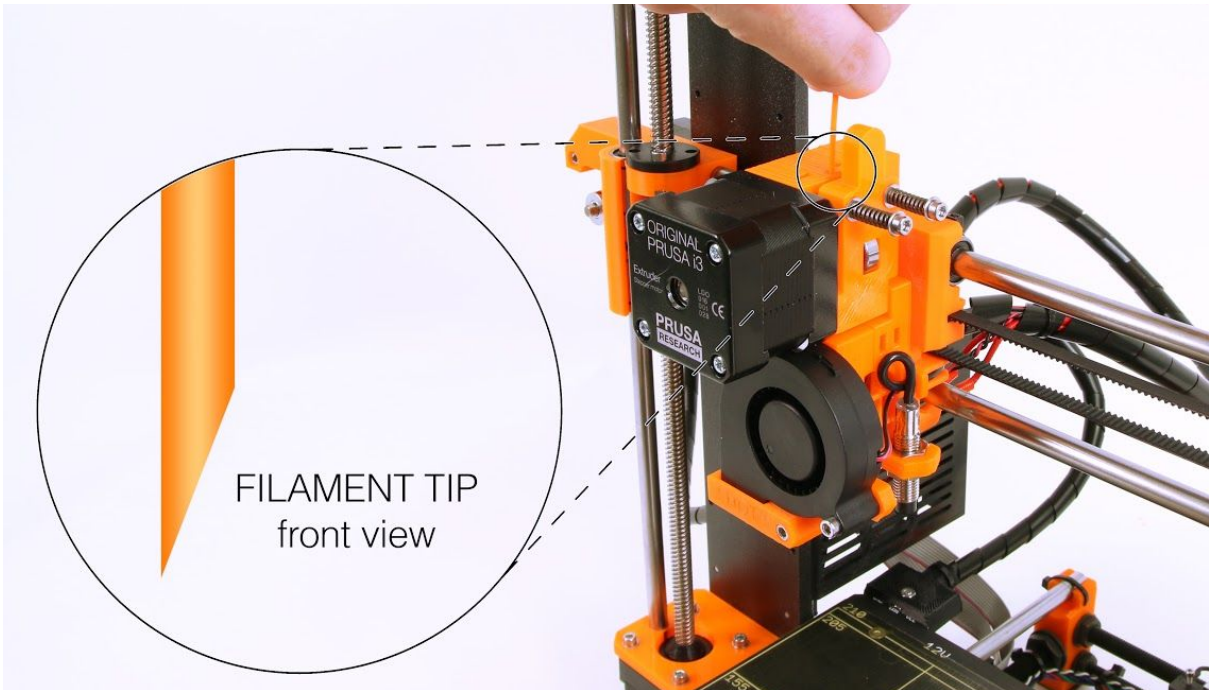
*Pict. 6 - The probe must be inside the circles (1st and 9th calibration point) to successfully level the bed.*

### 6.3.8 Loading the filament into the extruder

- 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 11 Materials.

- 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.

- 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.7 - detail).




*Pict. 7 - Loading the filament to the extruder*

- Check if the filament is flowing from the nozzle.
- If you change the filament for a different one do not forget to completely remove the old filament before printing by extruding the filament from **Settings - Move axis - Extruder** until the color is completely changed.

If your filament is running out during a print, you can easily change it for a new spool. Just go to the LCD menu, select **Tweak** submenu and press **Change filament**. Printer will pause, go out of the print area, unload the old filament and guide you on the LCD what to do. You can even insert filament of a different color and make your prints more colorful. Check out

chapter 10.6 Printing in color with ColorPrint to find out how to make more intricate color models.

#### 6.3.8.1 Unloading the filament

Similar procedure to the loading operation. **Preheat** the nozzle for material you used last time (pre assembled printers are shipped with PLA). Wait for temperatures to stabilize and use **Unload filament** option from the menu.

### 6.3.9 First layer calibration (kit only)

Now we will finally calibrate the distance between the tip of the nozzle and the probe.

**Check if your print surface is clean!** You can find instructions how to clean it in the chapter 6.3.2 PEI print surface preparation. Don't forget to complete 6.3.5 Calibrate XYZ chapter or **you can permanently damage the print surface**!

There are two ways to launch the first layer calibration.

#### 6.3.9.1 Launching first layer calibration directly from menu - preffered way

You can launch the calibration from menu **Calibration -> First layer cal.** This option is available since the **3.1.0 firmware**. If you do not see this option, you might be running an older firmware, but you can still launch it from an SD card.

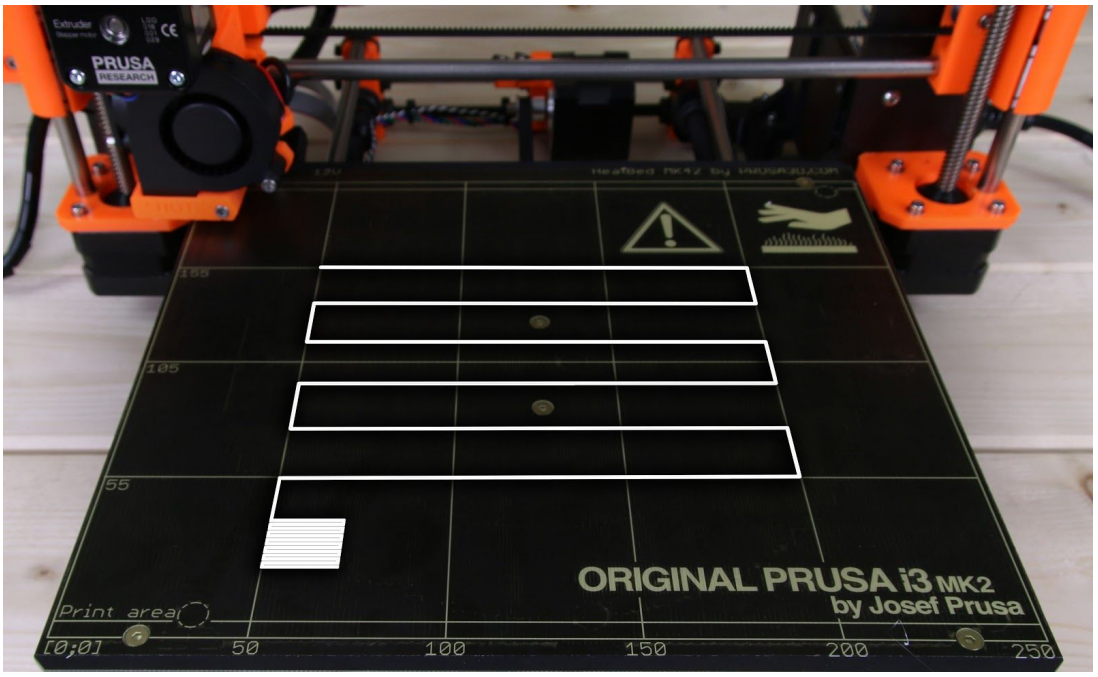
#### 6.3.9.2 Launching first layer calibration from SD card - deprecated

Preheat the nozzle for PLA. On the LCD menu, go to **Print from SD** and run **V2calibration.gcode** file from the **bundled SD card**.


*NOTE: 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.*

#### 6.3.9.3 Adjusting z-height during the calibration

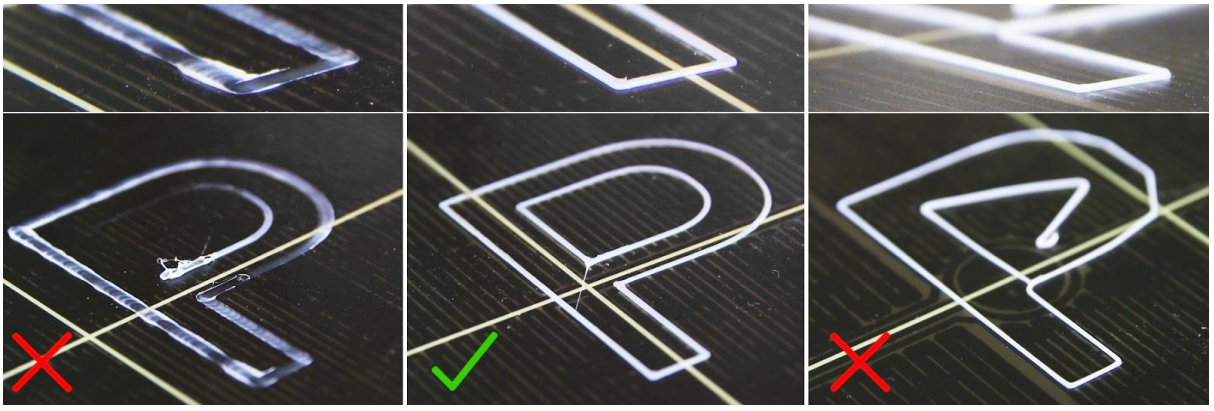
The printer will probe the bed and start printing a zig zag pattern on the print surface. The nozzle will be at the height based on the P.I.N.D.A probe setting, it must not by any means touch the print surface.





*Pict. 8 - How to tune the nozzle height live during the test print. Note: -0.640 mm is only for illustration. Your setting will be different!*

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. Set value should not exceed -1 mm, **if you have to adjust it more, move the probe slightly higher.** Loosen the two screws on the probe holder to make adjustments. **Rotating the probe counter clockwise will raise it 1mm per turn.** It is very handy for precise adjustments, but it can also be pushed in and out when set screws are loosened completely. Then rerun Calibrate Z followed by the First layer calibration again.



*Pict. 9 - The properly tuned first layer*

#### 6.3.9.4 Bed level correction (kit only)

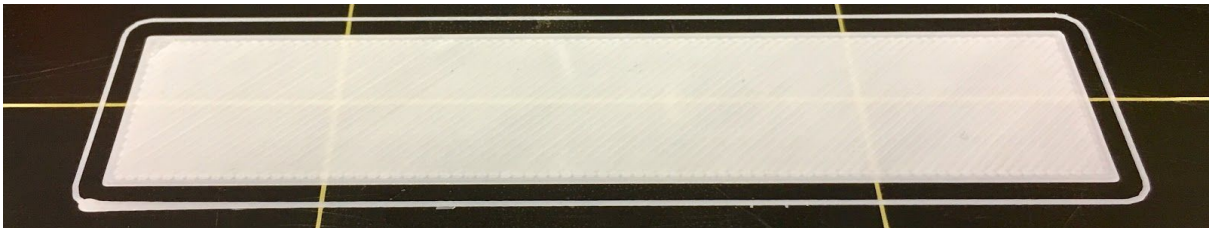
Bed level correction is an advanced feature that was introduced in the firmware 3.0.6 and it is designed to allow advanced users to correct for the slightest imperfections in the first layer. This feature can be found in **Calibration** - **Bed level correction.** For example if the first layer seems to be ever so slightly more squished on the right side, you can virtually raise the nozzle by **+20** microns on the right side. Settings are available for Left, Right, Front and Back. The limit is +-50 microns and even +-20 microns can make a huge difference. When you are using this function, do small incremental changes. A negative value will act as lowering the bed in the selected direction.

### 6.3.10 Fine-tuning the first layer

#### 6.3.10.1 Print Prusa logo

After finishing the calibration gcode, it is a good idea to print a simple object. The Prusa logo from the supplied SD card is a great example. The **Live adjust Z** function (described in 6.3.9. First layer calibration) works during every print, so you can finetune at any point. You can see the properly tuned first layer on the images below.

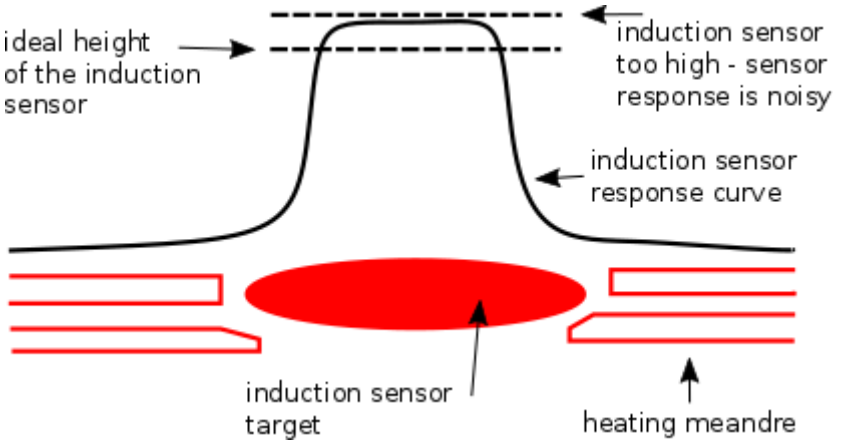
Calibration might be slightly different for multiple materials. It is a good practice to check the first layer and adjust accordingly with **Live adjust Z** when switching between different types of filament.



*Pict. 10 - Perfect Prusa logo first layer*

#### 6.3.10.2 Check probe height (kit only)

If the first layer seems inconsistent between multiple prints, the probe might be too high. Lower it slightly. Loosen the two screws on the probe holder to make adjustments. **Rotating the probe clockwise lowers it 1mm per turn.** It is very handy for precise adjustments, but it can also be pushed in and out when set screws are loosened completely. Then try again **Calibrate XYZ**. Keep in mind, the probe must be always higher than the nozzle tip, otherwise it will catch on prints.




*Pict. 11 - Probe response diagram.*

## Now you are done!

# 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 6.3.8 Loading the filament into the extruder.

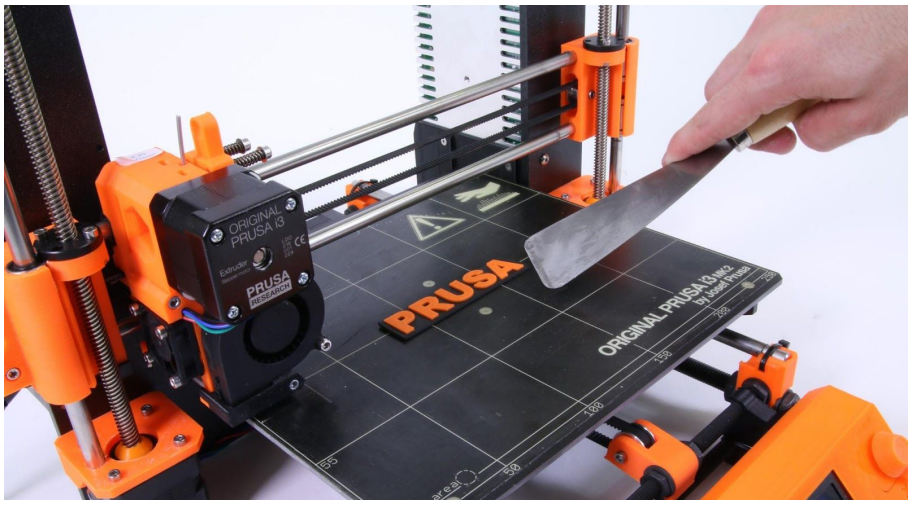
Do not let the preheated printer idle. When a printer is preheated and non-printing material in an extruder degrades over time - it may cause the nozzle to jam up.

- **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.

## 7.1 Removing objects from the printer.

- 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 **with rounded corners** to prevent damage of PEI**.** Slide the spatula under the corner of the object and gently push, until the print pops of.




*Pict. 12 - Removing the model from PEI print surface with spatula*

If your prints are stuck too well and don't want to separate, try using a **dental floss**. Slide a dental floss under the corner of the object which you previously separated by the spatula a in **saw like motion** gently separate the object from the print surface. Dental floss is generally covered in wax so the print surface needs to be **cleaned thoroughly** before a next print.

## 7.2 Printer Control

There are two ways to control 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.

### 7.2.1 LCD screen

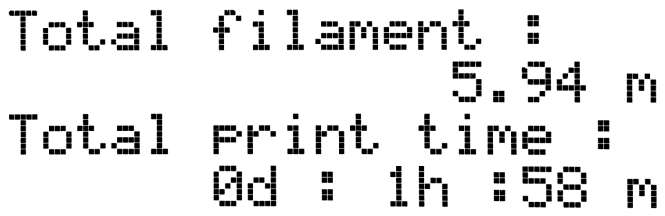
- Main screen is an **information screen** displaying the most important details. These are the temperature of the nozzle and the heatbed (1, 2), printing time (3) and the actual Z-axis position (5).


*Pict. 13 - 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 MK2 ready / Heating / model_name.gcode, etc.)
5. Z-axis position
6. Printing speed
7. Elapsed printing time - shown only when printing

### 7.2.2 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 are tracked.




*Pict. 14 - Print statistics*

### 7.2.3 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) or if you're manually adjusting speed of printing to more than 100%, use Hi-power mode.

From the firmware version 3.1.0, there is a new Auto Power Mode. It sets stepper motors power which lies between silent and high power mode. In Auto Power Mode stepper currents depend on Z height. Current starts low when Z height is minimal and increases slowly with the object being printed.

### 7.2.4 Factory reset

The factory reset is used when troubleshooting the printer and resetting it to the factory state.

**Entering the factory reset menu:**

1. **Press and release the reset button** (marked X and positioned under the control knob on the LCD panel)
2. **Press and hold the control knob** until you hear a beep
3. **Release the control knob**

**Options:**

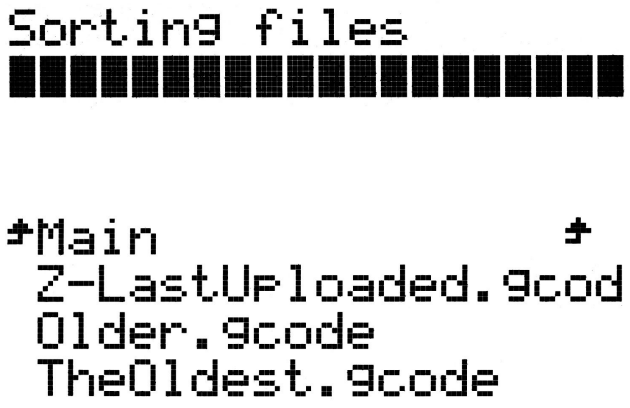
- **Language** option resets the language preference.
- **Statistics** will erase all the recorded print time and material from the memory.
- **Shipping prep** which resets only the printer language selection. All the calibration data including the Live adjust Z remain intact. Even though the calibration data are still present and functional, the printer will prompt user once to run the Calibrate Z function. This light factory reset is mainly used for resetting of assembled printers before shipping out of the factory, and users are expected to select their language and run Calibrate Z after unpacking.

- **All data** which resets everything including all calibration data and whole EEPROM is cleaned. After this reset, user is expected to go through the calibration flow again, except setting the probe height.

If you experience random glitches after firmware update or after printer upgrade, use the **All data** option.

### 7.2.5 SD card sorting

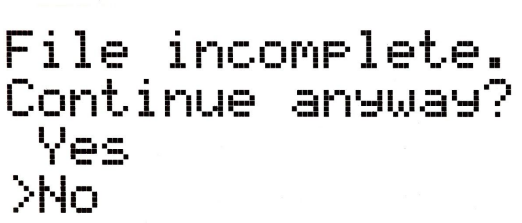
Files on the SD card can be sorted, you can change the sorting type in Settings -> Sort: [Type] you can select to sort by name, by date or no sorting. Best is by time where newest files are on the top.
Folders are shown on the top of the SD card menu and then other files follow.
Maximum number of files which can be sorted is 100. If there are more, some of them will remain unsorted.




Pict. 15 - SD card sorting

### 7.2.6 Testing if file (.gcode) is complete

The printer automatically looks for common gcodes which indicate the end of the generated file. If they are not detected, you will get a warning. You can still continue printing if you wish, but at least checking the file should be done.




Pict. 16 - Incomplete file warning

### 7.2.7 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 - 215/55
    - 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

- ❏ Disable steppers
- ❏ Mode - Hi power / Silent
- ❏ Live adjust Z
- ❏ Select language
- ❏ SD card - Normal / FlashAir
- ❏ Sort - Time / Alphabet / None

- ❏ Calibration
  - ❏ Auto home
  - ❏ Selftest
  - ❏ Calibrate XYZ
  - ❏ Calibrate Z
  - ❏ First layer cal.
  - ❏ Mesh Bed Leveling
  - ❏ Wizard
  - ❏ Bed level correction
  - ❏ Temperature Calibration
  - ❏ PID Calibration
  - ❏ Show end stops
  - ❏ Reset XYZ calibration
- ❏ Statistics
- ❏ Support
  - ❏ Firmware version
  - ❏ XYZ calibration detail

### 7.2.8 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 in 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 in the model being 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

.


*Pict. 17 - Print quality vs print time*

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.

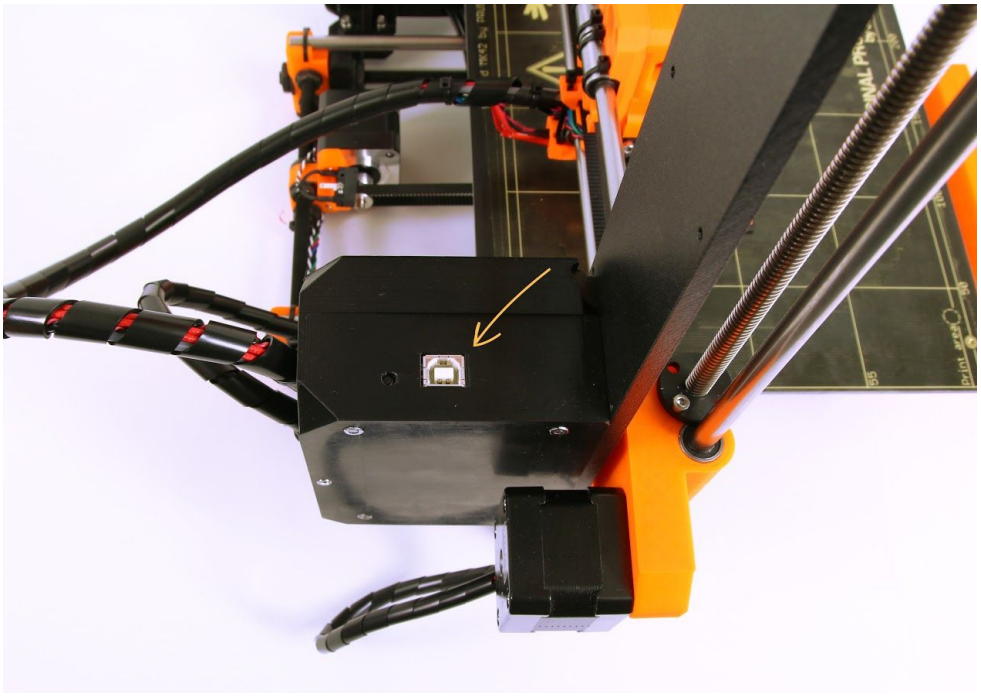
If the model shows lower quality than desired you can decrease the printing speed - turn the LCD-knob counterclockwise. Minimum usable printing speed is around 20 % of nominal speed.

### 7.2.9 USB cable and Pronterface

We strongly recommend to use LCD panel while printing on Prusa i3 MK2S - Pronterface doesn't support all functions of a new firmware (e.g. filament change while printing).

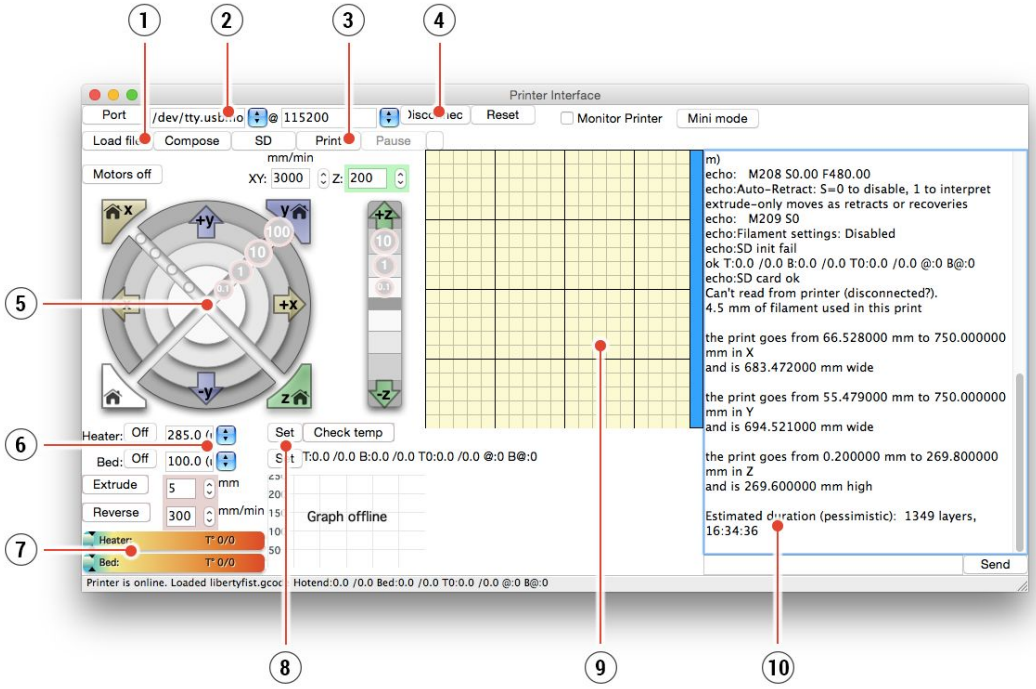
Keep in mind that when printing from the Pronterface the **computer must be connected to the printer during the whole printing process** - computer must be prevented from sleep, hibernation or shutting down. Disconnecting the computer during the print ends the printing without the option to finish the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

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



*Pict. 18 - You can find USB port here*

- Choose connection port in Pronterface (download available with the printer drivers, see the chapter 9 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. 19 - 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 starts.
9. 2D print process preview.
10. Info panel. Estimated print time, axis position and other info is displayed after loading the model.

## 7.3 Printer addons

### 7.3.1 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 have to use proper preset settings for different nozzles in Slic3r or PrusaControl.

You can check out how to change the nozzle in section **12.5 Replacing / changing the nozzle.**

#### 7.3.1.1 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 aren't possible to print as fast as with regular nozzle.

#### 7.3.1.2 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 Advanced calibration

With firmware 3.0.12 new calibration options were added. They are optional or experimental and are intended for advanced users.

## 8.1 PID tuning for Hotend (Optional)

In case you are experiencing wide swings in temperatures of your nozzle (e.g +/- 5 C°), you shall do PID tuning on your printer. *If you are experiencing major temperature fluctuations higher than that, check that your hotend thermistor is properly seated in the heater block and plugged to your miniRAMBO board first.*

You can find this feature in **Calibration - PID calibration**. In this menu you have the option to choose the temperature for which PID will be run. Set the temperature with which you print the most as it will tune it for that the best, however, general stability will improve for all temperatures (PLA/ABS/PETG). After that, nozzle will heat up to the set temperature in 5 cycles. During cycles it is mastering the amount of power needed to reach the temperature and maintain it.

Do not touch the nozzle during this process until it is fully finished as it will reach high temperatures!

Be aware that PID tuning is not a solution for all of the temperature fluctuation issues. Always make sure that your printer is located in a room with stable ambient temperatures, more about that in **Thermal Runaway and Temperature Drops** at *help.prusa3d.com*.

## 8.2 PINDA probe calibration/ Temp. calibration (Experimental/Optional)

*It is still an experimental feature available since firmware version 3.0.12. Each user has the option to turn this feature on and off, as it may not help out in all of the cases. Once you decide to use this feature, be aware that there is a shorter calibration run before each of your prints compensating for different temperatures (this adds about 2 minutes to the standard preheat process).*

New PINDA probe calibration is minimizing the issue of frequent re-doing of your Live Z adjust. That can occur once your PINDA probe readings are influenced by surrounding conditions, mostly different temperatures. This can happen if your Live Z was tuned once printing PLA and later on you print ABS or the other way round as the bed temperature is very different.

To use this new feature it is necessary to run the temperature calibration first. It can be found in **Calibration - Temp. calibration - Calibrate**. Before you do so, please make sure that your nozzle and heatbed are perfectly clean as the extruder will be moving around heatbed during this process.

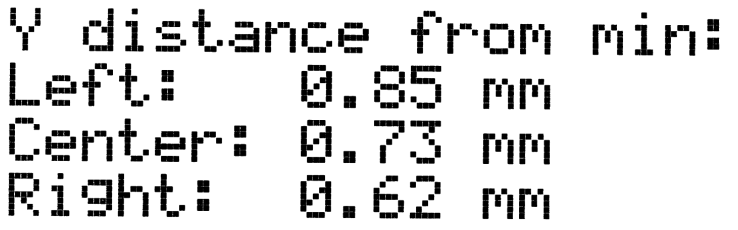
Do not touch the nozzle or heatbed during this process until it is fully finished as it will reach high temperatures!

Once calibrating your PINDA probe, it will be comparing its data readings under different temperatures and also on top of that it will include your Live Z data. This should help you to have stable Live Z.

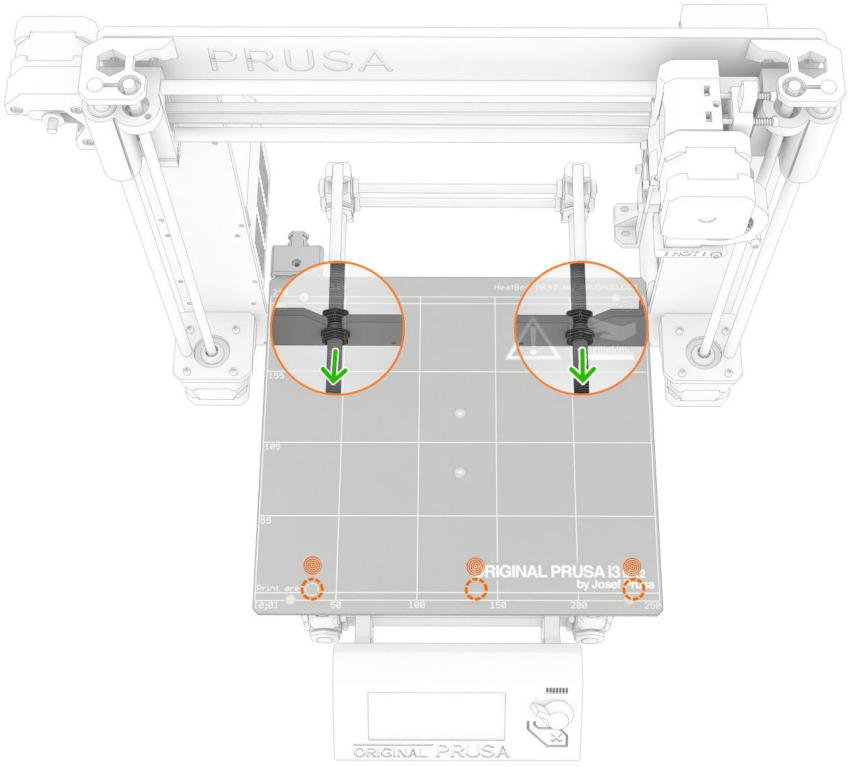
Still make sure that your 1st layer is done properly. More about that in **6.3.10. Fine-tuning the first layer**

## 8.3 View XYZ calibration details (Optional)

Since the release of firmware version 3.0.12, you have an access to more detailed info about XYZ calibrating results. This new feature can be found in **Support - XYZ cal. details**. The 1st screen tells you the distance of the “perfect” position of your front 1st, 2nd and 3rd calibration points. Ideally, all of these are positive and at least 0.5 mm or more. If the margin is between 0 and 0.4 mm, then there is a chance that your calibration will not work all of the time. In order to improve your results, you shall move with your threaded rods so that all of the points are reachable. **When you get your axes perpendicular or slightly skewed, nothing needs to be tweaked as printer will perform with the best accuracy**.




*Pict. 20 - Distance of the front calibration point from the axis start. Ideal value is between 2 and 3 mm but everything larger than 0.5 mm is good enough.*

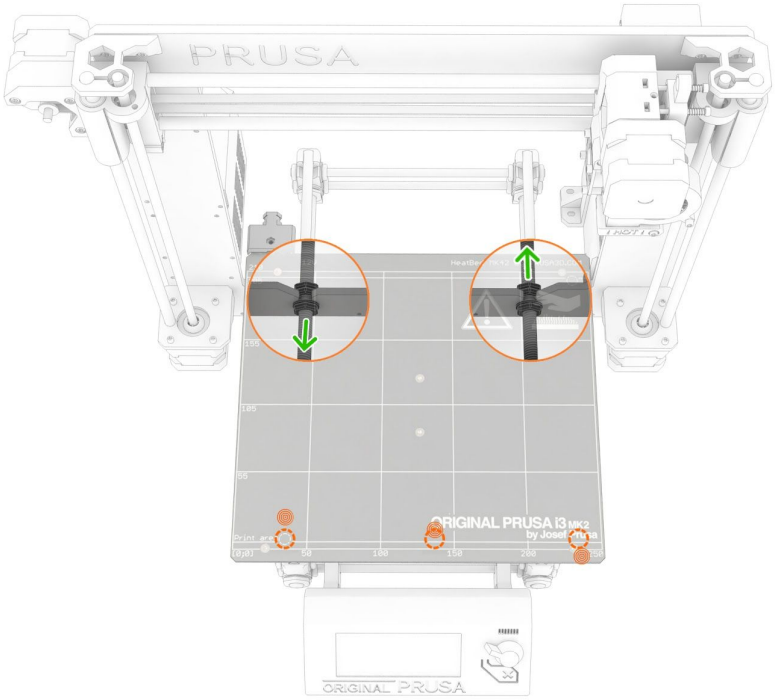



*Pict.21 - Example in the picture - moving the frame forward on the Y axis (increasing the 100 mm distance from the build manual) will increase the Y distance from min.*

Pressing the button will get you to the 2nd screen. This screen will identify how far you are from the perfect perpendicularity. It is measuring the skew of your X/Y axis.

*Up to 0.25° = **Severe skew compensating** for offset of 1.1 mm on 250 mm length*
*Up to 0.12° = **Slight skew compensating** for offset of 0.5 mm on 250 mm length*
*Under 0.12° = **No need to compensate**, X/Y axes are perpendicular. Congratulations!*

It may look that the compensations are not that high at the first sight. However, if we take into consideration the 250 mm X-axis length, 1.1 mm is a large margin. In order to improve your axis perpendicularity, make sure the the distance of the front calibration points (seen on the first screen) is the same.




*Pict. 22: moving one side of the frame forward (increasing the measured value) on the Y axis and other side backward (decreasing the measured value) according to the values from the first screen will improve the skew.*

## 8.4 Linear Advance (Experimental)

Linear Advance is the new feature from the firmware version 3.1.0. You don't need to enable it or tweaking - **everything is preset**. You just need firmware 3.1.0 and **drivers updated to 1.9.2** or higher.

Thanks to Linear Advance, all the print settings for MK2/S has slight 10-15 mm/s speed increase.

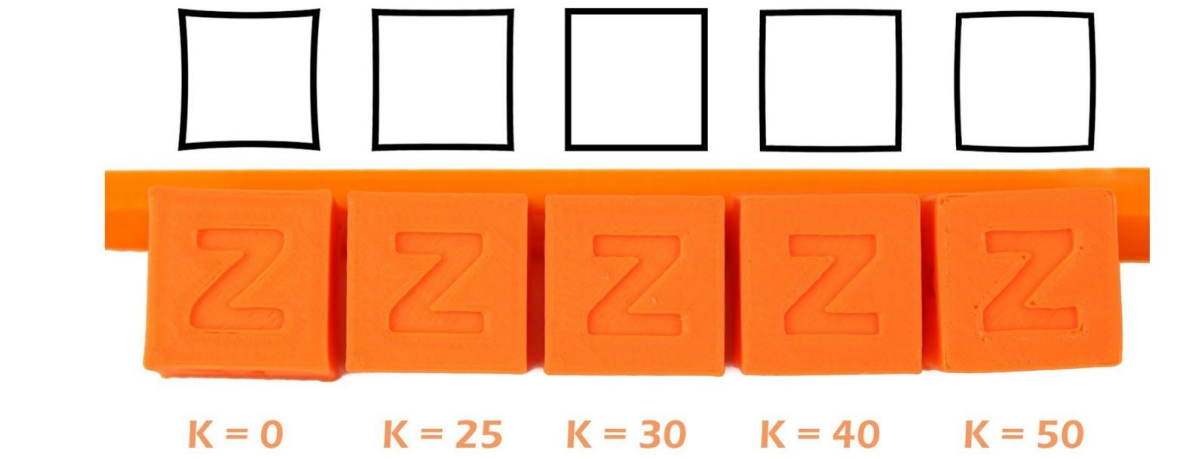
New print settings were also added to Slic3r PE for experimental faster printing – 0.15 mm 100 mm/s Linear Advance and 0.20 mm 100 mm/s Linear Advance where speed is increased even more, up to 100 mm/s, to get additional ~30% faster printing. You do not need to calibrate or tweak anything to get faster printing, just use these settings to generate your gcode.

To see the speed increase, you must **test on larger models**! Slic3r limits the minimum time per layer to 15 seconds and in case it is triggered, print speed is lowered. This prevents insufficient cooling and ugly prints. Printing one Marvin will not be any faster than before.

If you are using different slicers than Slic3r PE or PrusaControl or you just want to tweak and play around with different values, you can manually change the settings in gcode script. **However, if you do not understand the concept of gcodes yet or never played with editing it, stop reading here and skip for another chapter.**

The K values (the parameter affecting how much Linear Advance affects the print) we measured and tested are as follows:

- PLA: **M900 K30**
- ABS: **M900 K30**
- PET: **M900 K45**
- Multi material printer: **M900 K200** for all materials




*Pict. 23 - How K value affects the print*

These values are preset in our Slic3r PE. The K value is set in custom gcode section in the **Filament Settings tab**, NOT under the printer specific custom gcode. **PrusaControl** uses the same K values but will not allow users editing.

**Simplify3D, Cura**, … users just need to add “M900 K??” into the starting gcode script. Keep in mind you need to manually change that for different filament materials. Only Slic3r PE has custom gcode for each filament preset and therefore K value is changed automatically.

Set the speed you want, print something (large enough for speed to show up). If sharp corners have blobs, **increase K value**. If you see missing filament, **decrease the K value**.

Please note that different brands and colors of the same material may require slightly different K value when printing at extreme speeds, however our presets should be fine with all of them.

**DO NOT turn off filament retract completely.** Tune the K value first, then try slowly decreasing the retract distance until you see stringing, then increase a touch.

# 9 Printer drivers

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

Driver package contains following settings and programs:
**PrusaControl** - preparing the 3D models to .gcode format for printing.
**Slic3r Prusa Edition** - preparing the 3D models to .gcode format for printing.
**Pronterface** - legacy 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 and Mac drivers
**Test objects**

# 10 Printing your own models

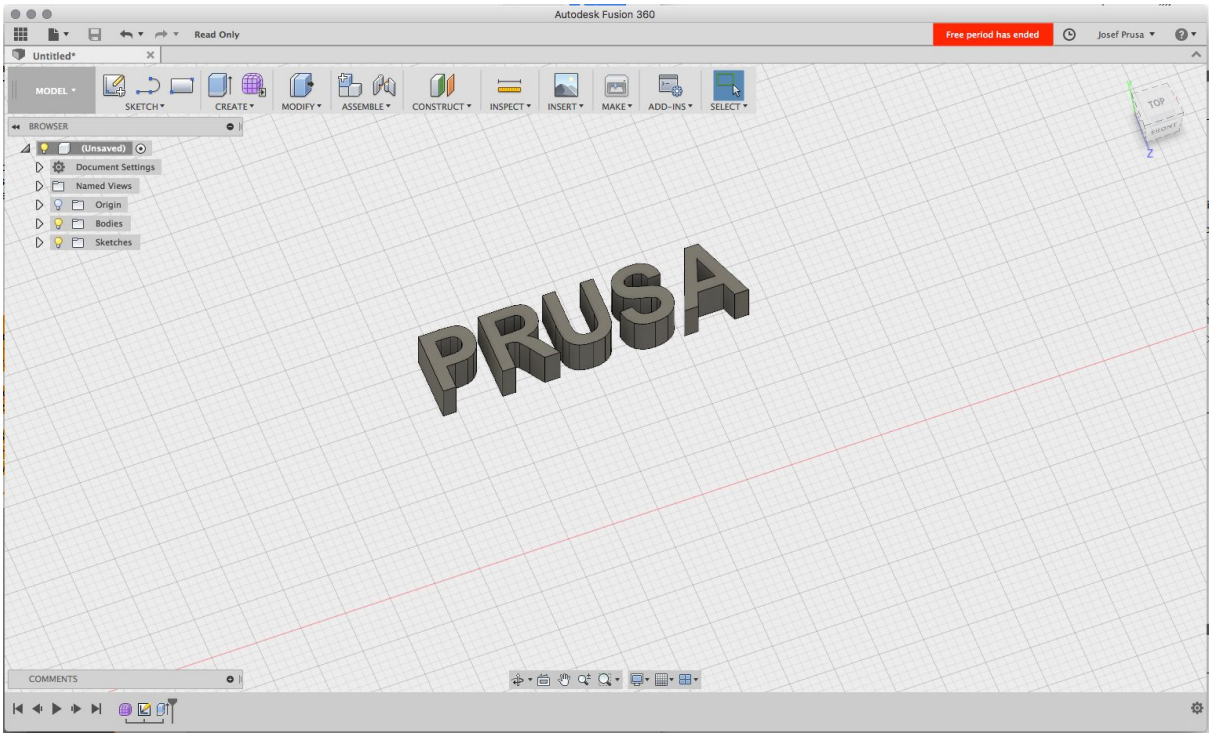
## 10.1 Where you can get the 3D models?

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/

## 10.2 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 Fusion 360 (https://www.autodesk.com/products/fusion-360/) 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…

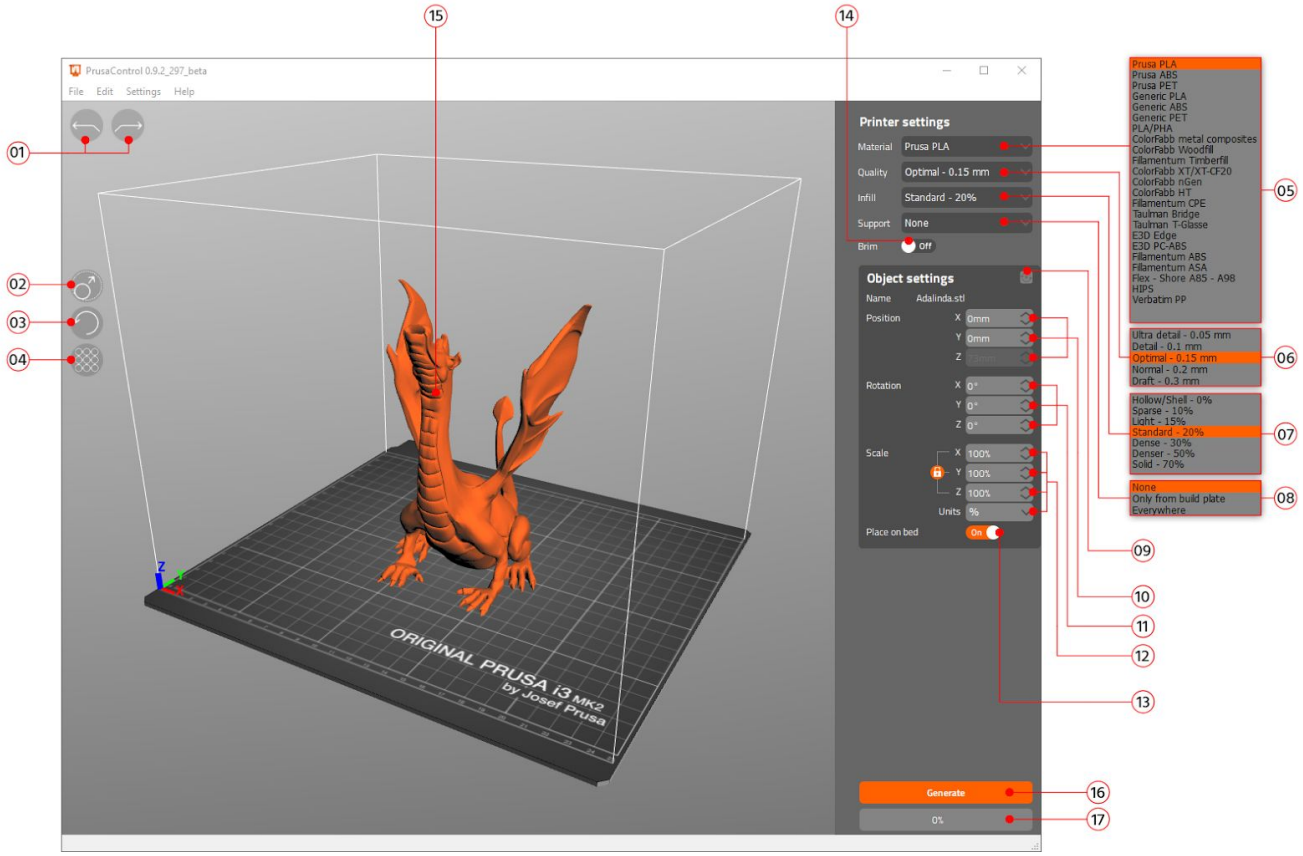



*Pict. 24 - Fusion 360*

## 10.3 PrusaControl

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

**PrusaControl** is the easiest way to get perfect prints on the MK2/MK2S and should be used when first experiencing the 3D printing world. When you get more advanced and want to tweak the print settings or add new materials, **Slic3r Prusa Edition** is waiting for you.

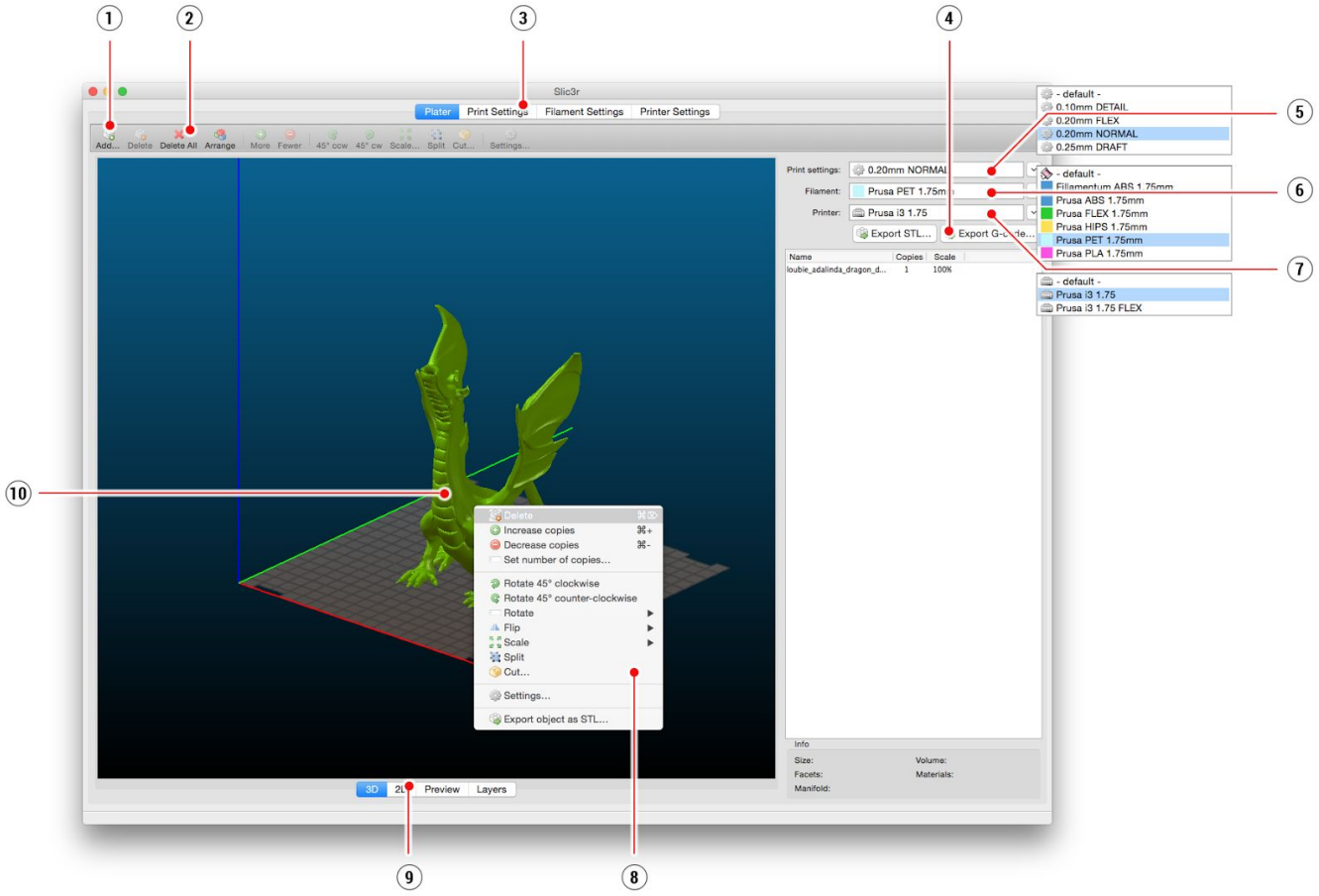



*Pict. 25 - Prusa Control interface*

1. **Undo/Redo** buttons return changes.
2. **Scale** button allows you to scale with the mouse while the model is selected.
3. **Rotate** button allows you to rotate with the mouse while the model is selected (outer circle step are 0,1°, inner circle step is 45°.)
4. **Auto arrange** button positions objects on the print bed.
5. Material selection menu
6. Quality / Speed setting of a print menu
7. Infill menu
8. Supports menu
9. **Reset transformation** settings button
10. **Position** values
11. **Rotation** values
12. **Scale** values
13. **Place on bed** button turns on automatic placing of objects to Z=[0]
14. **Brim** On/Off button
15. Model preview
16. **Generate** button slices the model
17. **Progress** bar

## 10.4 Slic3r Prusa Edition

PrusaControl is build on top of the **Slic3r Prusa Edition** and hides all the unnecessary clutter from having all settings exposed. If you choose to create your own specific print settings or tweak material settings heavily, you can use Slic3r PE directly.




*Pict. 26 - 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

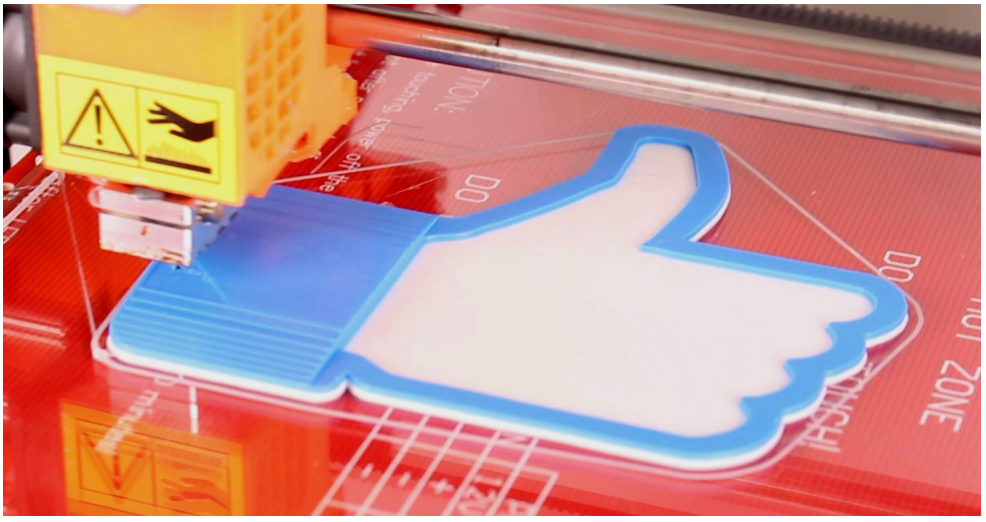
## 10.5 Bundled 3D models

We asked a couple of 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 driver's 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. 27 - 50 microns treefrog is commonly used as a 3D printing benchmark.*

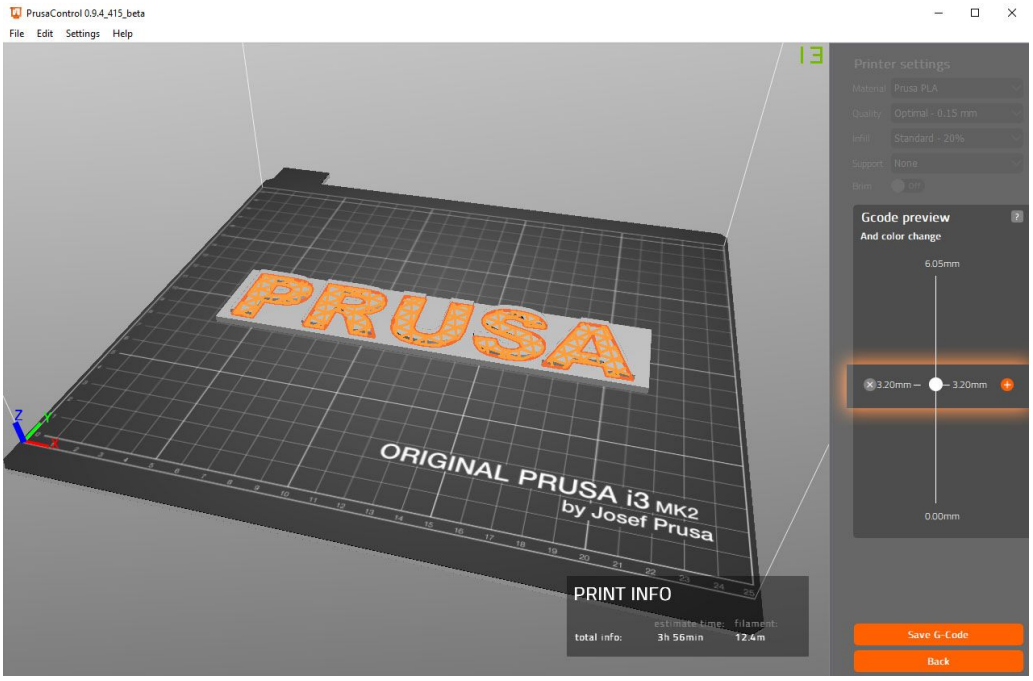
## 10.6 Print in color with ColorPrint

There is a simple way on how to create layer based multicolored 3D prints with PrusaControl or with our simple online ColorPrint app by manually changing the filament.



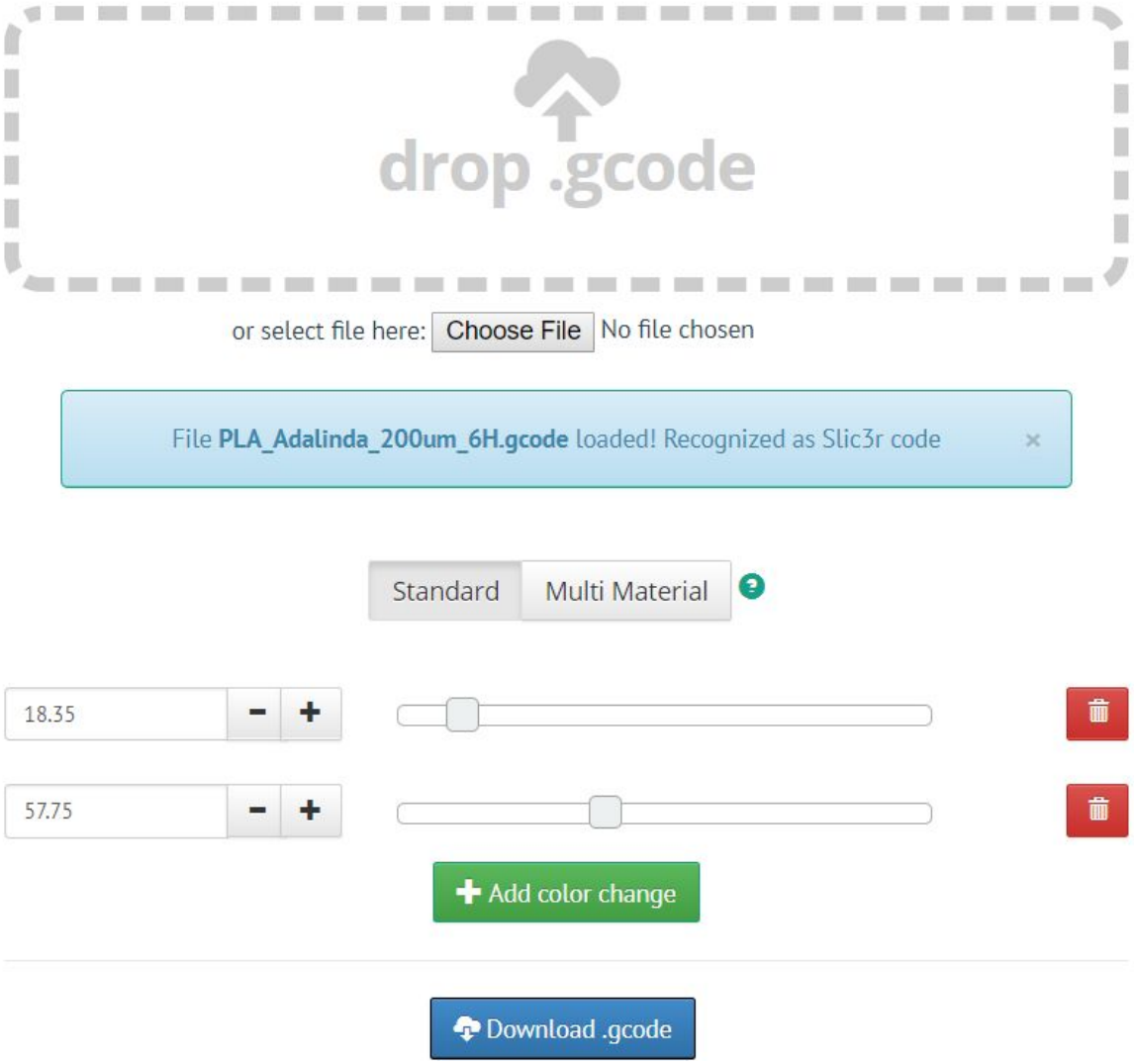
*Pict. 28 - Multicolored object printed with ColorPrint*

**ColorPrint is now directly integrated into the PrusaControl and filament changes can be added when the gcode is already generated before saving it to the file. PrusaControl can also add color changes to existing gcodes (generated in Slic3r for example). You can also use Web ColorPrint for gcode from other slicers including Slic3r Prusa Edition.**




*Pict. 29 - Adding color change in PrusaControl*

- 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!




*Pict. 30 - Web version of Colorprint interface at prusaprinters.org/colorprint*

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.

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.

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

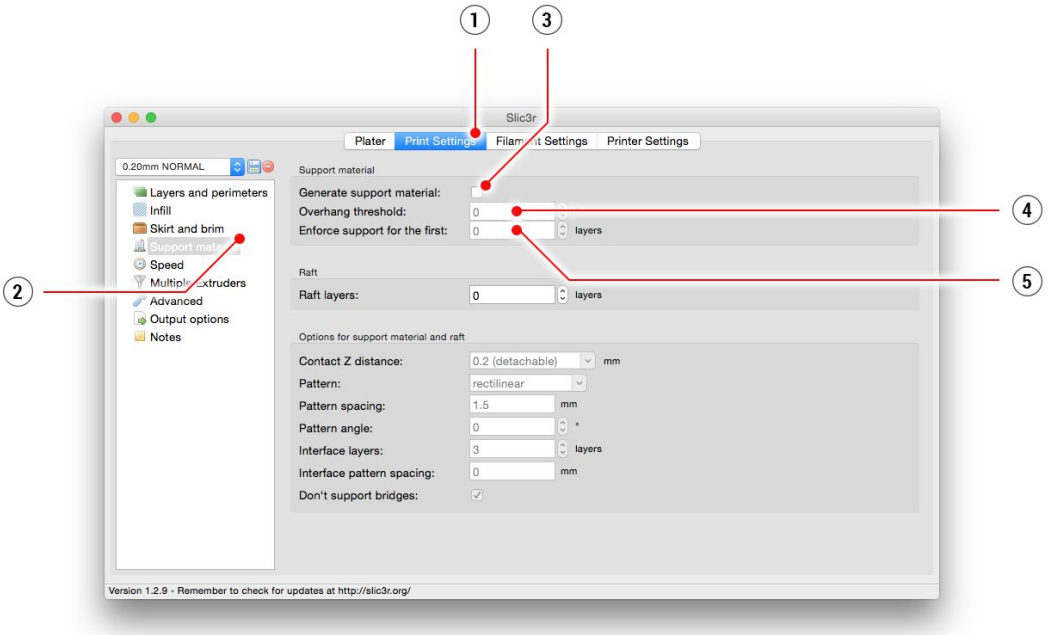
## 10.7 Printing of non-standard models

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

### 10.7.1 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.

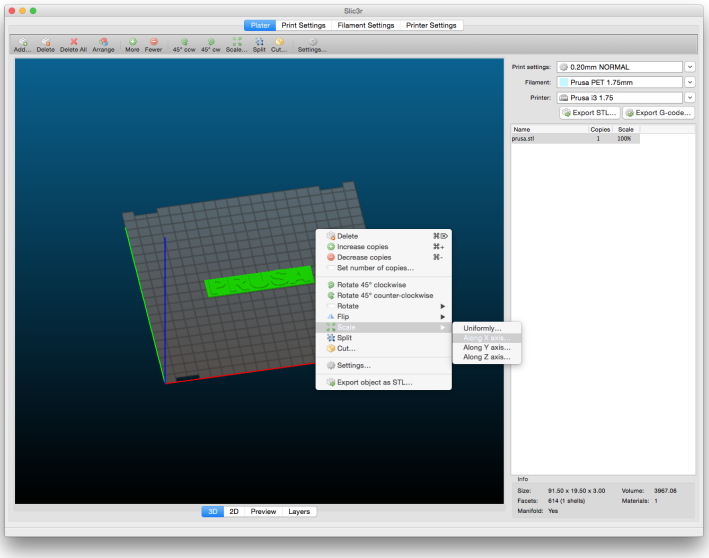
*Enforce support* option **(5)** is used mostly with small models or models with small base to prevent the object from breaking or tearing out from the bed.




*Pict. 31 - Print with support menu*

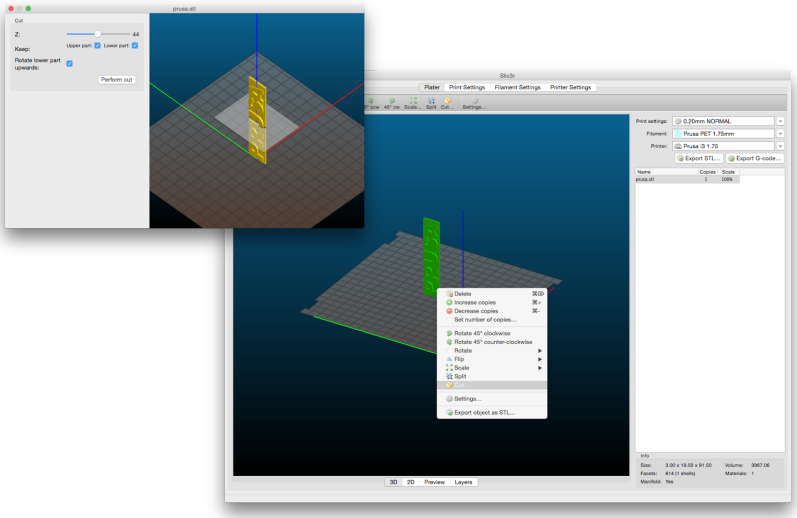
### 10.7.2 Large object printing

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. 32 - 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. 33 - Cutting the object with the Cut option*

# 11 Materials

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

## 11.1 ABS

ABS is very strong and versatile material with **great thermal resistance**. It's suitable for both indoor and outdoor use.
ABS is a thermoplastic polymer, that means that just like PLA, it can be melted and crystallized multiple times without degrading too much. ABS, however, melts at a higher temperature than PLA. Higher melting temperature gives ABS great thermal resistance, your prints won't show signs of deformation **up to 98 °C**.
ABS includes high wear resistance synthetic rubber, which makes it **very strong and impact resistant**. And last but not least, it's **soluble in acetone**! This makes it really easy to not only connect multiple parts together but also allows you to **smooth prints** with acetone vapors. You still have to be careful when handling acetone, but it's not anywhere near as dangerous as for example PLA solvents.

The best use of ABS is architectural models, concept models, spare parts (car interior, gears, phone cases), etc.

On the other hand, thermal contraction is where ABS makes it really hard to successfully print something. And that's especially true when printing anything big. Even with the heatbed at 100 °C, your part may start lifting from the build plate and warp. This and the **unpleasant smell** of ABS is why you should consider getting an enclosure for your printer when printing with ABS. Or at least place the printer in a warm room.
If you need to use your print outside or just need your print stronger, give ABS a shot. It's what **LEGO** is made of after all.

| ADVANTAGES | DISADVANTAGES |
|---|---|
| High impact and heat resistance | Bad smell |
| Strong and versatile | Worse resolution |
| Soluble in acetone (easy post-processing) | Needs warm room or enclosure |
| Can be vapor smoothed | |

- **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 6.3.2 PEI print surface preparation chapter

## 11.2 PLA

PLA is the most commonly used filament. It's **biodegradable**, **easy to print**, and very **strong** material. The perfect choice for printing **large** objects thanks to its low thermal expansion (little to no warping) and for printing **tiny** parts because of its low melting temperature. **This material only is proven for 50 microns layer height.**
PLA has a relatively low melting temperature of about 175 degrees Celsius. Unlike so-called thermoset materials, PLA can be heated past its melting point multiple times with very little degradation. It's also very hard material, but that also means it's somewhat brittle and once it breaks, it likes to shatter.

The best use of PLA is printing concept models, prototypes, low-wear toys, etc.

However, PLA is not a perfect material and just like every other plastic has some disadvantages. The low melting temperature also means **low-temperature resistance**, parts start to lose mechanical strength at temperatures over **60 °C**.
The combination of being both biodegradable and having low-temperature resistance means that it's **not ideal for outdoor use**, not to mention low UV-resistance. Also, PLA is only soluble in chemicals like chloroform or hot benzene. So when connecting multiple pieces, you're better off just using glue.
Even though PLA is biodegradable and the material on its own is food safe, we do not suggest to repeatedly **drink or eat from your 3D prints**. Because of the small fractures on the print surface, bacteria can build up in there over time. You can prevent this by applying food safe coating. When **post-processing PLA**, it's better to use wet sanding. Without water you'll quickly start heating the plastic by friction, it will melt locally and make it hard to keep sanding.

| ADVANTAGES | DISADVANTAGES |
|---|---|
| Easy to print | Brittle |
| Can print tiny parts | Low-temperature resistance |
| Can print huge objects | Difficult post-processing |
| Hard and stiff | |
| Low warping | |
| Environmentally friendly | |

- **Nozzle temperature:** 215 °C
- **Bed temperature:** 50 - 60 °C
- **Heatbed:** Make sure the surface is clean, as described in 6.3.2 PEI print surface preparation chapter

## 11.3 PET/PETG

PETG is a very tough material with good thermal resistance. It's universal but suitable especially for mechanical parts and both indoor and outdoor use. PETG has almost **no warping**, so printing large objects isn't a problem. We use PETG to print parts for our printers!
PETG is one of our favorite materials for 3D printing. It's almost as easy to print as PLA, but it can offer many mechanical properties that PLA prints just cannot achieve.
The G in the acronym PETG stands for Glycol which is added during the manufacturing process. Glycol modifies the properties of PET, so that it's **easier to print, less brittle and clearer** when printing with semi-transparent variants. PETG has low thermal expansion, so even when printing big and without an enclosure, it rarely lifts from the bed and warps. In addition to that, PETG is **ductile**, it has a healthy amount of flex which can prevent parts from breaking under stress.
Unlike PLA or ABS, PETG tends to ooze a bit and may leave **strings of plastic** on your print. You can fight this with increasing retraction and playing with hotend temperature, but if you use our filament presets in **Slic3r or Prusa Control**, 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 sticks very well to PEI, which is generally a good thing. But sometimes t could stick a little bit too well and you could rip a piece of PEI from the bed, so you should use a **separating agent** (e.g. gluestick).
If you can handle the oozing and strong adhesion, you'll be left with a very durable print, that is considerably temperature resistant and usable for both indoor and outdoor use.

| ADVANTAGES | DISADVANTAGES |
|---|---|
| Easy to print | Possibility of stringing |
| Good layer adhesion | Not soluble in acetone |
| Very tough, low warping | Prone to scratches |
| Temperature resistance | |
| Little shrinking | |
| Durable | |

- **Nozzle temperature:** 240 °C
- **Bed temperature:** 80 - 100 °C
- **Heatbed:** Make sure the surface is clean, as described in 6.3.2 PEI print surface preparation chapter. Do not use isopropyl alcohol to clean the bed, or the adhesion may be too strong, if you do not have anything else on hand use the bundled glue as a separator after cleaning it. Windex or similar windows cleaner is great option for PET and you don't need to use the glue after the cleaning. Pour a little amount on unscented paper towel and wipe the print surface.

## 11.4 HIPS

HIPS is high impact polystyrene, and as for behavior, it's similar to ABS, so it's easy to print. It's a universal and stable material with excellent heat resistance, and it produces very smooth layers. HIPS is also very malleable, and it can be dissolved using limonene. HIPS is mostly suited for printable mechanical components.

| ADVANTAGES | DISADVANTAGES |
|---|---|
| Smooth | High level of warping |
| Durable | Bad smell |
| Soluble | |

- **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 6.3.2 PEI print surface preparation chapter

## 11.5 PP

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

| ADVANTAGES | DISADVANTAGES |
|---|---|
| Tough | High level of warping |
| Semi-flexible | |
| Temperature resistance | |

- **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.

## 11.6 Nylon (Taulman Bridge)

Nylon is very tough material suitable for mechanical parts.

| ADVANTAGES | DISADVANTAGES |
|---|---|
| Durable | Demanding storage (it's hygroscopic) |
| Chemically resistant | |
| Flexible, but strong | |
| Chemical resistance | |

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

## 11.7 Flex

Flex is a very strong and flexible material. There are many use cases where hard plastic is not the ideal or even unusable at all. But whether you need a phone cover, an action camera case or wheels for your RC car, flexible is the way to go.

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 change function may not work properly.

Flexfill has very good abrasion resistance, remains flexible in cold environments, and is resistant to many solvents. It doesn't shrink much when cooling down, so you can be fairly accurate with your measurements and perfect fit models.

| ADVANTAGES | DISADVANTAGES |
|---|---|
| Flexible and elastic | Needs extra steps when loading filament |
| Little shrinking | Can be tricky to print |
| Good layer adhesion | Needs to be printed slowly |

- **Nozzle temperature:** 230 °C
- **Bed temperature:** 50 °C. You can set the bed temperature up to 65 °C depending on the size of an object. (larger object means higher temperature)
- **Heatbed:** Make sure the surface is clean as described in 6.3.2 PEI print surface preparation chapter. Some very soft flex materials can bond to the bed too much and require to use glue on the bed as separator to prevent PEI damage.

## 11.8 Composite materials

Composite materials (woodfill, copperfill, bronzefill, glow-in-the-dark, carbon or aramid composites and many others) consist of main plastic base and second material in the form of dust. These materials tend (with the exception of wood composites) to be very abrasive, therefore hardened nozzle is strongly suggested for long-term printing. Larger nozzle is recommended while printing with wood composites (0.5 mm and up). Please use corresponding print settings in Slic3r or PrusaControl as print parameters can be very different depending on plastic base.
The first step in polishing is sanding. It's a good idea to start with a coarse grit size (80) and slowly move up the grit table. After sanding a big improvement in polish can be achieved with steel wool or brass brush. If you're still not happy with the finish, you can try wet sanding with a very fine grit (1500).

| ADVANTAGES | DISADVANTAGES |
|---|---|
| Easy to print | Needs hardened nozzle |
| No warping | |
| Great look after post-processing | |

- **Nozzle temperature:** 190 - 210 °C
- **Bed temperature:** 50 - 70 °C (bigger object -> higher temp.)
- **Heatbed:** Make sure the surface is clean as described in 6.3.2 PEI print surface preparation chapter.

## 11.9 ASA

Acrylonitrile-styrene-acryl (ASA) is a material with properties similar to ABS, its main benefit is increased weather and UV resistance. Other advantage is overall dimensional stability. To achieve cast-like surface, acetone smoothing can be used…

| ADVANTAGES | DISADVANTAGES |
|---|---|
| Heat and UV resistant | Bad smell |
| Soluble in acetone (easy post-processing) | High level of warping |
| Can be vapor smoothed | |

- **Nozzle temperature:** 270 - 280 °C

- **Bed temperature:** 100 - 110 °C (bigger object -> higher temp.)
- **Heatbed:** Make sure the surface of the heatbed is clean. Usage of brim is suggested (see Prusa Knowledgebase).

## 11.10 nGen

Developed by Eastman Chemical Company and colorFabb, nGen offers increased resistance to heat as well as dimensional stability. Material is low-odor and styrene-free.

| ADVANTAGES | DISADVANTAGES |
|---|---|
| High gloss | Brittle |
| Good surface finish | A bit of warping |
| Good layer adhesion | |

- **Nozzle temperature:** 240 °C
- **Bed temperature:** 80 - 100 °C (bigger object -> higher temp.)
- **Heatbed:** Make sure the surface is clean. Do not use isopropyl alcohol to clean the bed, or the adhesion may be too strong, use window cleaner instead. If you do not have anything else on hand use the bundled glue as a separator after cleaning it. Windex or similar windows cleaner is a great option for nGen and you don't need to use the glue after the cleaning. Spray small amount on unscented paper towel and wipe the print surface.

## 11.11 PC-ABS (E3D)

Polycarbonate ABS (PC-ABS) is an enhanced version of traditional ABS. Offers easier processing, higher strength, stiffness and temperature resistance. PC-ABS is also suitable for structures with openings, its bridging capability is improved compared to ABS. Typical usage of PC-ABS is for durable plastic parts like television or computer casings.

| ADVANTAGES | DISADVANTAGES |
|---|---|
| Lightweight | Warps |
| Good for mechanical parts | Low elasticity |

- **Nozzle temperature:** 270 - 280 °C
- **Bed temperature:** 100 - 110 °C (bigger object -> higher temp.)
- **Heatbed:** Make sure the surface of the heatbed is clean.

## 11.12 Dialing in new materials

Each manufacturer produces slightly different material even though, they are under the same group. For example Prusa PLA and ColorFabb PLA will have slightly different output when printed.

To achieve the best possible output you should experiment with the **nozzle temperature**, **fan speed**, **print speed** and **flow**. All of these can be changed even during a print from the **Tweak** menu on the LCD panel.

**Same also applies even for materials which are not listed here.** Take the manufacturer suggested settings, find the closest match in Slic3r material profiles, modify and save as new. **Continue by printing few simple test pieces and continuously use the Tweak menu.** After each improvement, don't forget to modify the settings in Slic3r. Reset the tweak values before every print.

**Don't forget to share your settings on our forums or directly with us.**

# 12 FAQ - Printer maintenance and print issues

## 12.1 Regular maintenance

### 12.1.1 Bearings

Every couple hundred hours, the smooth rods should be cleaned with a paper towel. Then apply a little bit of general purpose machine oil on the smooth rods and move the axis back and forth a couple of times. This cleans the dirt and increases longevity.

If you feel the axis is not running smoothly anymore, bearings can be taken out and greased on the inside (they need to be removed from axis as plastic lip will prevent the grease from getting inside). Super-lube or any other multi purpose grease will do.

### 12.1.2 Fans

Both fans should be checked and cleaned every couple hundred hours, dust or plastic build up can decrease their efficiency or even damage them. Computer cleaner spray will get the dust away and tweezers can be used for little plastic strands.

### 12.1.3 Extruder drive gear

The hobbed drive gear on the extruder motor shaft can have build up of filament shavings in the grooves and cause under extrusion. A small brass brush is ideal tool to clean the grooves but regular toothpick will do the job as well. Check and clean from the access window on the left of the extruder assembly. Clean what you can, then rotate the gear and repeat. Nothing needs to be disassembled. Clean when you see signs of missing plastic in the objects, e.g. missing lines of extrusion.

### 12.1.4 Electronics

It is good practice to check and eventually reseat the electric connectors on the mini RAMBo board. Do so after first 50 hours of printing and then every couple hundred hours.

### 12.1.5 PEI rejuvenation

PEI can lose its adhesive powers after couple hundred hours. Wipe thoroughly with acetone when you see models getting loose to restore the adhesion.

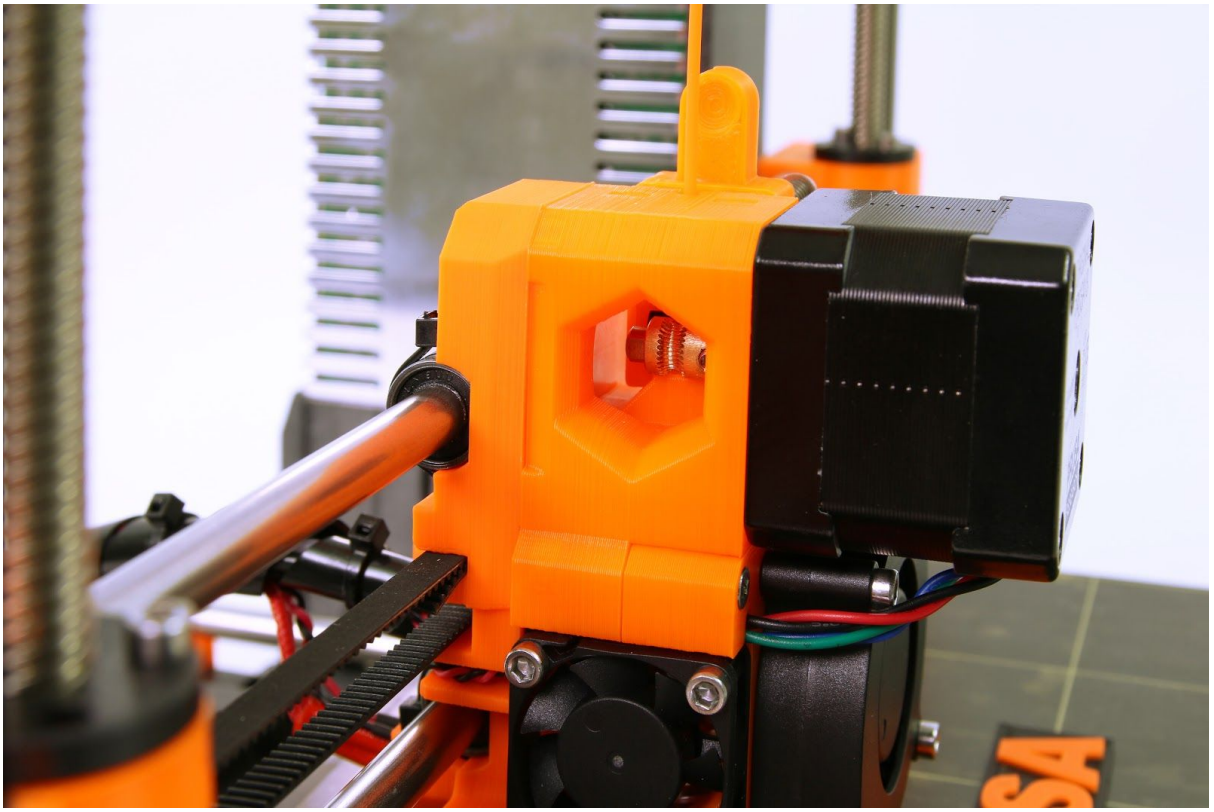
## 12.2 Print surface preparation

Print surface preparation is described in 6.3.2 PEI print surface preparation chapter.

## 12.3 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 about 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. 34).
- Clean the hobbed pulley, then heat the nozzle before reloading the filament.
- If a problem persists you will have to clean the nozzle.



*Pict. 34 - Cleaning the extruder - you can see the hobbed pulley through the service hole*

## 12.4 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 **bundled acupuncture 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 **acupuncture needle** into the nozzle again and repeat these steps a few times more. When the filament is extruded properly, the nozzle is clear.

## 12.5 Replacing / changing the nozzle

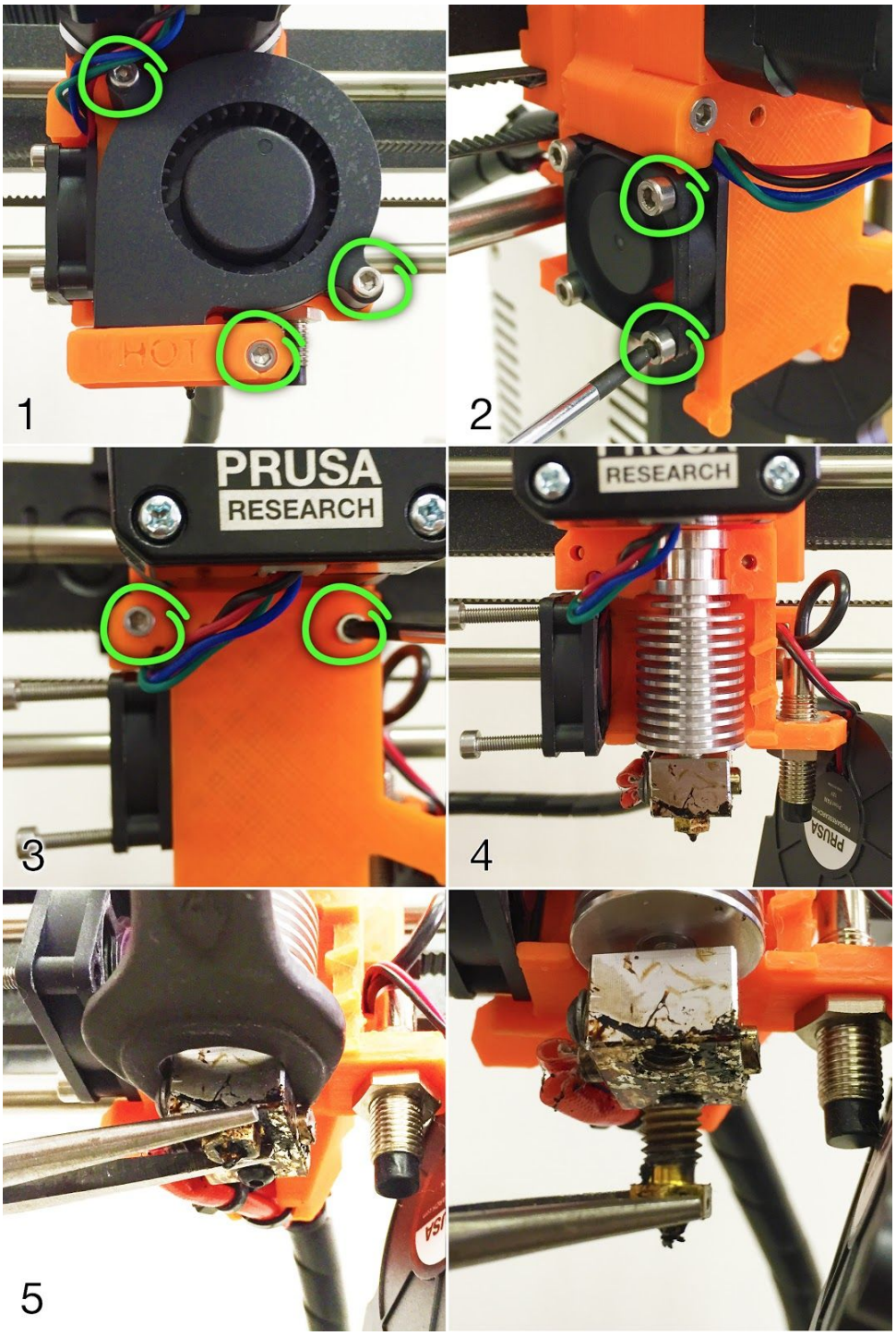
Preheat the nozzle (LCD menu -> Settings -> Temperature -> Nozzle) and set the temperature to at least 200°C. Heating the nozzle is crucial for removing the old nozzle and putting in the new one.

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. 35, part 1)**.
3) Remove the two front screws on the **nozzle fan (Pict. 35, part 2).**
4) Unscrew the two screws holding the **extruder cover (Pict. 35, 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. 35, part 4)**.
6) Hold the heating element with a spanner (size 17) and unscrew the nozzle **(Pict. 35, 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.**

It is a good practice to run **6.3.9 First layer calibration** after changing the nozzle!




*Pict. 35 - Nozzle change*

## 12.6 Printing problems

### 12.6.1 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.

### 12.6.2 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.

## 12.7 Problems with finished models

### 12.7.1 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.

## 12.8 Updating printer firmware

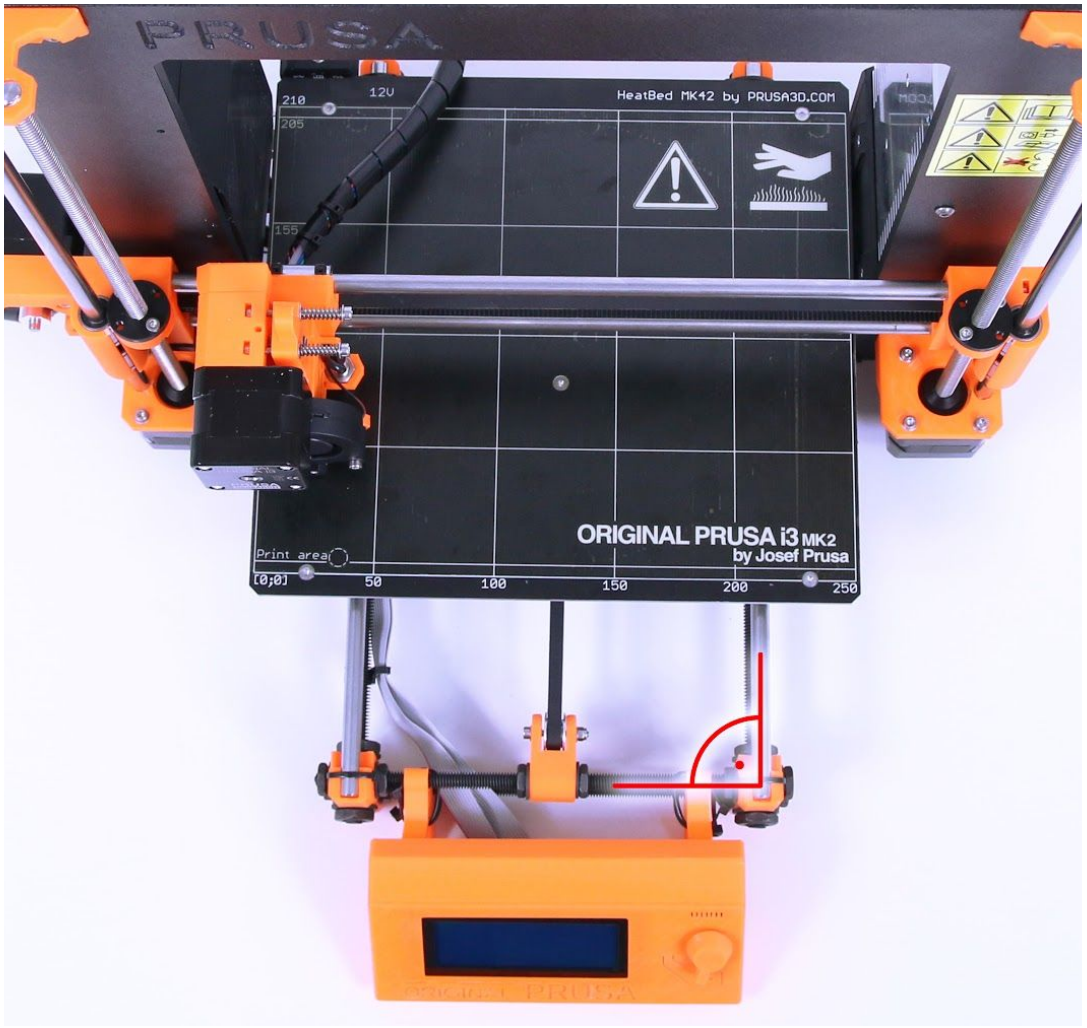
Firmware update is a simple process which is done via the USB cable and a computer. With the driver installation a program called **FirmwareUpdater V2** is installed on the computer. Latest firmware can be found on http://www.prusa3d.com/drivers/ where you can find the latest guide on which firmware to choose and **detailed instructions** for the process. Printer will reset automatically right before the update and after the update is finished. **First Layer Calibration** is required to finish after the firmware update, see chapter 6.3.9 First layer calibration.

# 13 FAQ - common issues when assembling the printer kit

## 13.1 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:

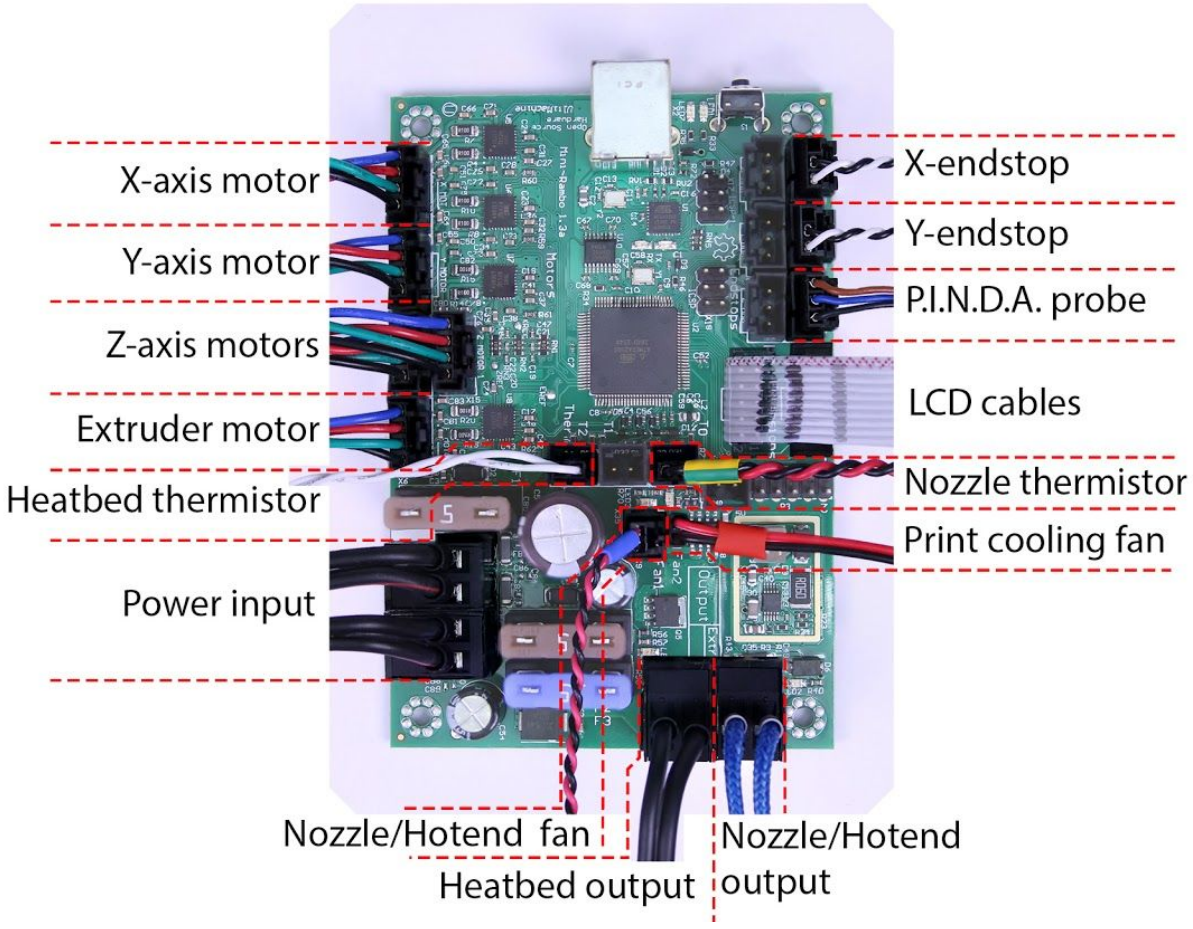
- Align the Y-axis so that each Y-corner is sitting on the table - none should be off the table.
- Align the tightening of each Y-axis threaded rod so that each Y-corner is perpendicular (facing upright) to the table surface.
- Align the tightening of each Y-axis threaded rod so that each M8 threaded rod is at a right angle to the M10 threaded rods - the Y-axis must form a perfect rectangle when viewed from above (pict. 36).




*Pict. 36 - Right angle between Y-axis and M8 / M10 rods*

## 13.2 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.



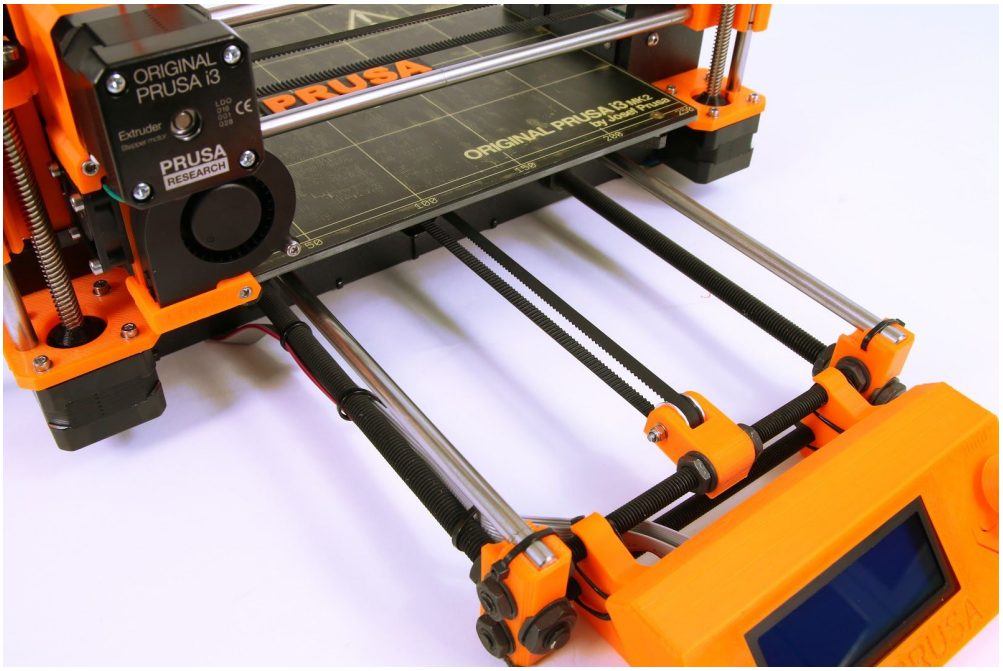

*Pict. 37 - Proper wiring of the connectors*

## 13.3 Printer can’t read SD card

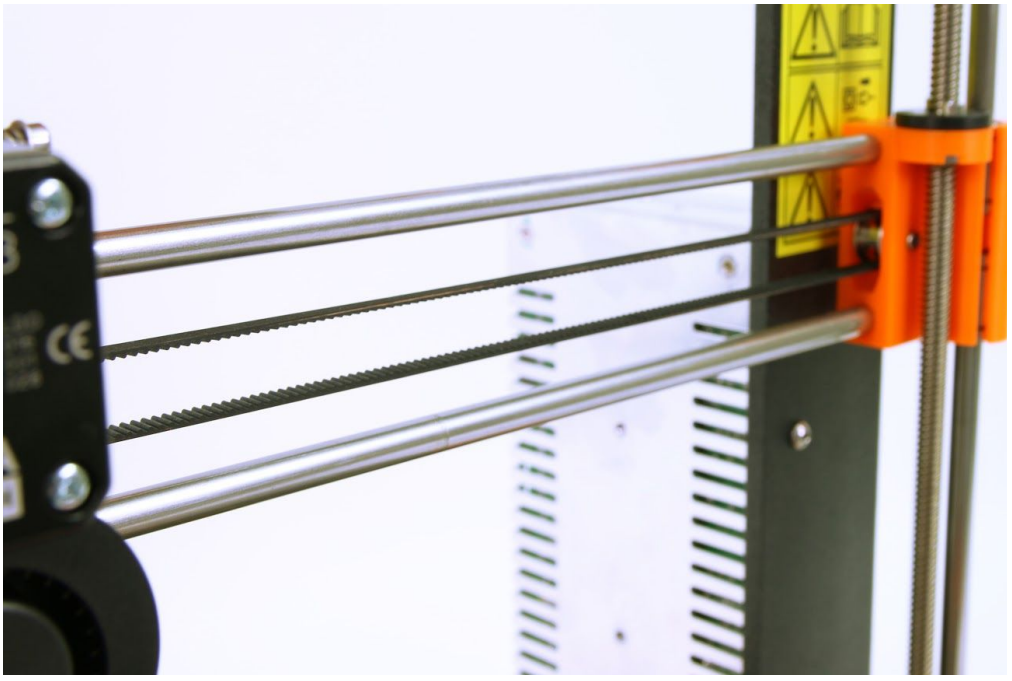
First, make sure that the **file name** on the SD **does not contain special characters** - otherwise the file could not be displayed on LCD. If there is no error in the file name, check the EXT2 wiring (from electronics to LCD). If the cable is connected properly, try to swap the cables.

## 13.4 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. 38 - A properly tightened Y-axis belt under the heatbed*



*Pict. 39 - A properly tightened X-axis belt*

## 13.5 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. 40 - Cables to be wrapped in a spiral wrap*

# Print and share!

Do not forget to tag your prints with #prusai3mk2 while sharing so we can find, pin and showcase them with our

Pinterest

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

*Happy Printing :)*