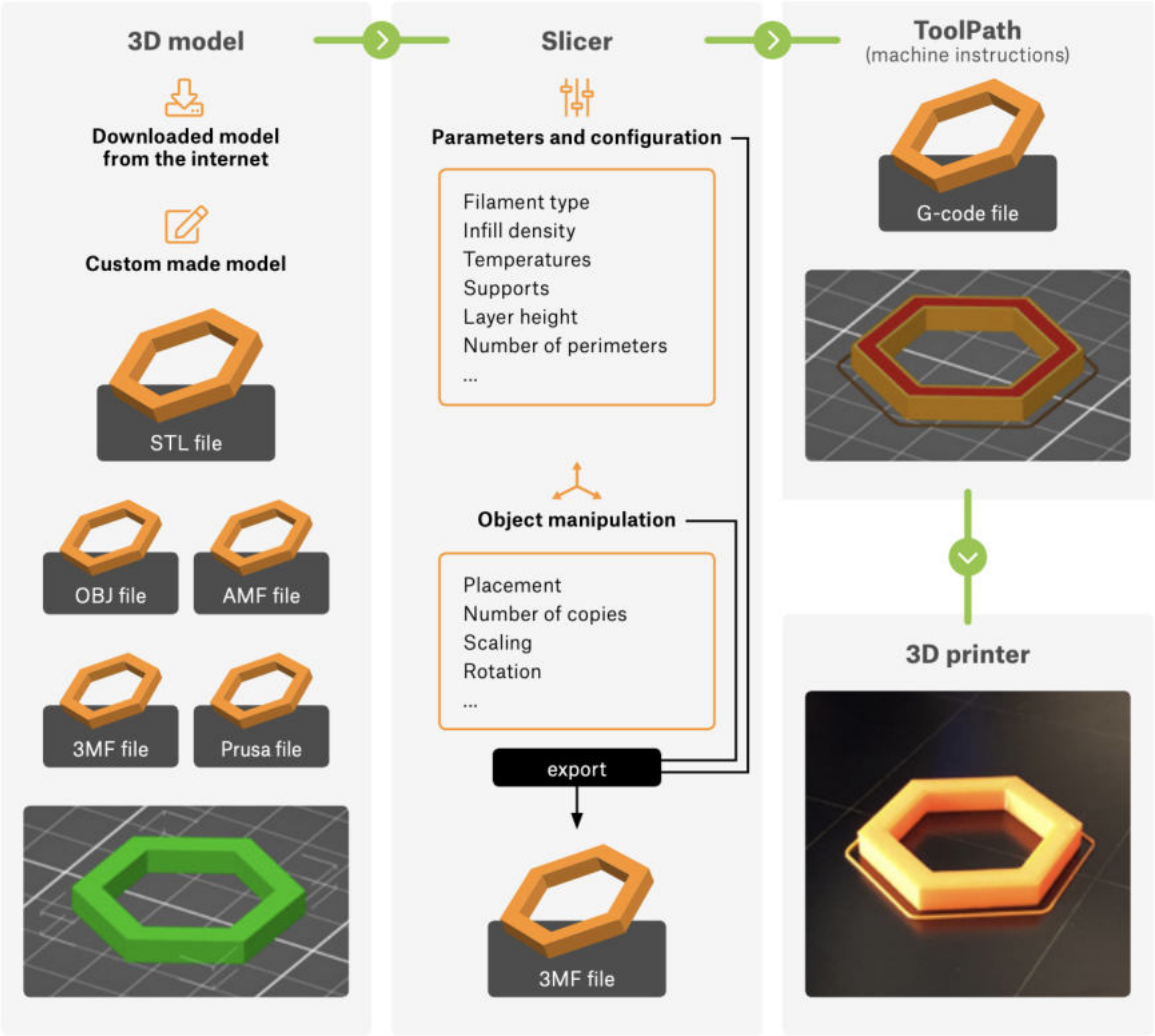
Instructions for the preparation of 3D models for 3D printing

This document contains the instructions for the correct execution of the complete 3D printing process, including preparation. It also provides tips and hints on how to prevent printing problems.

3D printer Original Prusa i3 MK3S+

Content:



The whole process of 3D printing is divided into three main parts.

Intro:

3D printing was, at first, used as a means for creating cheap and quick prototypes. As technologies became less expensive, 3D printers found their uses in other industries. It is the same for the laboratory tasks for Fluid Mechanics or Plumbing. The 3D printing process consists of three main steps. The first step is to obtain a 3D object, which is typically an STL file. However, this format is not recognized by 3D printers and it is not directly printable. To process an STL file, you need to use a specialized tool, commonly known as ‘slicer’. You can import an STL file into the slicer of your choice, configure printing parameters and then export the final result as a ‘G-code’, which is basically the original 3D object sliced into thin layers and converted into a set of movement commands recognized by 3D printers. The resulting G-code is printer-specific, so that’s why 3D objects are usually shared as STL files - users can then slice them for their printer / filament individually.

1. 3D models for the laboratory tasks

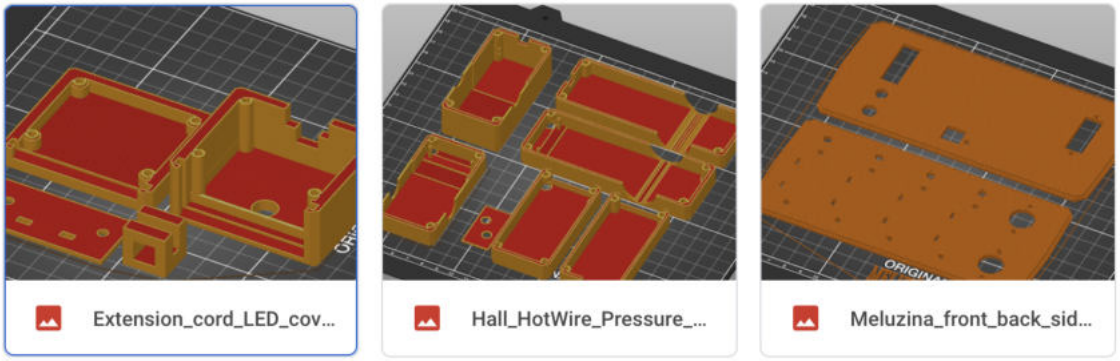
In attachment, there are available 3D models (STL format), which are ready for 3D printing in case the currently delivered parts are destroyed. The appendix is divided into several subfolders according to laboratory tasks or part type.

Each folder contains pictures of the models and the corresponding model in STL format.

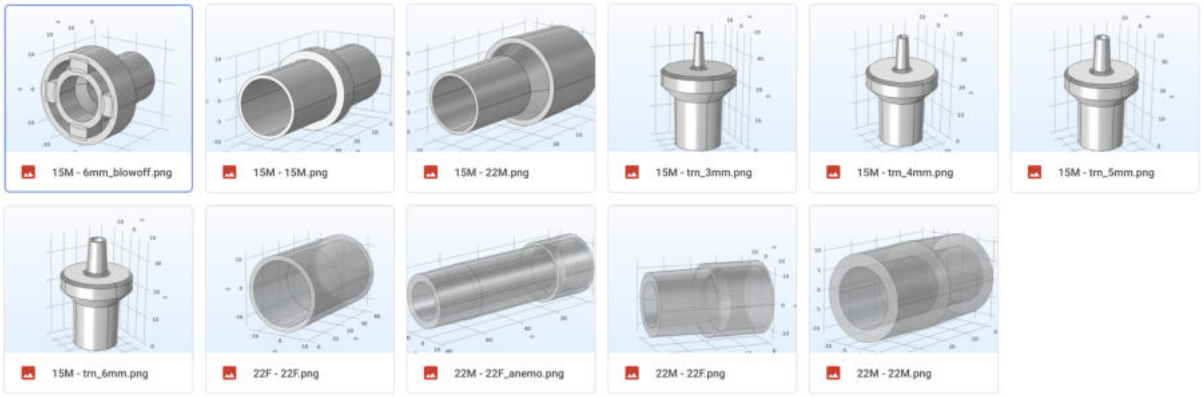
STL is a file format native to the stereolithography CAD software created by 3D Systems. This file format is supported by many other software packages; it is widely used for rapid prototyping, 3D printing and computer-aided manufacturing.STL files describe only the surface geometry of a three-dimensional object without any representation of color, texture or other common CAD model attributes.

The list of folders:

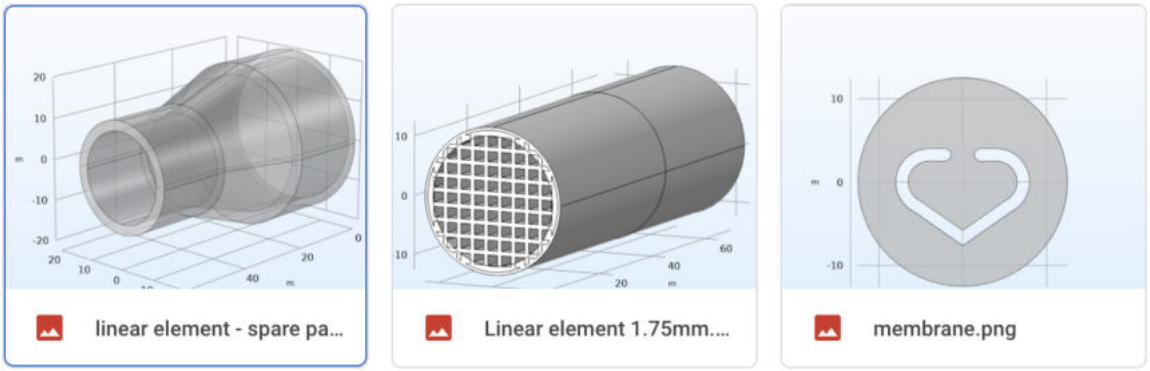
Covers (boxes)



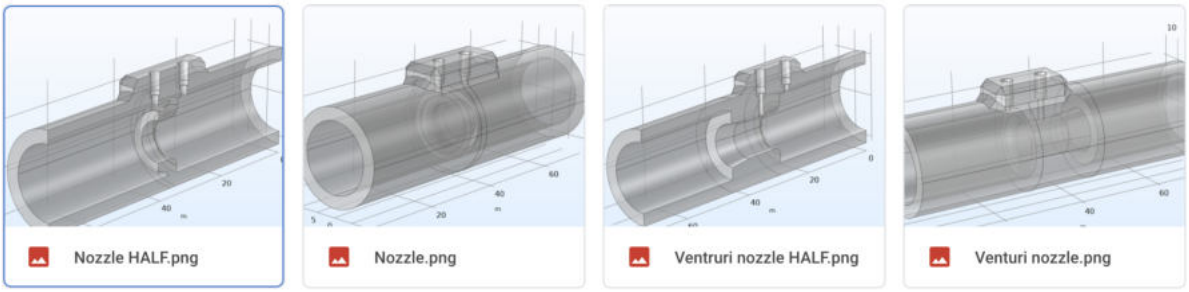
Junctions, spare parts



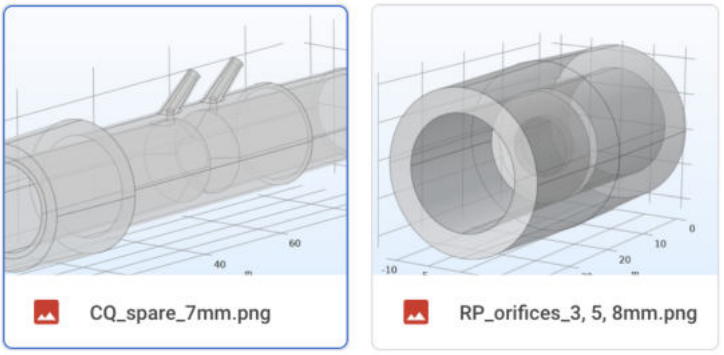
Linear elements



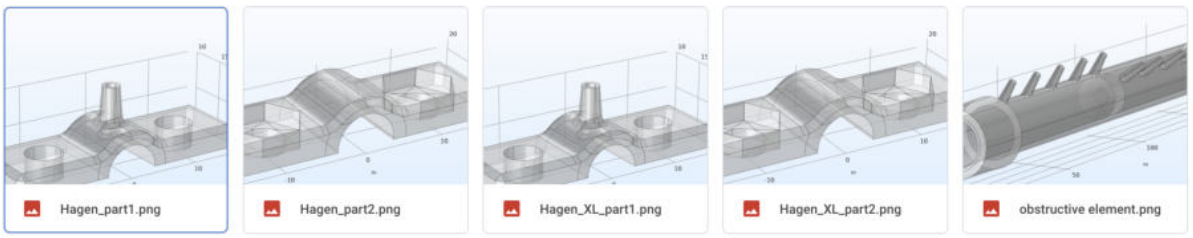
Nozzle, Venturi nozzle



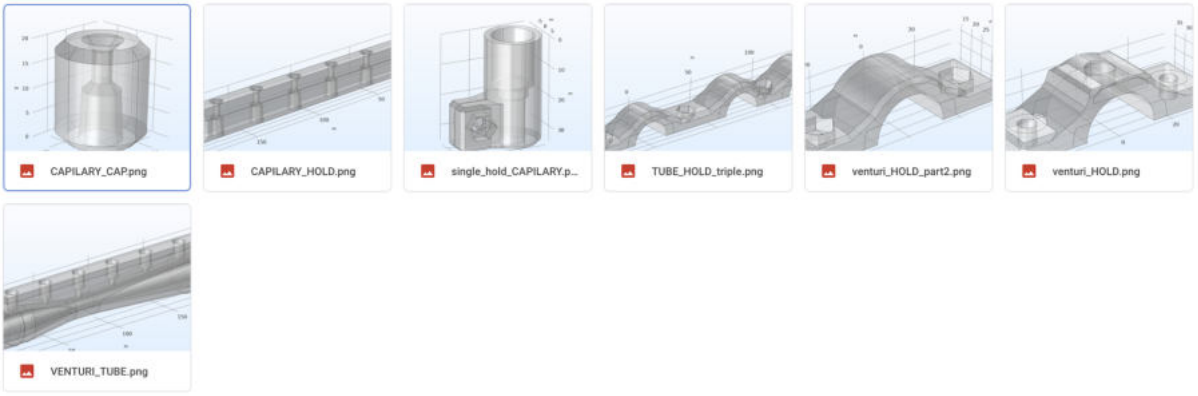
Parabolic element



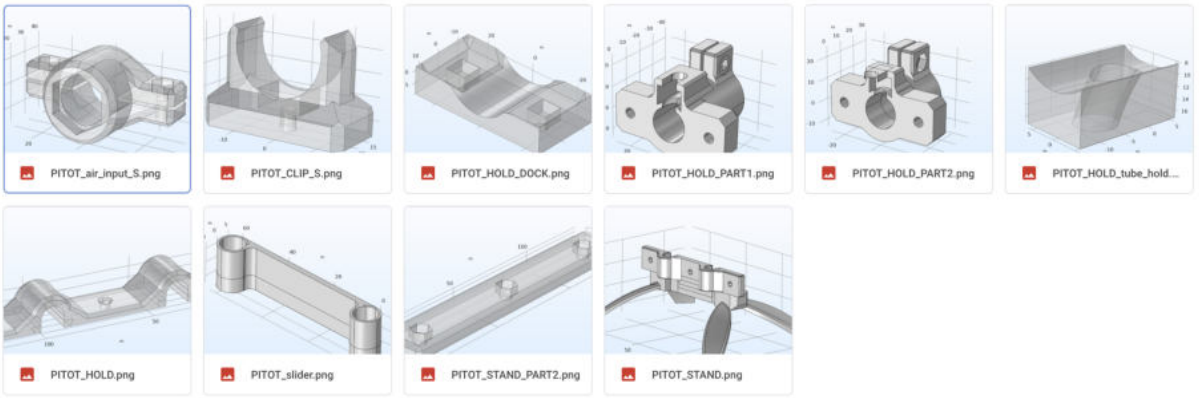
Laboratory task No. 2



Laboratory task No. 9

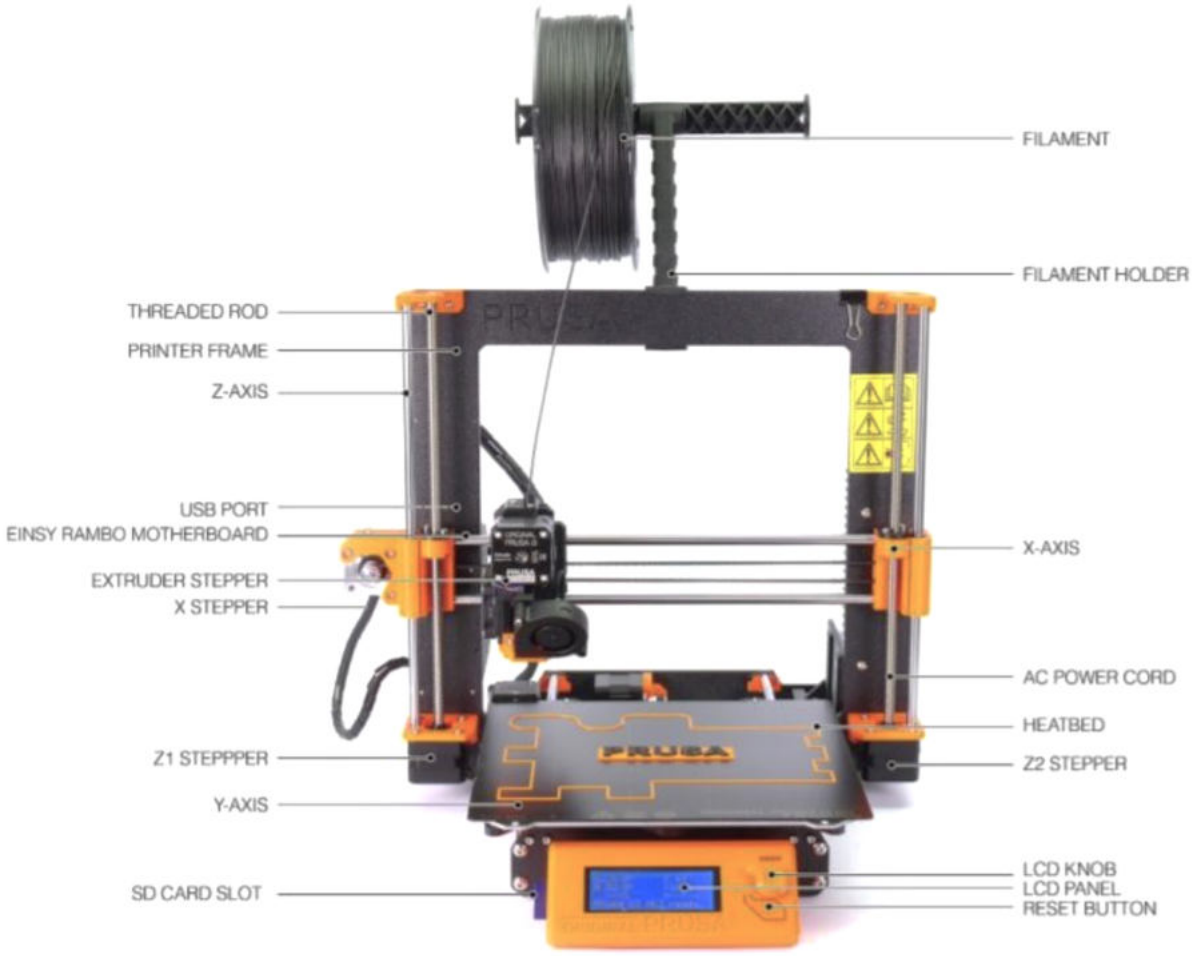


Laboratory task No. 10



2. 3D printer, functions, abilities and limitations

The original Prusa i3 MK3S+ is the successor to the successful MK2 model. The main advantages of the MK3S+ are an improved extruder, a range of sensors and smart features. In addition, the MK3S+ has a magnetic heated bed MK52 with replaceable flexible printing plates with a PEI surface. The printer has a large working space - 11.025 cm3 (25 x 21 x 21 cm) (X × Y × Z). It has an integrated LCD, printing from an SD card (8 GB in the package) or from a computer via USB. In the package, there is a 0.4 mm nozzle (easily replaceable) for a 1.75 mm printing string. The printer also has a fully automatic calibration of the printing surface with a layer height ranging from 0.05 mm. The heated pad compensates for cold corners, ensuring even cooling when printing all materials.



The main components of the 3D printer Original Prusa i3 MK3S+

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

Spring steel sheet - comes in two variants, smooth and textured. This is where the printed object appears. Please read the maintenance instructions carefully to ensure optimal conditions for 3D printing.

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

LCD Knob - main control of the Original Prusa MINI 3D printer: rotate left/right to scroll through options, press the knob to confirm your selection.

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

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

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

Z-Axis - the entire vertical axis assembly.

Extruder - the part of the 3D printer that ejects material in liquid or semi-liquid form in order to deposit it in successive layers within the 3D printing volume.

Troubleshooting - Extruder disassembly:
https://help.prusa3d.com/en/guide/2-extruder-disassembly_54504#54784

Print head - Lightweight print head consisting of the hotend (printing nozzle), PINDA sensor and two fans.

Nozzle - 0.4mm nozzle, can be replaced with e.g. 0.25mm or 0.6mm E3D-compatible nozzles to achieve various effects.

PINDA sensor - Induction sensor for auto-leveling.

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

The whole guide for new users is quickly described at this video:

https://www.youtube.com/watch?v=GE-lrRbU124&feature=emb_logo

3. Materials

Original Prusa i3 printers are compatible with a wide range of materials: easy to use and very popular PLA in a variety of colors, universal PETG suitable for printing mechanical parts, very strong and heat resistant ABS, composite materials resembling wood, bronze or copper, luminous materials in the dark and many more. The complete list of supported materials - PLA, ABS, PET, HIPS, Flex PP, Ninjaflex, Laywood, Laybrick, Nylon, Bamboofill, Bronzefill, ASA, T-Glase, Carbon-fibers enhanced filaments, Polycarbonates… Printing with individual materials differs according to several parameters - nozzle temperature, bed temperature, price, conductivity, strength,...

The most common materials are PLA and ABS and we will focus on them in next chapters.

Material	Printing with enclosure	Dry box recommended	Hardened nozzle required	Nozzle temperature	Bed temperature	Printable on powder coated sheet	Printable on smooth PEI sheet	Printable on satin sheet	Soluble with common solvents	Heat deflection temperature	Impact resistance Charpy	Tensile strength	Price
› PLA (12)	No	No	No	210 - 215 °C	60 °C	✓	✓	✓	✗				
› PETG (7)	No	No	No	240 - 270 °C	90 °C	✓	✓ with window cleaner	✓	✗				
› PETG HT (1)	No	No	No	270 °C	110 °C	✓	✓ with window cleaner	✓	✗				
› ASA (2)	Yes recommended	No	No	260 - 265 °C	95 - 110 °C	✗ not recommended	✓	✓ with enclosure	✓				
› ABS (6)	Yes recommended	No	No	240 - 255 °C	110 °C	✗ not recommended	✓	✓ with enclosure	✓				

Different parameters for each material

3.1 PLA

PLA is the most commonly used material for 3D printing. It is biodegradable, easy to print and PLA prints are very hard. The perfect choice for printing large objects due to low thermal expansion (prints do not twist on the substrate) and for printing detailed small models. It is the only material that is certified for printing 50 micron layers.

PLA has a relatively low melting point of 175 degrees Celsius. Unlike so-called thermosets, it is possible to repeatedly heat PLA through its melting point with very little degradation of the material. It is also a very hard, but therefore brittle material, and once it breaks, it often shatters.

Nozzle temperature: 215 °C
Bed temperature: 50-60 °C

3.2 ABS

ABS is a very strong and versatile material with excellent heat resistance. It is suitable for indoor and outdoor use.

ABS is a thermoplastic, which means that like PLA, it can be repeatedly melted and crystallized without degrading the material. However, ABS melts at a higher temperature

than PLA. The higher melting point gives ABS excellent heat resistance, your prints will not show signs of deformation up to 98 °C.

ABS contains an admixture of synthetic rubber, which makes it more resistant to wear and impact. And last but not least, the ABS filament is soluble in acetone! It is really easy to join several prints together, just lightly coat the contact surfaces with acetone. In addition, prints can be smoothed with acetone fumes to obtain a perfectly glossy finish. You need to be careful when handling acetone, but it is not nearly as dangerous as PLA solvents.

Nozzle temperature: 255 °C
Bed temperature: 100 °C (up to 110 °C for printing larger objects)

4. Preparation for 3D printing, pre-processing

Each model must be converted from stl format to g.code format before printing. The g.code format contains metadata that carries information such as nozzle temperature, print speed or retraction.

The complete instructions are on the following website:
https://help.prusa3d.com/en/article/first-print-with-prusaslicer_1753

4.1 Installation of the Prusa Slicer program

Before the first use, you need to download Prusa Slicer - download is possible here:
https://help.prusa3d.com/en/article/install-prusaslicer_1903

PrusaSlicer is an open-source, feature-rich, frequently updated tool that contains everything you need to export the perfect print files for (not only) your Original Prusa 3D printer. PrusaSlicer is available for Windows, Mac and Linux.

After installing the program, it is necessary to add a type of printer to the slicer (in our case it is an **i3 MK3S** printer). The printer is added in the “Printer Settings” tab after selecting the option to add a printer from the printer drop-down list.

4.2 Work in the Prusa Slicer program

After selecting the appropriate printer in the “Printer Settings” tab, go to the tab called “Print bed”. Using the menu in the top panel you can add new STL files designated to printing.

All the information are on this link:
https://help.prusa3d.com/en/article/configuration-wizard_1754

After adding the model to the printing bed, the important part comes, which is the correct positioning of the model on the printing bed. Improper rotation of the model leads to more frequent errors, reduced print quality and higher material consumption (for example due to the need to generate more supporting stands). First of all, it is necessary to decide what function the printout will perform. Based on the purpose, the printing will be set.

The ideal rotation of the model is such that a minimum number of supports is generated (ideally none). It is usually not possible to select a suitable rotation, which would mean printing without supports. In this case, it is necessary to rotate the model so that as many holes of circular cross-section are situated in the direction of the bed (for example tubes are ideal to print upright), because printing is not perfect and the circular cross-section may collapse and lose its shape and required properties.

For correct printing, it is also possible to divide the model into two or more parts, which can be glued after printing (the option of dividing the printout can be found in the side panel).

After positioning the model, it is necessary to select the parameters for the correct printing in the “Print settings” tab. First of all, it is necessary to think about how much the given part will be stressed. If it is a part that is more stressed, it is necessary to set a larger filling for it. For stress-free parts, the filling can be kept low (this saves material in the first place). The ideal filling for average stressed prints ranges from 25 to 50 percent. For printing on higher models, it is recommended to adjust the collar, which prevents the model from detaching from the bed during printing.

The option to print supports is also selected here (CAUTION! Slicer does not automatically add supports to where they need to be. Therefore, in this step, the intervention of the user is needed, who should have a clear idea of the printout.)

Next, the material from which the print is to be made is selected (suitable material for printing is described in more detail in Chapter 3). After selecting a material in the drop-down list above, the default values for this material are automatically pre-set. We know from experience that for PLA it is advisable to keep the setting at 215 °C for the nozzle and 60 °C for the bed in all layers and for ABS the temperature setting of 255 °C for the nozzle and 100°C for the bed in all layers is suitable.

For more advanced settings, we recommend using the mentioned links in chapter 8 to tutorial videos about working with the 3D printer

After setting the parameters, it is necessary to check the menu on the right side in the “bed” tab, in which it is necessary to select the required material for printing once again (IMPORTANT). If this step is skipped, the printer may start printing with settings for another material, resulting in a failed print.

After making all the settings described above, you need to click on the button in the lower right corner called "Slice". After slicing the model, it is possible to view the individual layers and parts of the model with the steps of the print head. The estimated printing time is also shown. This makes it possible to perform the last check before exporting the model.

After performing the check, it is necessary to click on the “Export g-code” button, which will appear instead of the slicing button. After exporting the model to the required storage (standard SD card included with the printer), it is possible to proceed with the preparation of the printer itself.

4.3 Preparing the 3D printer

After generating the g.code file for printing, it is also necessary to prepare the printer for printing. First of all, it is necessary to check its condition (whether it is damaged or significantly soiled).

After such an inspection, it is a good idea to make sure that the spring steel sheet is properly degreased and, if necessary, clean it again. Use the supplied chemical isopropyl alcohol (IPA) to clean the spring steel sheet.

It is also necessary to start preheating. This depends on the material used for printing. It is also very important to keep in mind that the previous printing could have been done with another material that remained in the nozzle. Therefore, it is necessary to clean the nozzle with the material to be used, but it is also mandatory to use the temperature of the material from which it was last printed (for example when printing with PLA it is necessary to introduce the material into the nozzle at 255°C if it was printed with the ABS material before this).

After preheating, it is necessary to select the option of introducing filament into the nozzle in the printer menu. The filament must be ready for insertion so that its end is cut off at the tip so that it can be easily picked up by the extruder rollers. If the printer is calibrated within the Z-axis setting, it is ready to print after the filament is fed into the nozzle.

Also, before printing, make sure that an SD card containing a g.code file for printing is present in the printer.

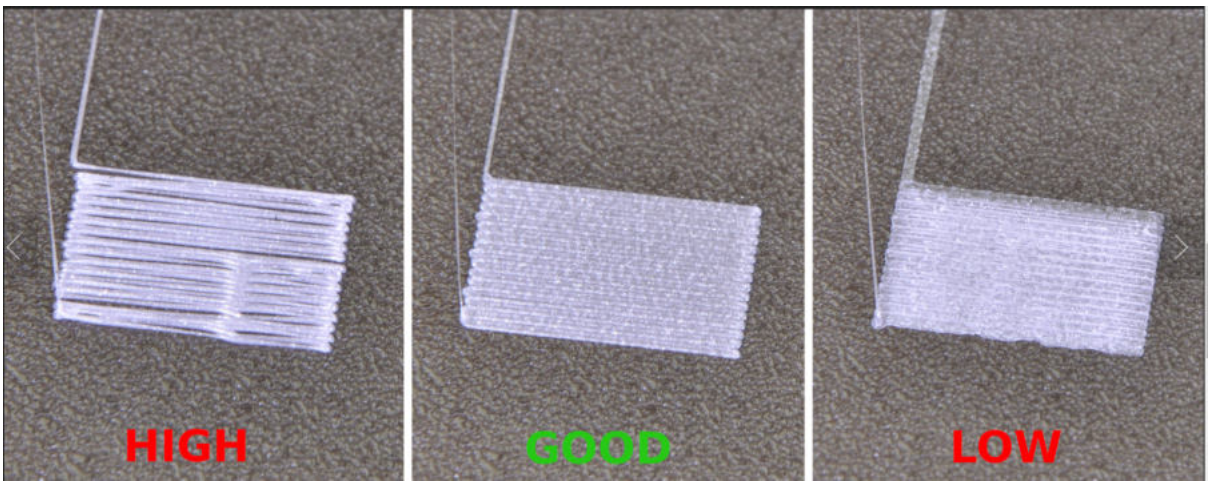
If the Z axis calibration has not been performed, please pay attention to chapter 4.3.1 and perform the calibration before printing.

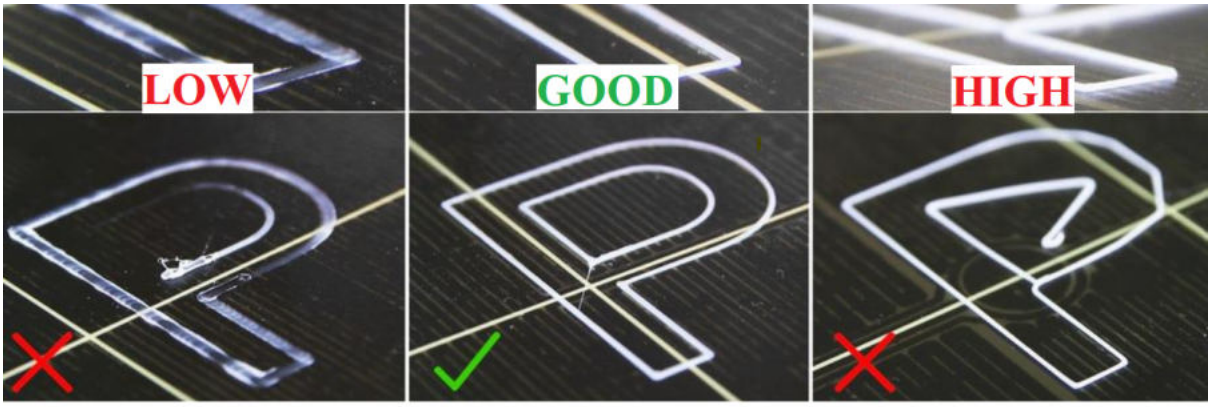
4.3.1 Calibration of the Z-axis and the first layer

Calibrating the printer is a very important step when you first start with a new printer and when you regularly maintain the printer. When calibrating the Z axis, the default distance of the printhead print head from the bed is selected. The Z axis setting must be correct.

If the print head is too high and far from the heat bed, there is a risk that the printed filament will not adhere to the bed at all and will start to wind on the bottom of the extruder with the nozzle. Conversely, if the head is too close to the bed, the nozzle will squeeze the filament to the sides after extrusion, creating an uneven surface for the first layer. In addition to the risk of damaged printing, there is also a risk of the nozzle scratching the printing bed. This can damage the bed and nozzle.

This figure shows what the printed filament should look like if the Z-axis calibration is performed correctly.





How the printed filament should look like if the Z-axis calibration is performed correctly

5. Initiation & process of 3D printing

After performing the Z-axis calibration, it is possible to proceed to the printing itself. Make sure an SD card is present in the printer and filament is loaded in the extruder.

After selecting the g.code to be printed, the printer will start working on its own. Initially, it prints a strip on the front of the spring steel sheet. This strip serves as a check for the user that the print head is properly prepared for printing (that the nozzle is not dirty, that filament is inserted, ...)

From this point on, the printer requires no further user intervention if it is set up correctly.

During printing, it is possible to fine-tune the Z axis (recommended for advanced users) and change the print speed. The print speed is changed by turning the main button directly on the information page of the given print. It is verified that a lower print speed (85 - 90%) improves print quality (especially for ABS printing). Further reduction of the printing speed does not have a significant effect on the quality (perhaps only in selected special cases). If you have time, we recommend downloading the print speed to about 90%.

6. After 3D printing

When the printing is finished, you will remove the printed models from the printing area and then you should take care of your 3D printer and turn it off.

6.1 Postprocessing of printed models

After printing, you need to remove the printed models from the printing area. To remove the printed models, it is recommended that you remove the entire spring steel sheet and flex it several times. This makes it easy to separate the printed parts.

Do not use sharp tools to remove models from the printing bed! The bed may be scratched when removed. Due to the scratches, there is a growing risk that further prints will not adhere sufficiently to the spring steel sheet and the print will spoil, or that the first layer of printing will never be printed completely smooth and uniform.

If the print contains supports (supporting stands), it is necessary to remove them from the printing bed with a suitable tool (pliers, tweezers, scalpel) after removal.

If the print is made of ABS material, it is also possible to perform acetone vapor smoothing (see Prusa manuals).

6.2 Maintenance of the 3D printer and preparing for another printing

After removing all pieces of plastic from the printing bed, it is necessary to degrease it with isopropyl alcohol (in an emergency, water with detergent will suffice, e.g. for washing dishes). If the printer is not used immediately, it can be turned off immediately after all phases of printing. No need to wait for the printer to cool.

7. Troubleshooting

The following section describes the most common printing problems.

The model does not attach to the print bed:

If the model does not stick to the spring steel sheet when you start printing, you need to check the correct printer settings. The detachment of the model can occur most often for two main reasons:

a) Bad calibration of the first layer
- Solution: Recalibrate the Z axis and the first layer

b) Incorrect nozzle temperature setting and heated heatbed
- Solution: Check the print temperatures and adjust them accordingly

TIP: To better attach the models to the heatbed, it is possible to choose in the Prusa Slicer program the option of creating a collar that helps the attachment. It is also possible to use glue on paper. Use this adhesive to coat the platen before printing.

Complete instructions: https://help.prusa3d.com/en/article/first-layer-issues_1804

The model twists:

a) Model twisting often occurs when printing from ABS material, which requires a higher temperature for printing. Models that are thin, tall, or thin layers of material are formed during printing (possible control via Prusa Slicer).
- Solution: The model must be correctly positioned in the slicer on the printing plate, or it must be divided into two or more parts so that there are no places during printing that could cause the model to twist.

Complete instructions: https://help.prusa3d.com/en/article/warping_2011

Shift layers when printing:

When moving the layers, very often the model is completely destroyed and the printing needs to be repeated. This defect most often has two main causes:

a) Twisting of the model and subsequent displacement of the layer due to nozzle jamming
- Solution: Proceed as for solving the twisted model

b) Unlubricated extruder guide bars
- Solution: Lubricate the guide rods with the supplied lubricant.

Complete instructions: https://help.prusa3d.com/en/article/layer-shifting_2020

Wavy filament when printing a simple line:

a) If the printed filament is curled and wavy when printing the starting line, it means that there is dirt in the nozzle
- Solution: Clean the nozzle. The nozzle can be cleaned either by heating to a higher temperature (min. 250 °C) and cleaning with a filament or manually using a suitable tool (included cleaning wire or thin metal string)

TIP: We recommend cleaning the nozzle from time to time as part of maintenance.

Complete instructions:
https://help.prusa3d.com/en/article/extrusion-problems-prevention_2027

Model printing repeatedly fails:

a) Occasionally, it happens that the printed model begins to decay in a certain part and the creation is destroyed
- Solution: It is necessary to adjust the settings of printing parameters (reduce the printing speed, or change the material) or change the properties of the model in the Prusa Slicer program (remove or add a perimeter, adjust the model fill, overhangs, supports, ...) or change its rotation relative to the printing bed.

Wind:

a) Sometimes, when printing, an error occurs that destroys the entire printout. This error can be caused by a draft (wind) in the room with the printer. This problem occurs especially when printing from ABS material.
- Solution: Avoid drafts in the room, making sure that there is not too much movement around the printer when printing. It is also possible to use the "skirt" function, which allows you to create a thin wall around the model, which protects the printed model from drafts and keeps a suitable temperature around it.

Filament extrusion repeatedly fails:

a) This problem occurs when the extruder has too tight or loose rollers.
- Solution: Adjust the extruder roller pressure setting. This adjustment is done with two screws on the side of the extruder body.

Complete instructions:
https://help.prusa3d.com/en/article/extrusion-problems-prevention_2027

Complete instructions:
https://help.prusa3d.com/en/article/under-extrusion_2007

For more troubleshooting information, see:
https://help.prusa3d.com/en/tag/mk3s/print-quality-troubleshooting_794

8. Tips and tricks

We hope the Original Prusa i3 MK3 helps you to complete all your Fluid mechanics and Plumbing laboratory tasks. Although it may seem difficult to use a printer, the opposite is true. At the same time, you will learn new skills during printing, which will move you a bit further in life.

Five things you may not know about your Original Prusa i3 MK3:
https://www.youtube.com/watch?v=ECE5zQJwjqM&feature=emb_title

Guide for the Original Prusa i3 MK3
https://help.prusa3d.com/en/tag/mk3s-2/

Basics of 3D printing:
https://www.prusa3d.com/wp-content/uploads/basics-of-3D-printing.pdf