VILNIUS TECH
LinkMenų fabrikas

3D PRINTING GUIDEBOOK

Safety 101

Printers are safe to use. They are made to be user-friendly. However, you migh hurt yourself if not handled properly or carefuly.

Heat

Beds and nozzles are hot. Don't touch them or be careful when doing-it.

CUT

When handling sharp tools like pliers or cutter blades, be careful not to injure yourself

VAPOR

Don't stay for too long in the lab. Melting plastic produces fumes.

SLA printers use toxic resins

There is a **First Aid Kit** in every lab. Look for it when using the lab for the first time. In case of emergency ask for a **staff member** or call **112.**

ALAWAYS report any incident.

Few ground rules

1

Reserve your printer and time slot before use.

2

Always clean after you. Don't leave trash on the tables. It's always nice to work in a clean environment

3

Don't touch other users belongings, this includes filaments.

4

When the print has ended, remove it so others can use the printer.

5

Don't hesitate to ask for help. It's better to ask for help than to do something wrong.

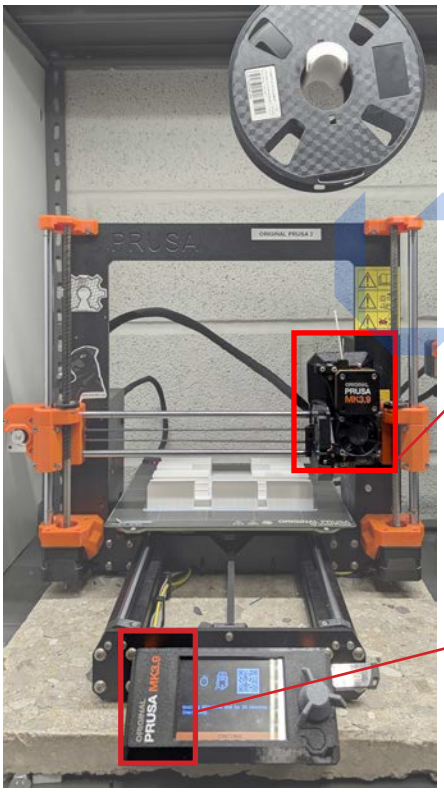
VILNIUS TECH
LinkMenų fabrikas

The Printers

FDM and SLA

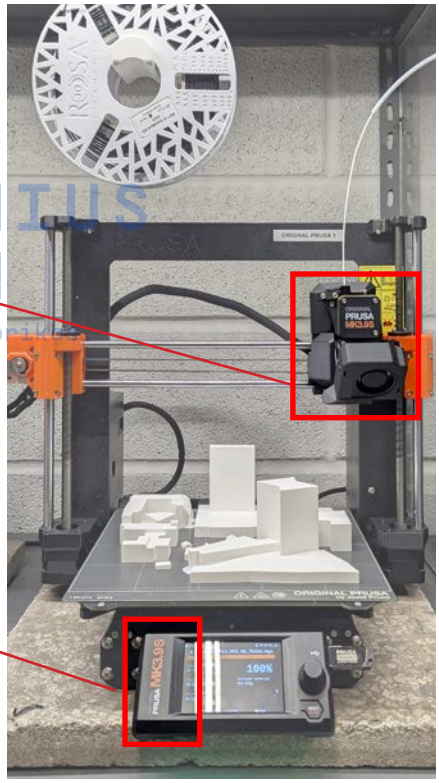
4 FDM printers

"Fused Deposition Modeling"



Extruders are differents. The MK3.9s have an improved ventilation

The model can be found next to the screen



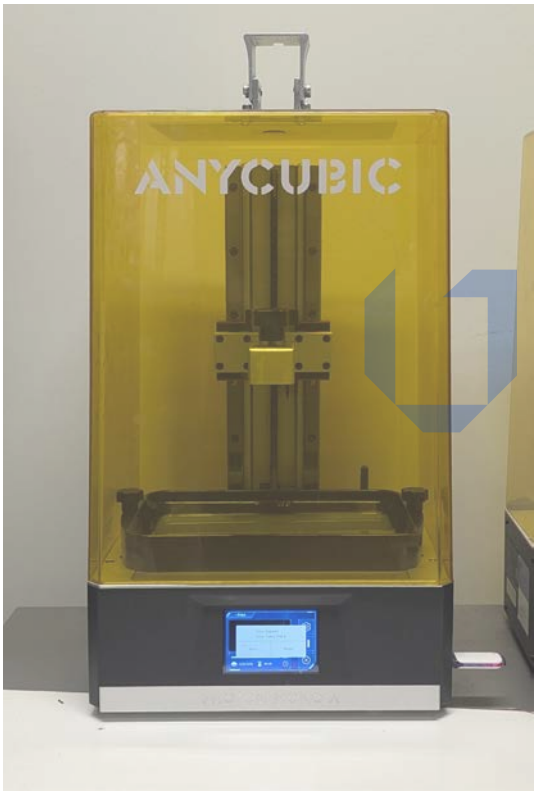
Original Prusa MK3.9
0.4 mm nozzle

Priginal Prusa MK3.9s
0.4 mm nozzle

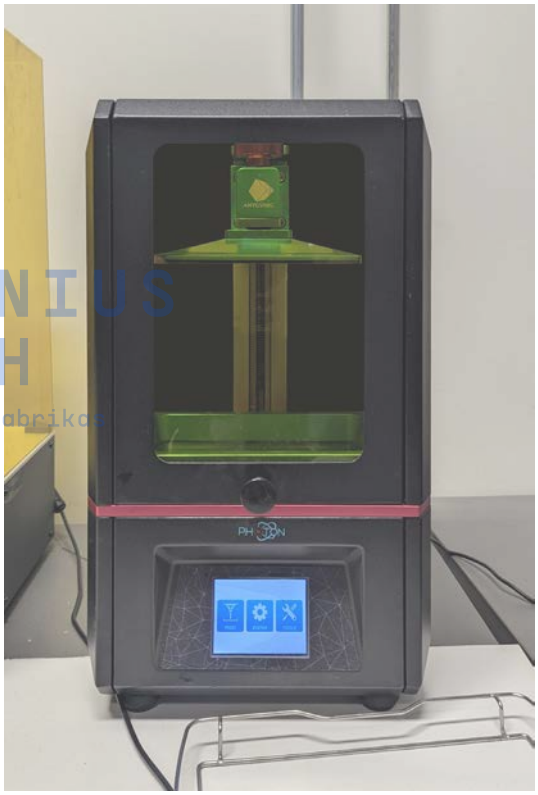
Original Prusa mini+
0.4 mm nozzle

X2

2 SLA printers “Stereolithography”



Anycubic Photon Mono X
192mm X 123mm X 245mm



Anycubic Photon
115mm X 65mm X 155mm

1 Wash and Cure

Anycubic Cure and
Wash

Words of wisdom

SLA prints uses toxic resin that reacts to light. They need to be washed and cured after each prints to get rid of resideual resin.

Always use SLA printers under the supervision of a **staff member**!

Four basic steps to 3D print

(More details in the following pages)

1

Export your file in **.STL**, **.3mf** or **.Step** format

2

Import your file in PrusaSlicer. If needed, add supports, change the infill type/density and change the speed.
Export the **G-Code** file on the flash drive.

3

Prepare the **booked** printer, load the filament, clean the bed before and after use.

4

Start the print and **STAY** at the beginning (at least for the first layer)

Words of wisdom

There are 3 types of files used in 3d printing:

STL: Simple mesh format, no colors or metadata.
3MF: Advanced format with colors, materials, metadata support.
STEP: CAD format with precise geometry, ideal for editing.

VILNIUS TECH
LinkMenų fabrikas

How To ?

How to setup PrusaSlicer
How to print using FDM
How to prepare your file
How to print using SLA

PrusaSlicer?

https://www.prusa3d.com/page/prusaslicer_424/

PrusaSlicer is a software used to turn 3D models into instructions (G-code) that a 3D printer can understand. It's like the translator between your design and the printer.

On PrusaSlicer you can :
import your 3D model, adjust size, position, and orientation, choose settings like layer height, speed, and material, generate a printable file for your machine

How to setup PrusaSlicer?

1

Download the software from the website or QR code shown prevously

2

Click “next” untill the download starts

3

A window will oppen to setup the softaware according to your needs

4

Go to “prusa research” and select the printers you have. In our case, select:

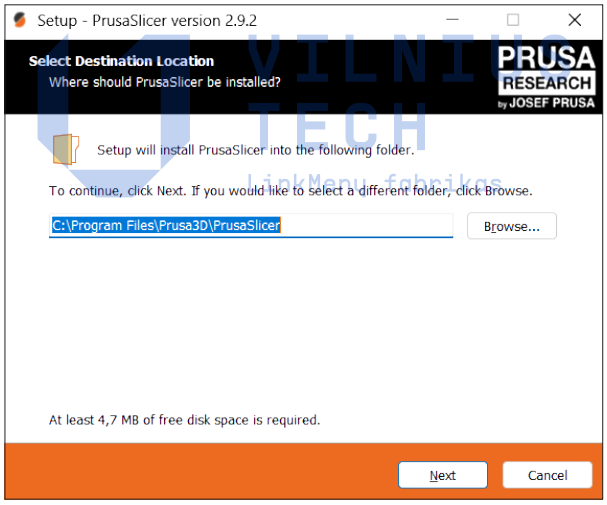
Prusa MK3.9 input shaper (0.4mm nozzle)
Prusa MK3.9s (HF 0.4mm nozzle)
Prusa MINI and MINI+ (0.4mm nozzle)

5

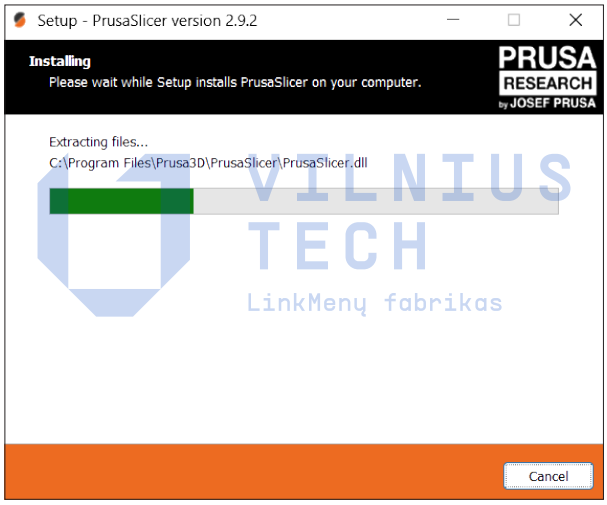
Time to choose the filaments, select “generic PLA” and “generic PETG”

6

Choose “simple mode”, it will show basic functions, you can change it later if needed



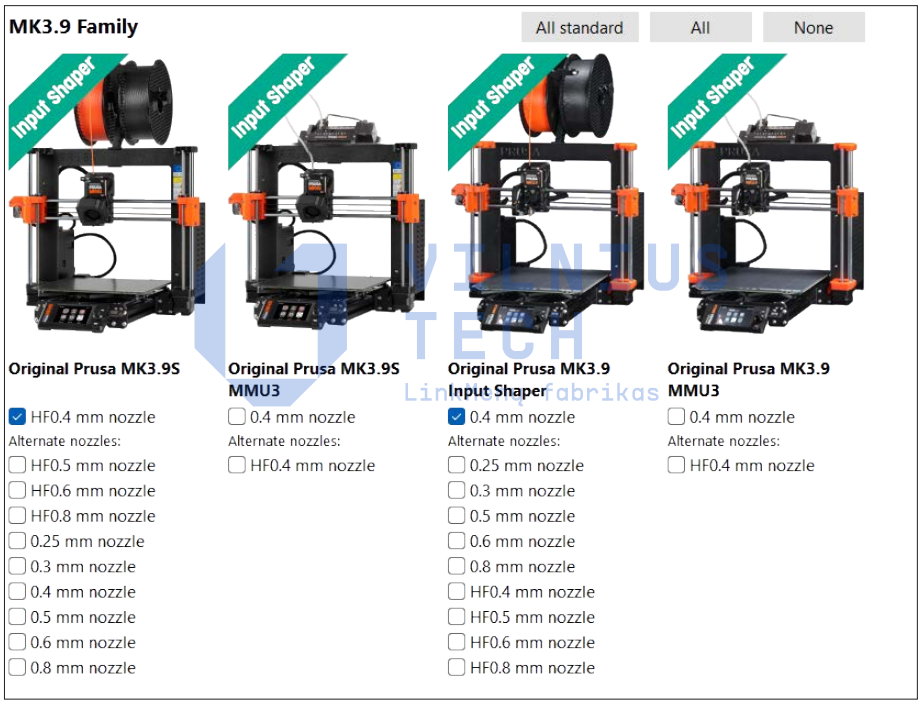
1. Download PrusaSlicer.



2. Click “next” untill the download starts.

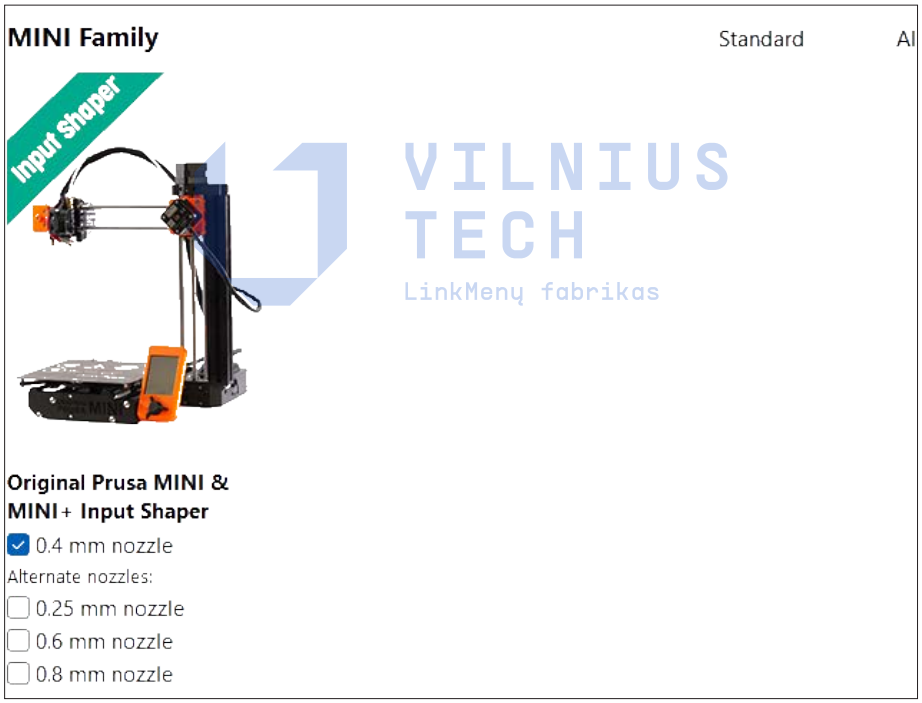
3. A window will oppen to setup the softaware according to your needs.

4. Go to “prussa research” and select the printers you need.

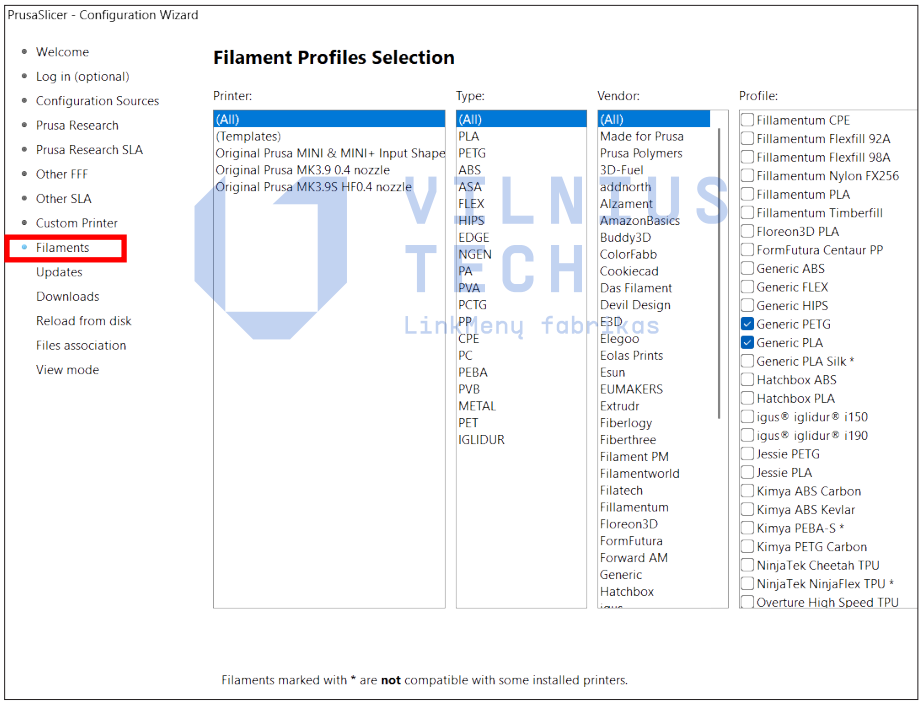


In “prussa research”, select the printers you need. In our case, select:

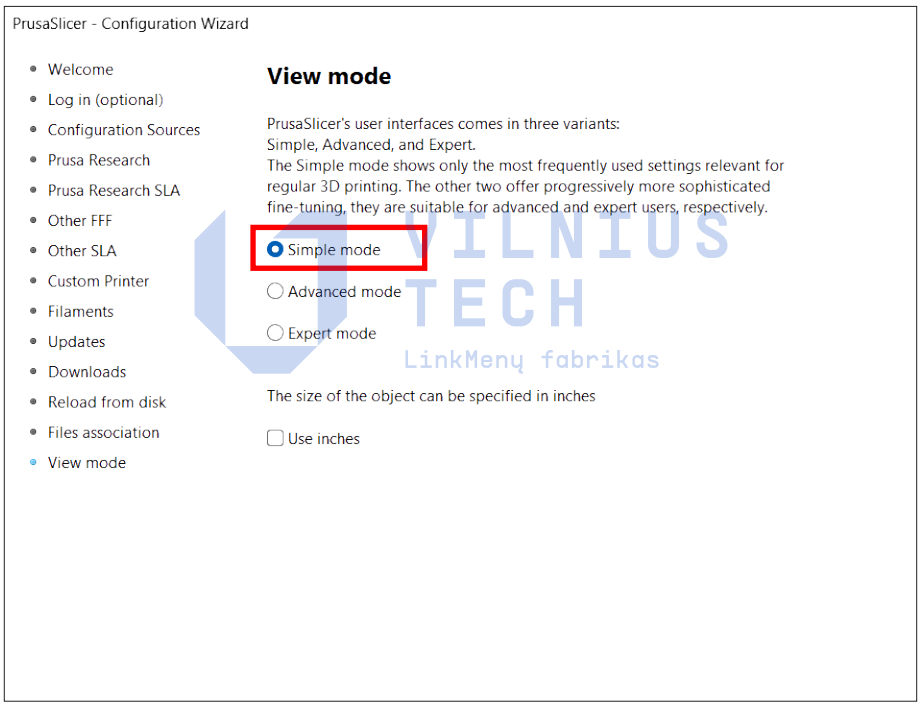
Prusa MK3.9 input shaper (0.4 mm nozzle
Prusa MK3.9s (HF 0.4 mm nozzle)



Prusa MINI and MINI+ (0.4mm nozzle)



5. Click “next” until you get to “Filaments” menu then select “generic PLA” and “generic PETG”.



6. Choose “simple mode”, it will show basic functions.

How to prepare your file?

1

Import your **.STL**, **.3mf** or **.Step** file into PrusaSlicer. Make sure that every imported pieces are touching the bed.

2

Chose the recommended infill pattern and density according to your needs. Add the minimum amount of supports when required. (annex p.29)

3

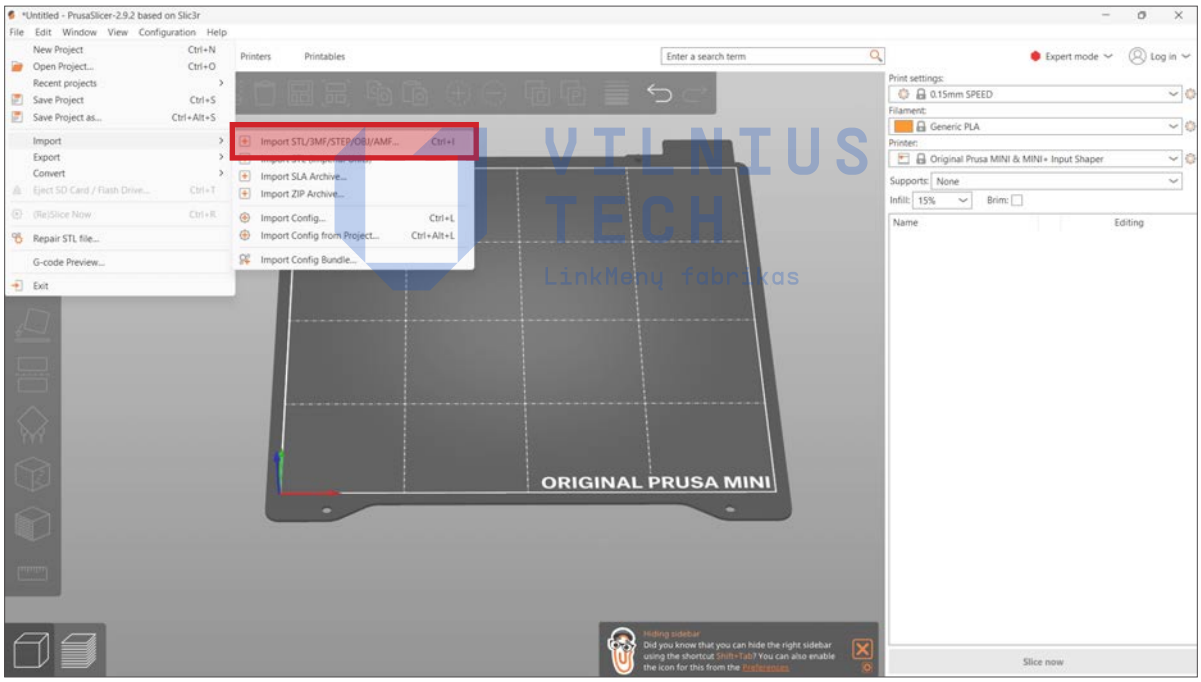
Use the arrange function (A) to automatically arrange the printings for better time consumption during printing.

4

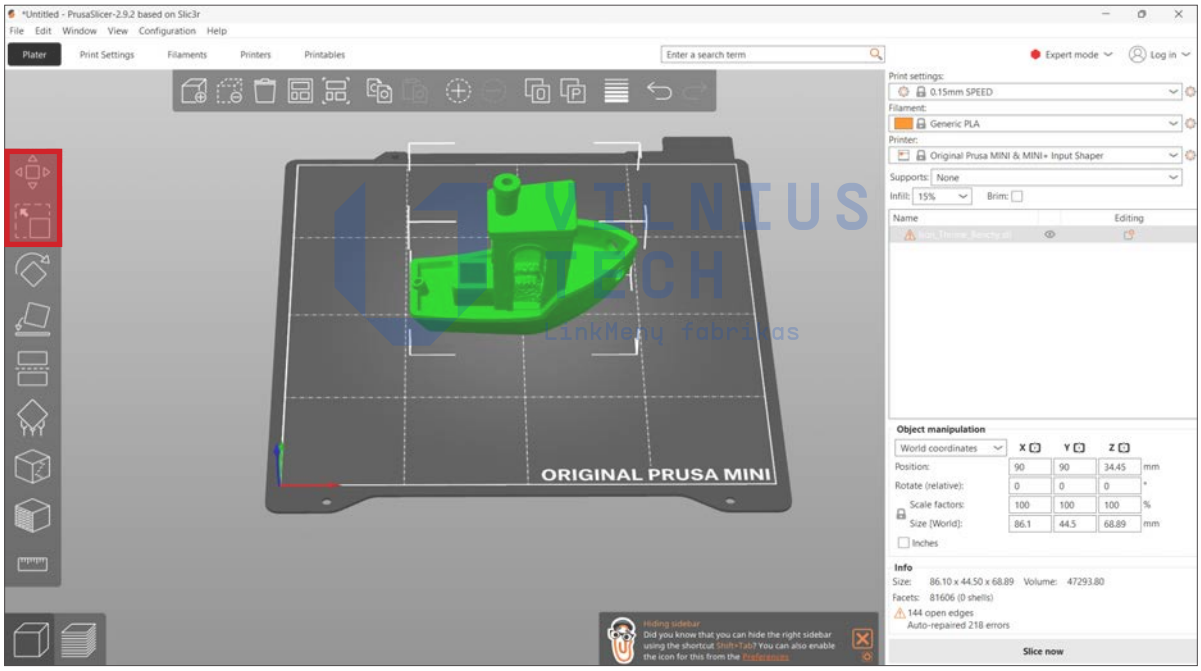
Adjust the print settings according to the strength and details needed. Select the **CORRECT** filament (PLA or PETG). Select the **CORRECT** printer. Each printer has its own pre-set. (annex p.27)

5

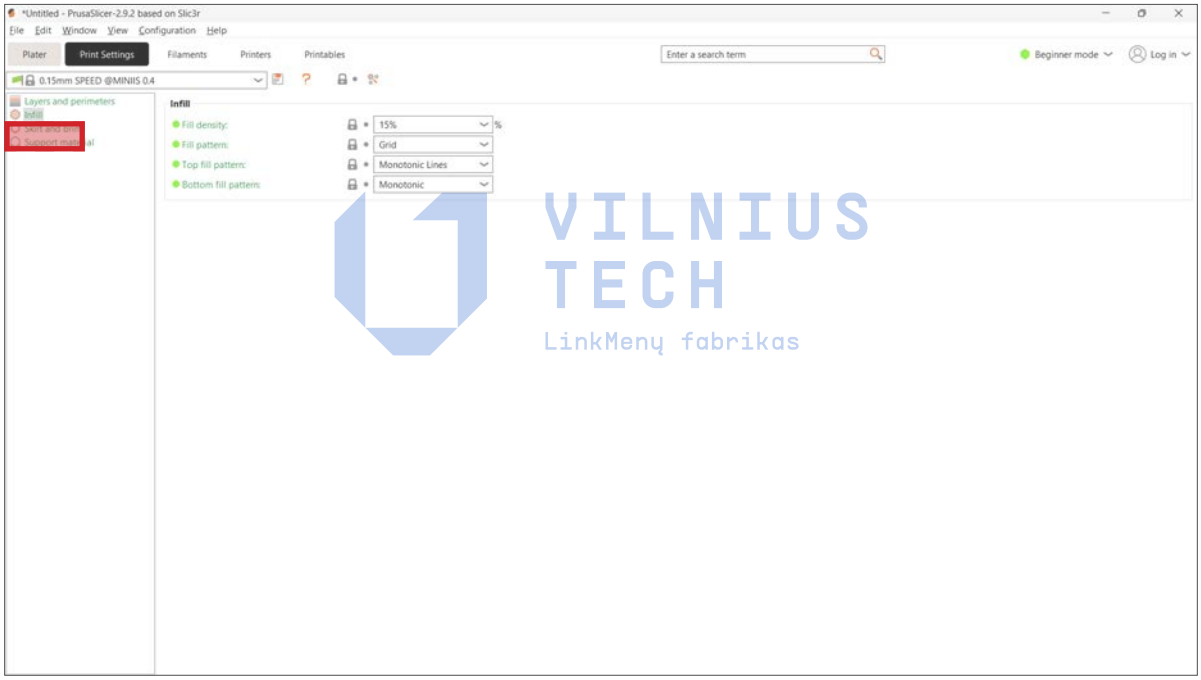
When everything is ready, export the **G-Code** file onto the flash drive. It should appear automatically next to the “export” button.



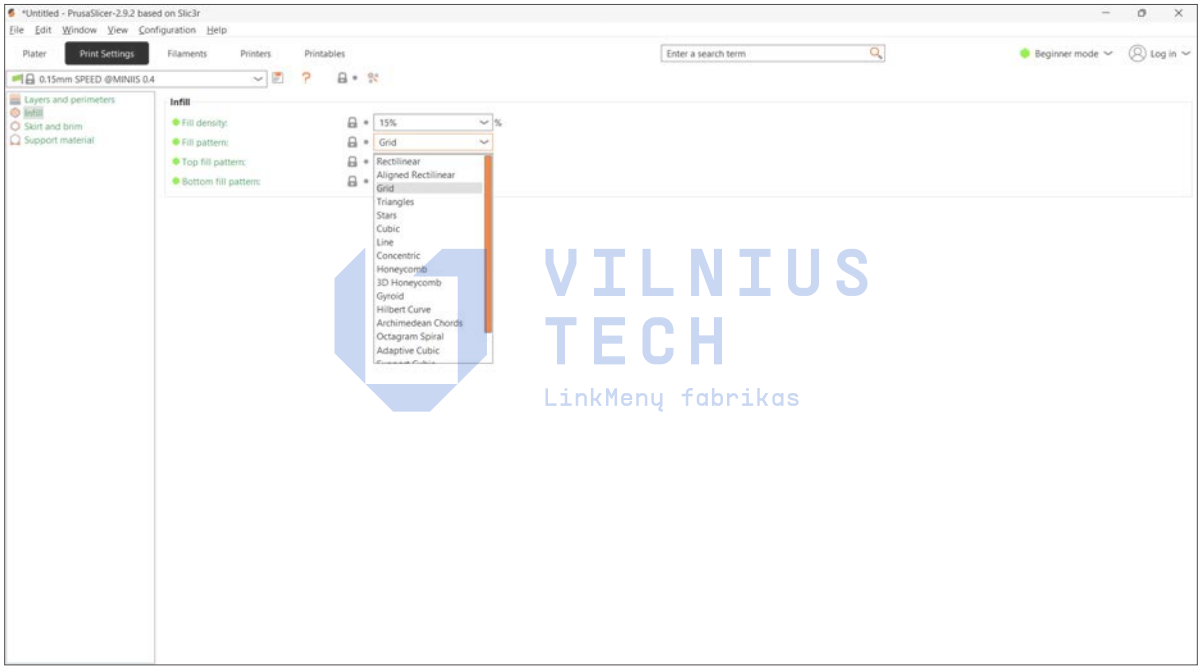
1. Import your .STL, 3mf or Step file into PrusaSlicer. Make sure that every imported pieces are touching the bed.



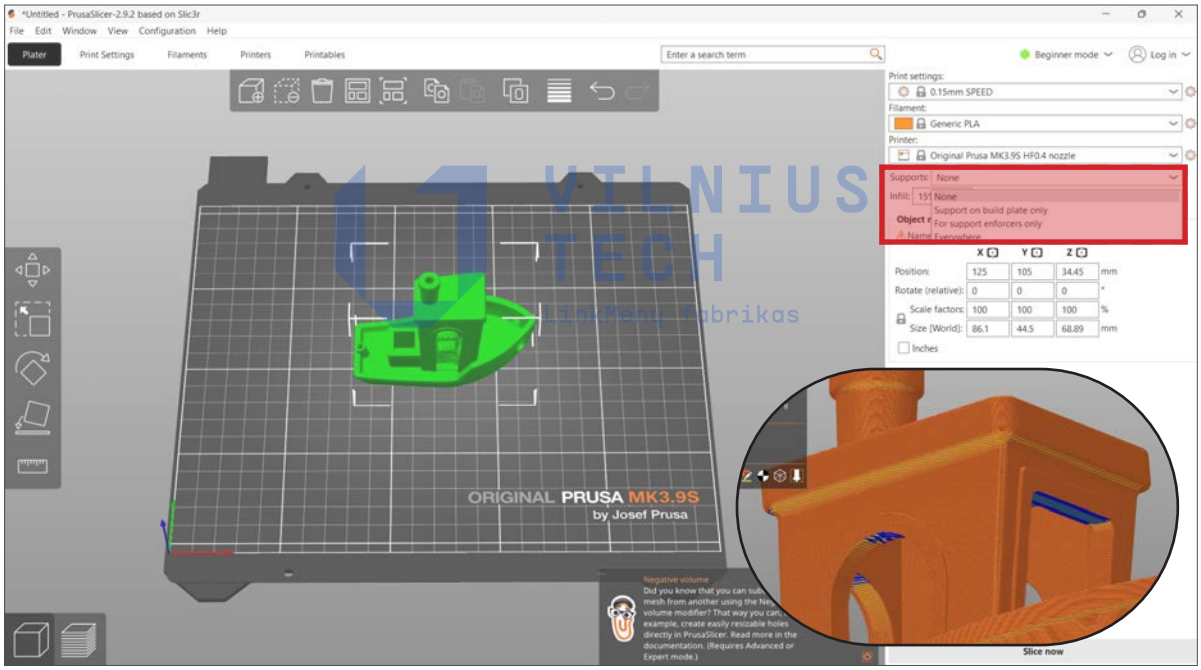
The model appears in the center. You can reorient or move it if needed by using these two highlighted functions.



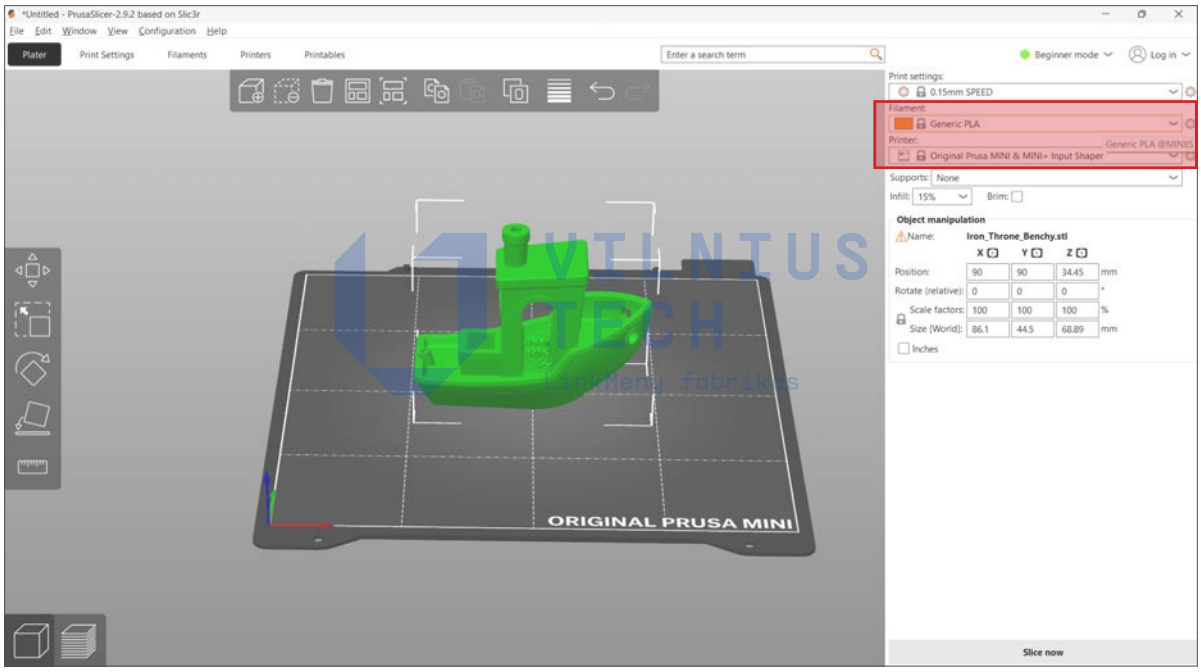
3. Choose the infill pattern and density that best suit your needs. (more info about the patterns in the annex)



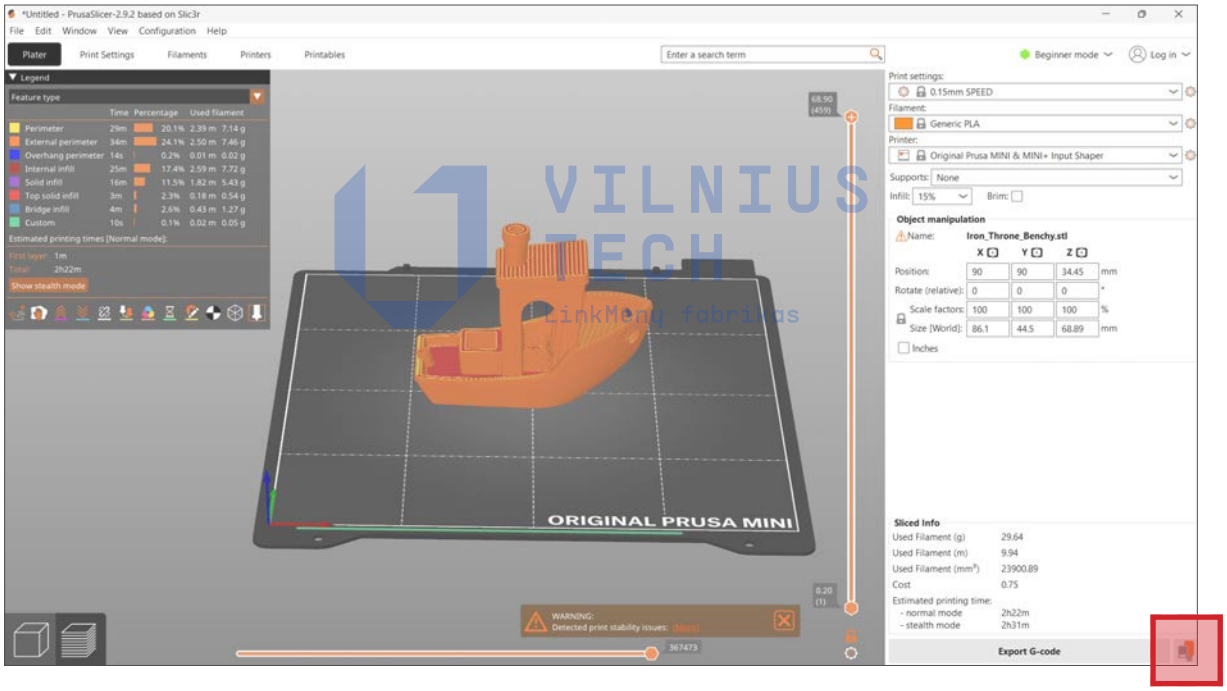
Changing the type of infill and infill density will change the strength of your prints.
To improve this strength you can also rotate your model to change the layer's orientation.



Add supports when required. The parts that need support appear in **blue** on the model when you slice it.



4. Adjust the print settings according to the strength and details needed. Select the **CORRECT** filament (PLA or PETG). Select the **CORRECT** printer. Each printer has its own pre-set.



5. When everything is ready, export the G-Code file onto the flash drive. It should appear automatically next to the button “export”.

How to print using FDM?

1

Clean the bed using tissu and alcohol solution in order to remove any residu on the bed. Don't use sharp tools, that will damage the bed.

2

Load your filament in the printer's nozzle. Depending on the printer, it should detect it automatically. In the menu, select your filament type (PLA or PETG). It will start extruding to clean the nozzle. Follow the instructions on the screen.

3

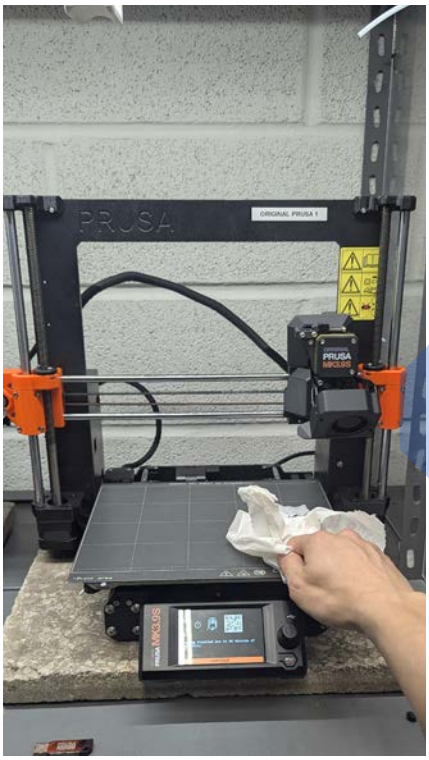
Plug the flash drive into the printer usb port. It will detect and show you your file. Press "print" when ready.

4

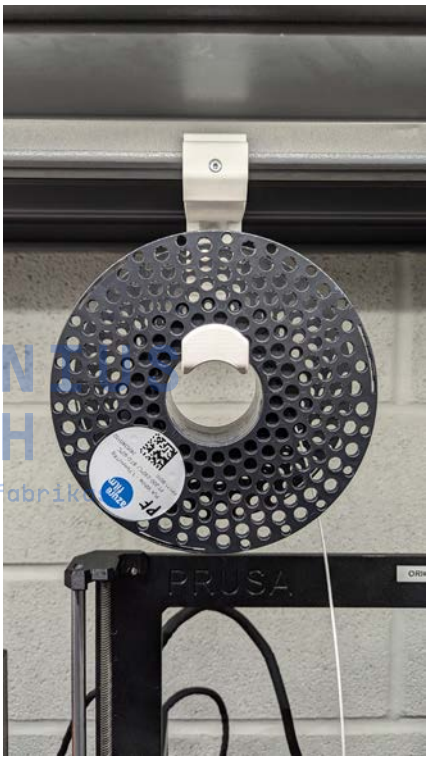
Stay at least for the first layers to insure that everything goes correctly. The first layers are the most important ones.

5

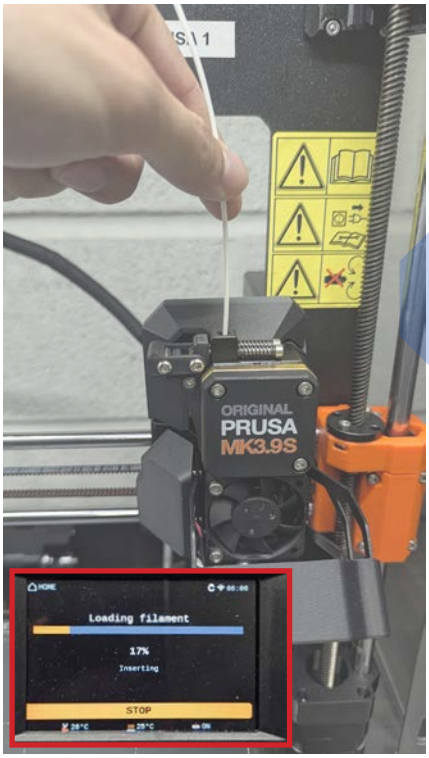
When the print is done you can remove the bed from the printer. Becareful, the bed can be hot. Like the step 1. , clean the bed in the same way.



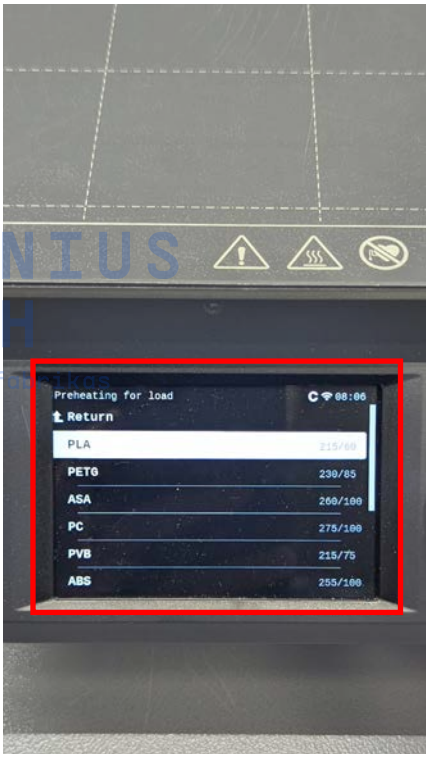
1. Clean the bed using tissue and alcohol solution to remove any residue. The supplies can be found next to the sink.



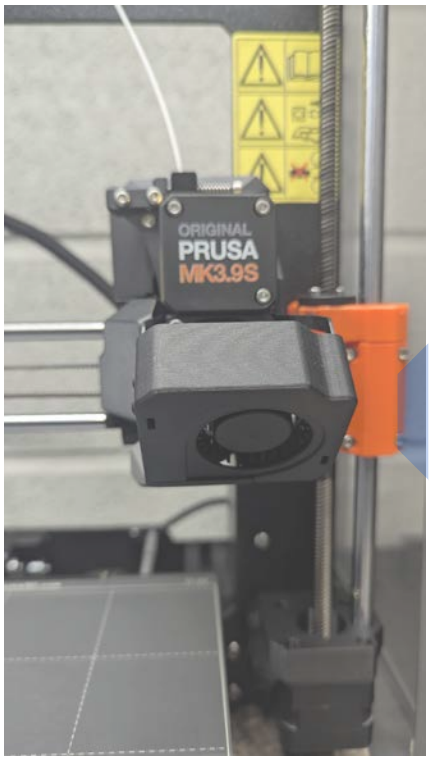
2. Try to always put your filament with the label facing you so you know what type of filament you are using.



3. When inserting the filament, the printer will automatically load it in the nozzle.



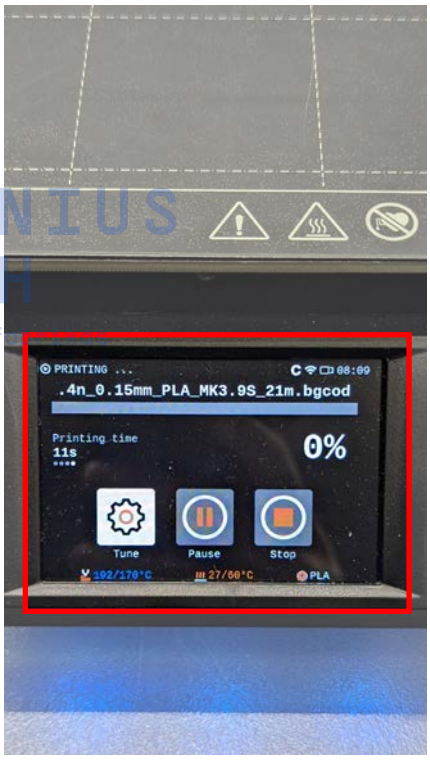
4. When the nozzle has fully loaded the filament, a screen will show up. Choose the filament type to load the right preset.



5. The nozzle will then star to purge it-self by extruding some material. Caution, the nozzle is very hot.

6. When the loading is complete, check the nozzle. The filament color should be the same as on the roll. If not, purge one more time.

7. You can now insert the flash drive. Your file should appear automatically.

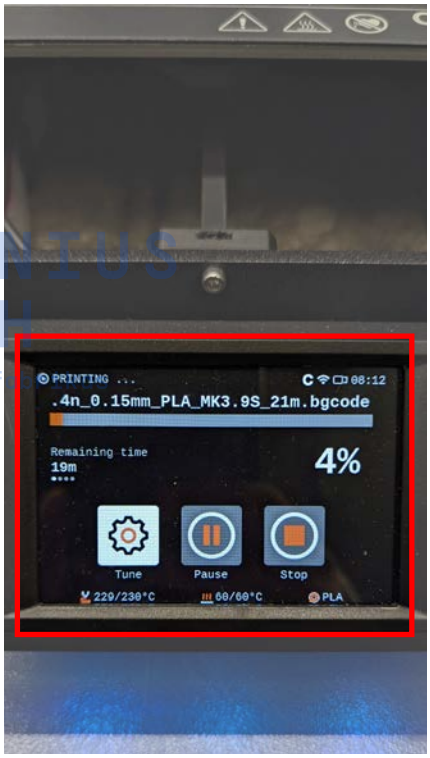


8. Press “print” to start the printing. The file is loaded and the printer will start to pre-heat the bed and nozzle.

9. The printer will do som calibrations. No need to manipulate the printer from now on.

10. When the calibration is done, it will show you where it will be printing.

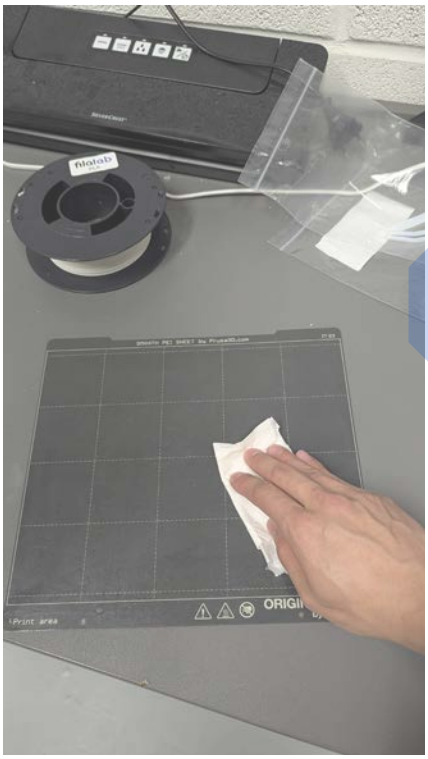
11. The print may now start. It will do a test on the side to extrude the nozzle one last time.



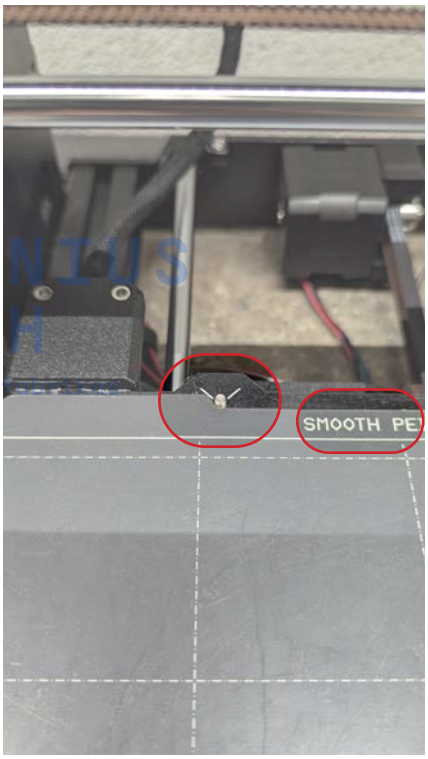
12. On the screen you can see the remaining time and the percentage of progression.

13. The print is now done. It's time to remove the bed. Caution, it might be hot.

14. Put your thumbs ont the indicated spot and lift the bed. You can gently bend the bed to remove the print.

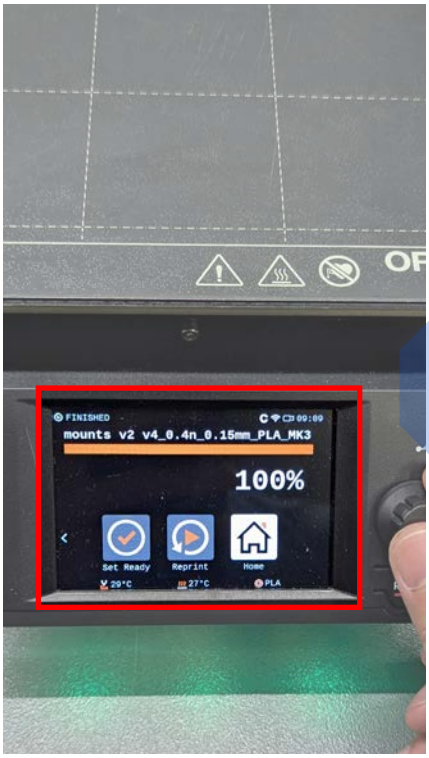


15. Like in step 1. , clean the bed after the print.



16. To put back the bed, align the notch to the spikes in the back.

Tips: the text on the bed should be facing you.



17. To remove your filament click one time on the turning knob and select “home”.

18. Then turn the knob and select “filament”.

19. Then select “remove filament”.

20. The nozzle will unload the filament. Wait for the printer to confirm that the process is done before removing it.

ANNEX

VILNIUS TECH
LinkMenų fabrikas

PLA or PETG?

Choosing the right filament

When booking a 3D printer, it's important to choose the right filament for your project. The only two materials we use in the lab are PLA and PETG.

PLA (Polylactic Acid)

Ease of Use: PLA is the easiest filament to print with. It requires a lower printing temperature (around 190–210 °C), sticks well to the print bed, and doesn't warp much — which makes it great for beginners.

Finish: It produces clean, smooth prints with a matte or slightly glossy surface.

Strength & Durability: PLA is rigid but brittle. It can break or snap under stress and doesn't handle heat well (starts deforming around 60 °C).

Best for: Prototypes, visual models, low-stress parts, or decorative objects.
What not do: Do not use for outdoor or mechanical projects. Do not sand. Due to the low melting point, it will deform.

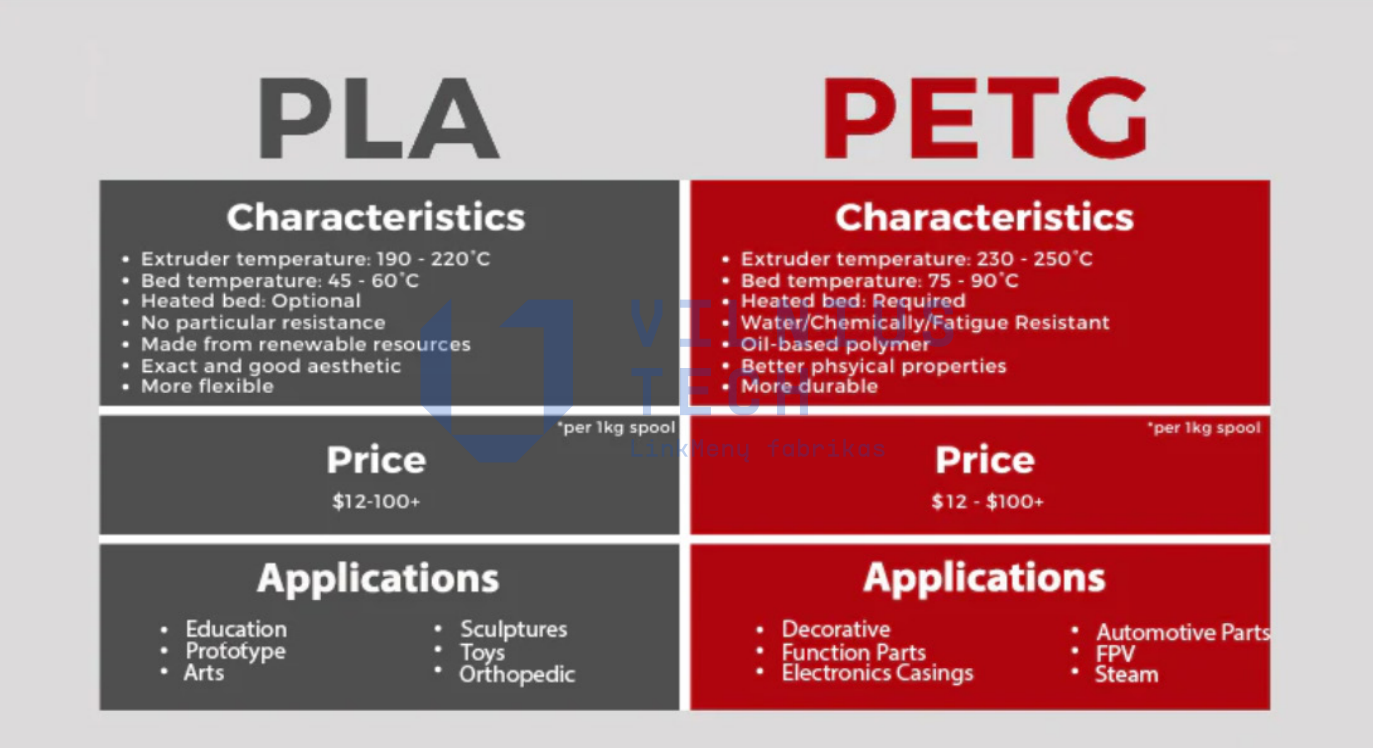
PETG (Polyethylene Terephthalate Glycol)

Ease of Use: PETG is slightly trickier to print than PLA but still accessible. It prints at higher temperatures (220–250 °C) and may require fine-tuning to avoid stringing.

Finish: PETG has a glossy finish and a slightly rubbery feel. It's more flexible and less brittle than PLA.

Strength & Durability: PETG is tough, impact-resistant, and can handle moderate heat (~85 °C) and some outdoor exposure.

Best for: Functional parts, mechanical components, containers, or anything that needs strength, flexibility, and durability.
What not do: Don't choose it for prints requiring strong rigidity and for small parts. Also, if not set correctly, PETG tends to blob (appearance of small, unwanted blobs of filament on the surface of the print).



Infills

What are they?

Infill refers to the internal structure of a 3D printed object. It isn't visible from the outside, but it plays a key role in determining the print's strength, weight, material usage, and print time. When slicing your model, you can choose different infill types (patterns), and each one serves a different purpose.

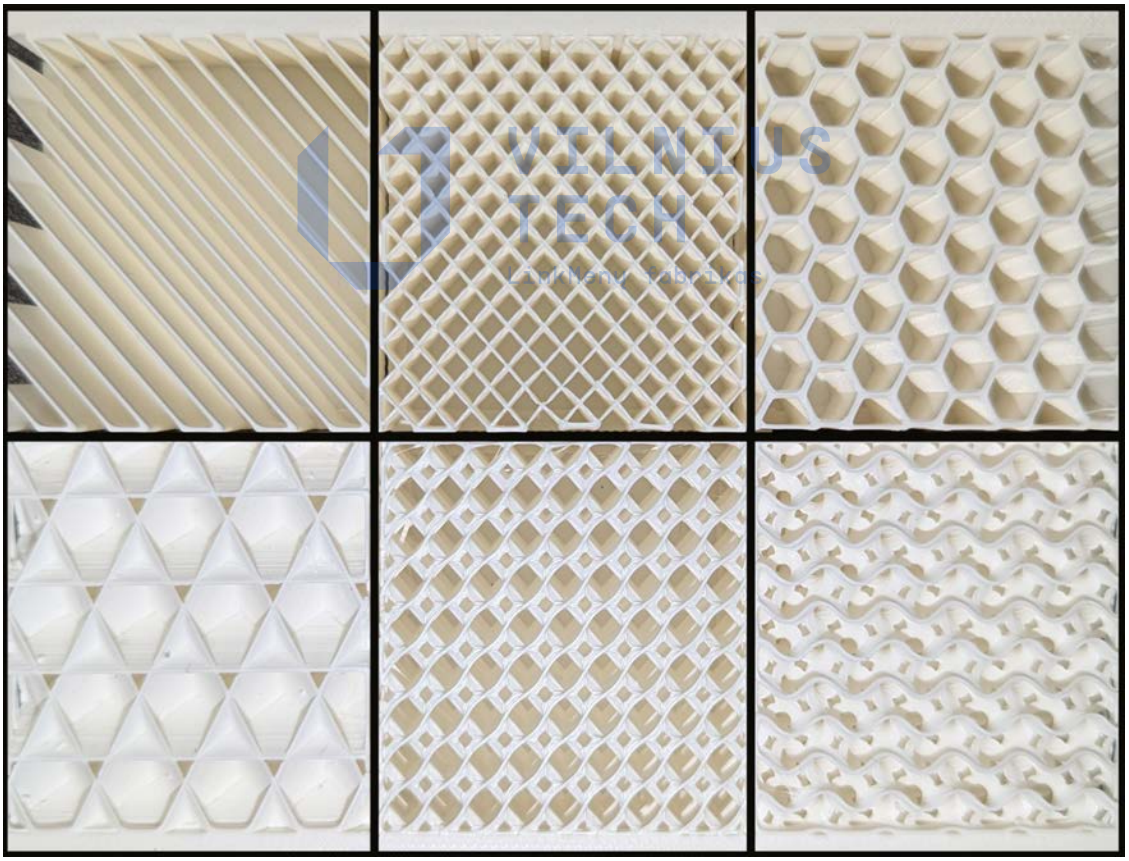
Why are they important?

Strength: Denser or more supportive infill patterns make the part stronger.

Flexibility: Some patterns allow more movement or flexibility in the print.

Weight: Less infill means a lighter part, which can be useful for prototypes or non-load-bearing pieces.

Print Time & Material: Higher infill = more filament + longer print time.



“Aligned”

Description: Straight parallel lines, stacked layer by layer without rotation.

Why use it: Fastest to print, uses minimal material.

Best for: Visual prototypes, early drafts, and low-stress objects.

Suited for: Architecture students needing quick massing models or volumetric studies.

“Rectilinear”

Description: Alternating horizontal and vertical lines on each layer (like a grid).

Why use it: Good balance between strength and speed.

Best for: Simple, general-purpose prints.

Suited for: Everyone, especially useful for basic prototyping across disciplines.

“Honeycomb”

Description: Hexagonal pattern similar to a beehive.

Why use it: Strong and rigid, distributes loads well.

Best for: Functional or mechanical parts that need strength without being fully solid.

Suited for: Product design students printing mechanical or structural parts.

“Cubic”

Description: 3D cube-like pattern, strong in all directions.

Why use it: Excellent for prints under multi-directional stress.

Best for: Parts that must resist torsion or compression.

Suited for: Product design students making functional components or structural tests.

“Cross hatch”

Description: Two sets of diagonal lines crossing at 45°, creating a mesh-like pattern.

Why use it: Offers good surface support, slightly more flexible than grid.

Best for: Medium-strength prints with faster build times.

Suited for: Architecture students or general users wanting a balance of strength and speed.

“Gyroid”

Description: Smooth, organic wave-like 3D structure.

Why use it: Great balance of strength, flexibility, and material efficiency.

Best for: Durable prints needing strength without heaviness.

Suited for: Product design students working on enclosures, casings, or parts under moderate load.

What about density?

Density?

Infill density controls how much of the inside of your object is filled. It's measured as a percentage (%). The higher the density, the stronger and heavier the print but it also takes longer and uses more filament.

Why is it important?

If the infill is too low, your part may be too weak and break under stress.
If it's too high, your print will take longer, use more filament, and may be heavier than needed which can lead to material waste or even failed prints due to overheating or warping.

A well-adjusted infill means:
Efficient use of time and material, reliable strength where needed, better print success rate and a part that performs as expected.

For each project, ask yourself: Does this part need to be strong, flexible, fast to print, or lightweight? Then adjust your infill accordingly. If you have any doupts or question, feel free to ask one of the staff member.

5% 10% 20%

This manual was made by Dario Arrault,
at VILNIUS TECH „LinkMenų fabrikas“,
for internal VILNIUS TECH use and reference.

Date: 2025 06 06 | Version: 1.0