3D Printing

Slicing

In order to print your model, you must follow these steps:

Slice the model using PrusaSlicer in FabLab.

- Import your 3D model in .stl form.

- Pick the filament profile according to your chosen filament.

- It is good to use Input Shaper because it cuts the printing time to half.

- Starting with the Input Shaper, there will be two profiles (print settings) for the most common layers heights:

  o **Structural** - settings are focused on quality, precision and structural integrity while retaining reasonably high printing speed (still much faster than pre-input-shaper era).

  o **Speed** - short printing times are the top priority here. This profile pushes the printer closer to its limits while keeping good quality and accuracy.

- Export the G-code onto the USB stick.

Tips for 3D models and slicing:

- **Layer height** has the largest effect on details and printing time.

- **Infill** value determines how much material is extruded inside the solid parts of the print. Infill is required to print roofs. 100% infill is solid.

- If you need a stronger part, instead of increasing infill, increase **perimeters** in print settings.

- Place large flat surfaces onto the build plate.

- Try to avoid overhangs.

- Avoid printing round shapes vertically, try to place the model so that round shapes are horizontal.

- Favor chamfers over fillets.

Supports:

- Use supports when the part has large overhangs and bridges.

    - PETG and ASA prints may require supports on most bridges.

- Slopes greater than 45 degrees usually require supports.

- Try to place the part so that all supports can be generated from the build plate.

    - Generating the supports only from build plate saves time and material.

    - If not possible, choose support from everywhere.

Brim and Raft:

- Brim is required if the contact surface between the build plate and the part is small.

    - Especially when printing PLA on the textured metal sheet.

- If Brim is not enough to keep the part attached to the plate, try using raft or both.

Note: Do <u>not</u> print PETG or ASA on smooth sheets.

Preparing the printer

- Choose filament for printing

- Wipe the print bed with Isopropyl alcohol.

- Insert the USB containing the <u>sliced</u> model.

- Check below how to unload and load filament if you need to change the filament

- Select "Print" and select your sliced file.

- The printer will now heat and calibrate itself before printing.

    - Watch the printer as it prints the first layer to ensure the print sticks to the bed!

To unload the filament:

- Press the knob to bring up the menu.

FabLab Tampere

- Select "Filament" and select then "Unload Filament".

- Check the type of the loaded filament on the side of the filament spool and choose the unloading temperature accordingly.

- Follow the instructions on the printer's screen when to pull the filament out.

- **After unloading, wind the filament onto the spool and pull the end of the filament through a hole on the side of the filament spool!**

To load the filament:

- Pull the filament out from the hole on the side of the spool.

- Cut the end of the filament if there is a blob.

- Press "Filament" and select "Load Filament".

- Push the filament into the hole on top of the extruder until the printer pulls it through.

- Follow the printer's instructions on it's screen to complete the filament change.

Note: The printer is operated by pressing and rotating the knob:

- Rotate to navigate.

- Press to select.

- The "X" button is emergency stop.

3D printing materials

PLA - Suitable for prototyping, decorations and toys

- **Pros:**

  - Easy to print

  - Non-toxic

  - Prints fine bridges

  - Detailed small prints

- **Cons:**

FabLab Tampere

- Brittle

- Low temperature resistance (60C)

- Difficult post-processing

- Not for outdoors use

PETG - Suitable for functional parts and toys

- **Pros:**

  - Easy to print

  - Non-toxic

  - High temperature resistance

  - Strong layer adhesion

  - Prints are watertight

  - Suitable for outdoor use

- **Cons:**

  - Poor details

  - Poor bridges

  - Possible stringing

ASA - Suitable for functional parts

- **Pros:**

  - Durable

  - High temperature resistance

  - Suitable for outdoor use

  - UV stable

  - Can be smoothed with acetone vapour

- **Cons:**

  - Difficult to print

  - Poor details

  - Larger prints warp easily

For more detailed info: https://help.prusa3d.com/en/materials

FabLab Tampere

Common failures

Problem: Print doesn't stick to bed

Result: Printer creates either a random pile of spaghetti or a blob of plastic forms
                 around the extruder.

How to avoid?   Wipe the bed with isopropyl alcohol and check your slicer settings and part's
                 orientation.

Problem: Print warps during printing

Result: The print will bend and the shape will be inaccurate.

How to avoid?   Wipe the bed with isopropyl alcohol. Use brim.

Problem: Printer prints outside the build plate

Result: Models will most likely fail.

How to avoid?   Choose the right printer in slicer.

Problem: Print has stringing or fails midway

Result: Model looks nasty or fails completely.

How to avoid?   Ensure you have chosen the right filament in the slicer.

Problem: Printer does not extrude anything

Result: No print.

How to avoid?   Cut the end of the filament. Insert the filament all the way in so that the printer
                 pulls it in. Ensure you have chosen the right filament and printer in the slicer. If
                 these didn't help, the nozzle may be clogged and you should ask the staff for
                 help.

FabLab Tampere

Contact details:

Tampere University, FabLab Tampere, Korkeakoulunkatu 3, 33720 Tampere.

tuni.fi/fablabtampere (FI)