

# 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.
- Remember to choose the right printer!
- Export the G-code onto the SD card (MK3) or USB stick (MINI).

### 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.
  - Favour chamfers over fillets.

### Supports:

- Use supports when the part has large overhangs and bridges.
  - PETG and ABS 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 not print PETG or ABS on smooth sheets.

## **Preparing the printer**

- Choose filament for printing.
- Wipe the print bed with Isopropyl alcohol.
- Insert the SD card containing the sliced model.
- Find and select "Print from SD" 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.
- Find "Unload Filament" and select it.
- 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.
- Find "load filament" and select it.
- 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:**
  - 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

## **ABS/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
  - Toxic printing fumes (ABS)

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

## 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.

**Contact details:**

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

[tuni.fi/fablab tampere](https://tuni.fi/fablab tampere) (FI)