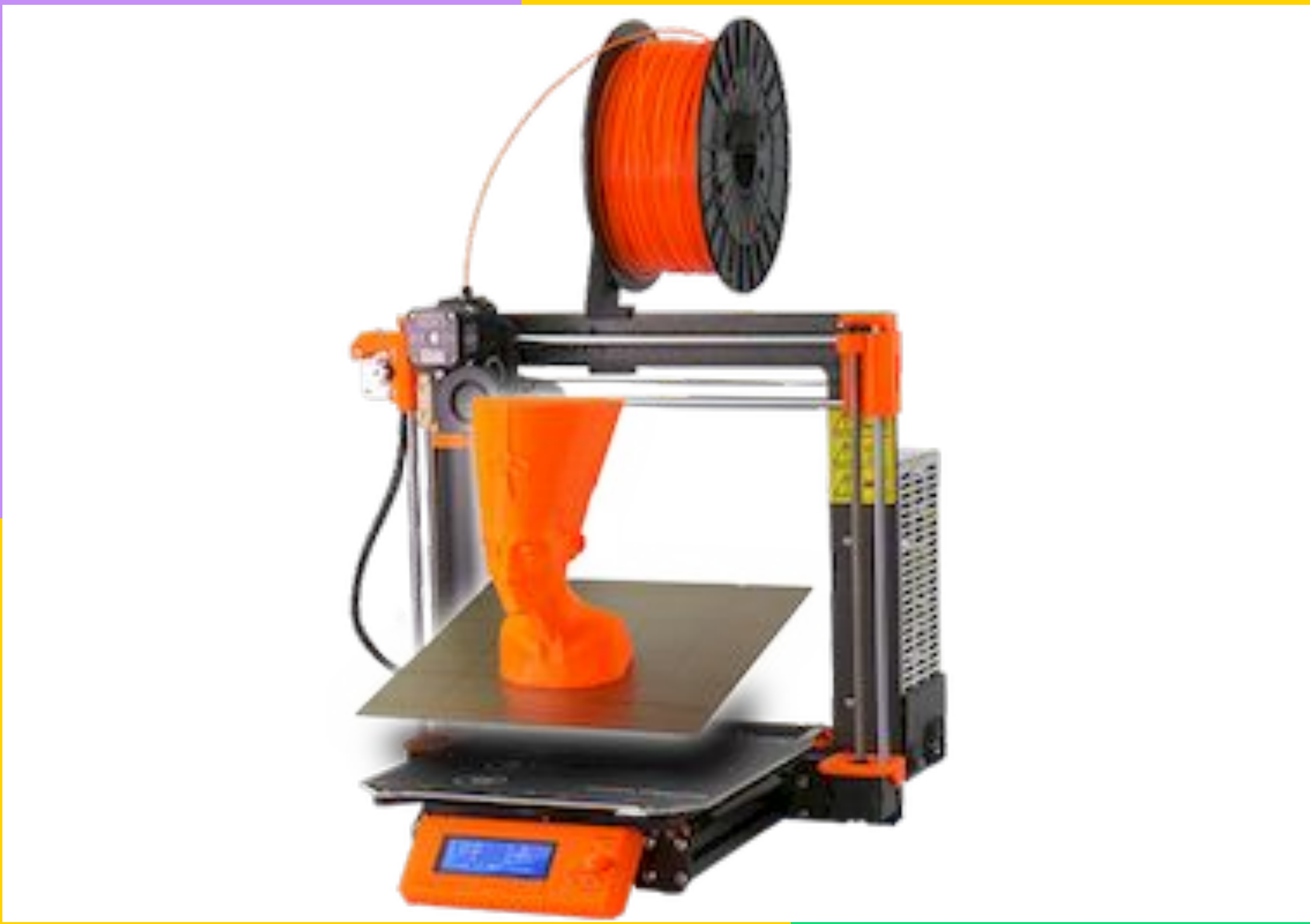
ETH STUDENT PROJECT HOUSE

PRUSA MK3/MK3s 3D PRINTER

FDM style printer
Build volume 25 x 21 x 20 cm
Up to 0.05 mm layer height
Materials: PLA, PETG



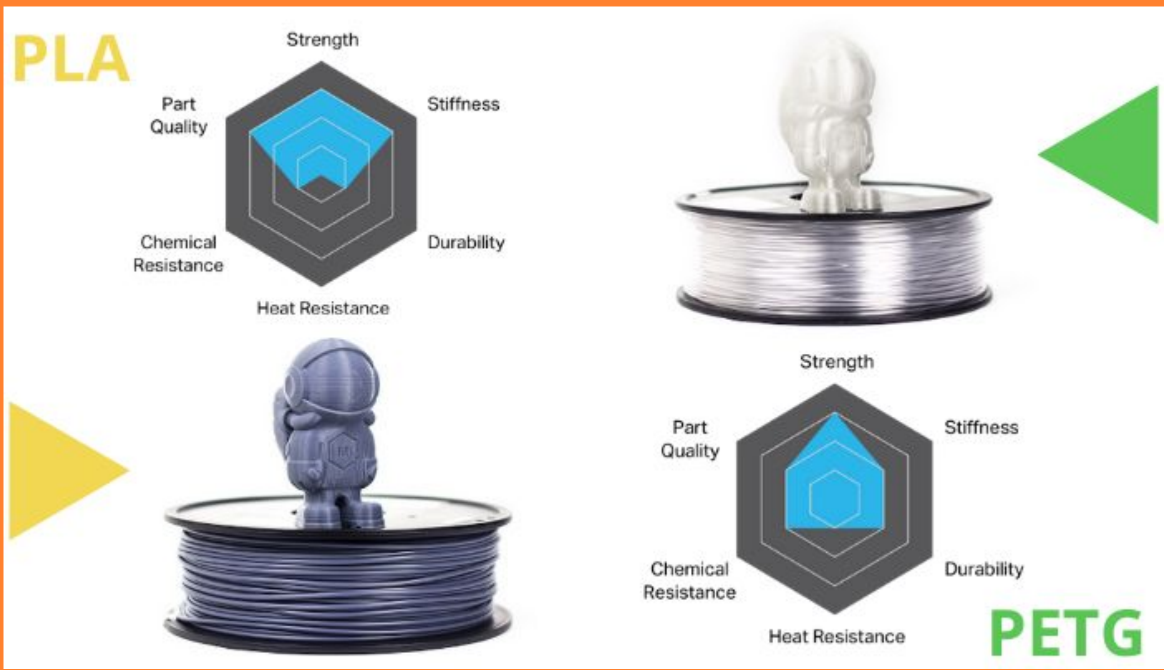
MATERIALS PROPERTIES

PLA

Biopolymer
Hotend: 180 - 210 °C
Heatbed: 50 - 70 °C
Small amount of warping
Easy to print
Sensitive to temperature

PETG

Modification of PET
Hotend: 220 - 240 °C
Heatbed: 80 - 100 °C
A bit of warping
Fairly easy to print
Not sensitive to temperature



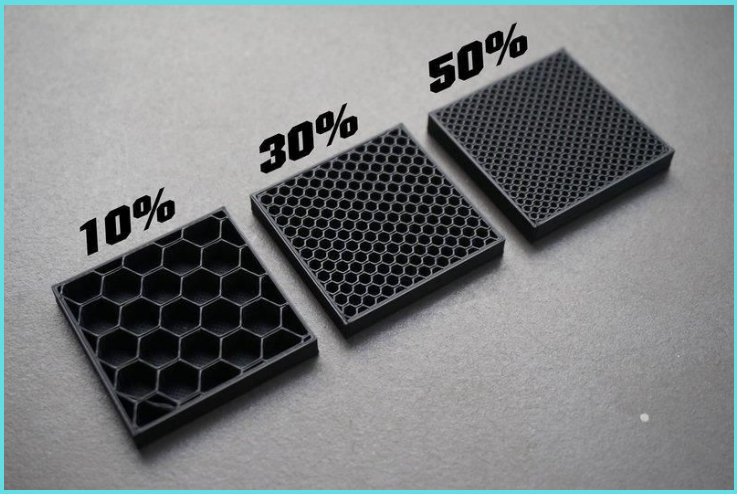
The main differences between the two are their properties, applications, and material costs. **PETG is stronger and more resilient than PLA**. PLA, on the other hand, is widely used as FDM/FFF filaments because of its better melt and cooling properties. In terms of cost, PETG is more expensive than PLA.

ETH STUDENT PROJECT HOUSE

TERMINOLOGY

Infill

- Parts are not solid
- Most of the time 15% is enough



Supports

- Machine can’t print mid-air
- Supports the print above
- Threshold value 45°

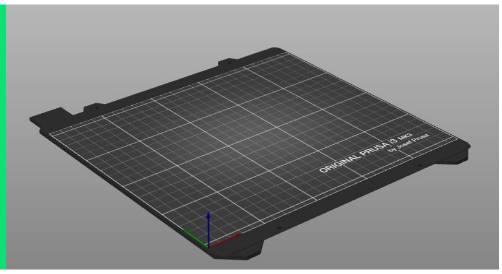
Brim

- Needed if only a small area of contact to the bed is available to increase contact area
- Prevent lifting of the part from the print bed

PRUSA SLICER

Base view

- Open PrusaSlicer
- Make shure the simple mode is activated
- Drag and drop your .stl file on the print bed



TRANSFORMATIONS

Move

Use this to move your part around the build plate

Scale

This tool can be used to make a part bigger/smaller

Rotate

With this tool you can rotate your part around x,y and z axis

Place on face

Select a face and the software will place it on the build plate

Cut

This tool lets you cut your part into two

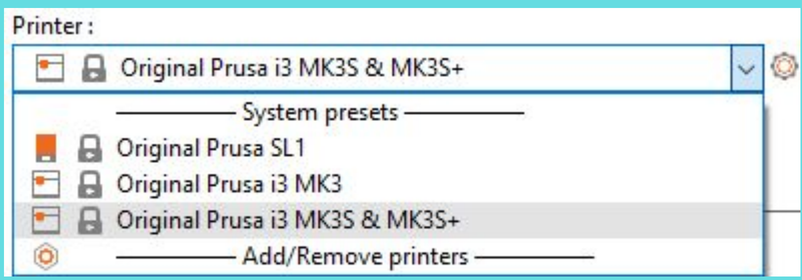
PRUSA SLICER

Add More

- Use the buttons on the top to add, copy, paste and arrange parts

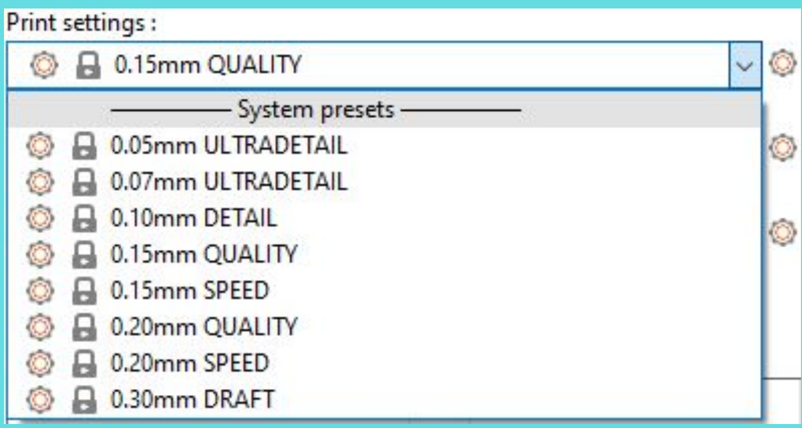
Machine

- Make sure to select the correct machine.
- The machine type can be found on the label attached to the machine itself.



Layer Height

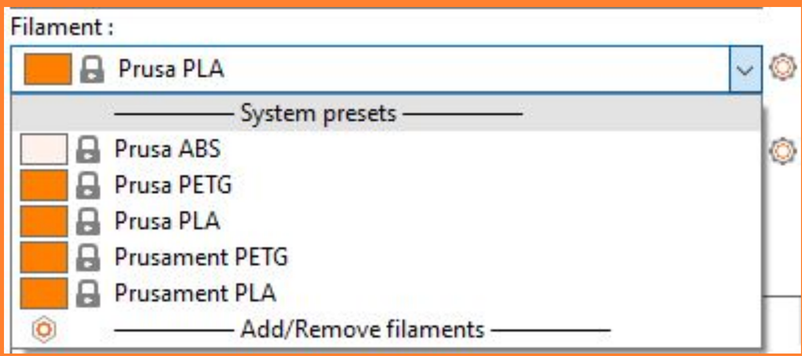
- Select the layer height to decide the print quality.
- Smaller layer heights take longer to print but with better quality.
- Check example frogs if in doubt.



PRUSA SLICER

Material

- Now you can select your print material.
- The color does not make a difference.
- Please use Prusa PLA/ABS/PET.

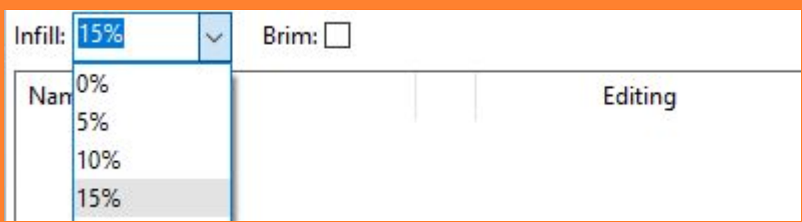


Supports

- Set your desired support setting.
- For beginners only use None/Everywhere.

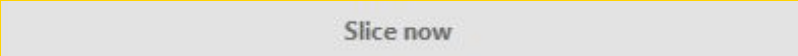
Infill

- Set your infill percentage.
- More infill means more weight, longer print times and higher costs.
- For most parts 15% is fine.
- If the area of contact with the bed is really small remember to add a Brim.



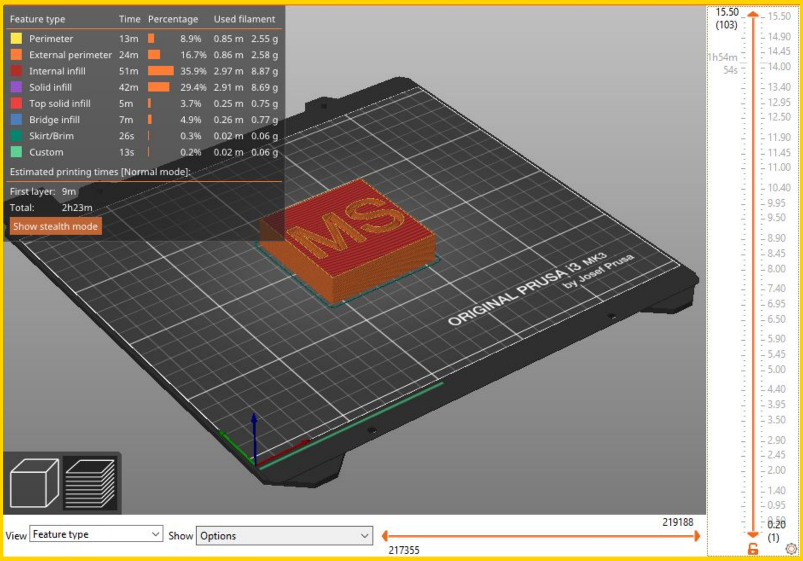
Slice

- Use the Slice now button to prepare your part for printing.



Preview

- Use the slider on the right to go through the different layers.
- Check that at no point something is appearing out of nothing.



Print Info

- On the lower right corner you can see some information about your print.
- Make sure that there is enough filament left for your print.
- Use the Export G-code button to export the code and save it on the SD card.
- Note: The costs are not correct.

Sliced Info

Used Filament (g) (including spool)	24.33 (254.33)
Used Filament (m)	8.16
Used Filament (mm³)	19619.24
Cost	0.68
Estimated printing time:	
- normal mode	2h23m
- stealth mode	2h24m

AT THE MACHINE

Step 1

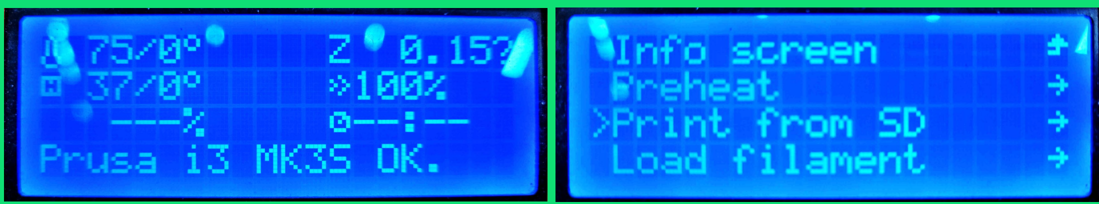
- Turn on the machine on one of the terminals (select machine → use machine).

Step 2

- Make sure the correct filament is loaded and that it is enough for your print.

Step 3

- Press the knob once on the printer and navigate to [Print from SD].

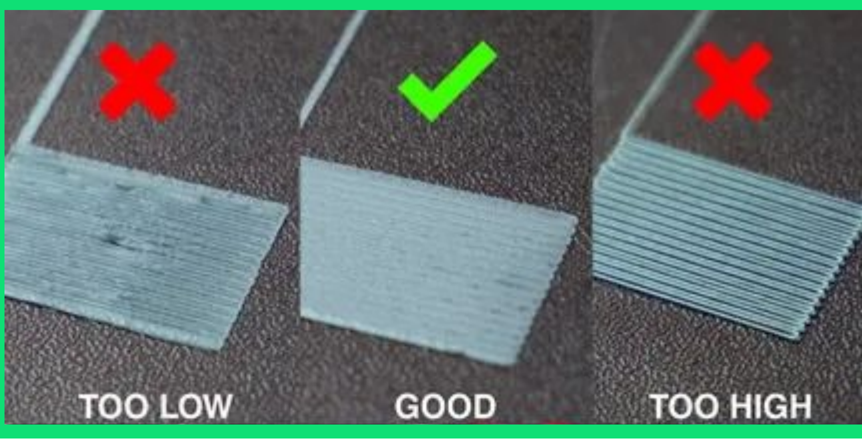


Step 4

- Select your part (usually the first in the list) and press the knob to start the print.

Step 5

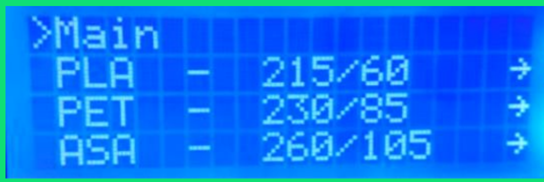
- Check the height of the first layer.
- If is too high/low please contact a Makerspace Manager



CHANGING THE FILAMENT

Step 1

- Press the knob on the printer and navigate to [Preheat]
- Select the temperature according to the filament with the highest temperature.



Step 2

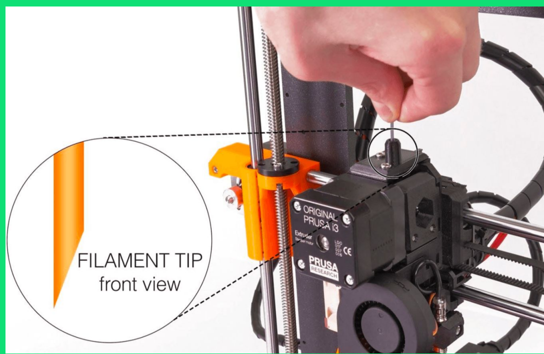
- As soon as the nozzle is up to temperature select [Unload Filament] on the printer menu.

Step 3

- Press the knob to start unloading the filament, hold on the filament and firmly pull it until is all the way out of the printer.
- Secure the end of the filament on the filament spool.

Step 4

- Cut the new filament at an angle and insert it into the extruder.
- Select the [Load Filament] option on the printer menu and press the knob.
- The filament should start to be pulled in.



Step 5

- When asked if filament is of the correct color. If you see that the filament is not clearly of the correct color select [No], more filament will be extruded. When you have the correct color extruding select [Yes] and you are good to go.