Prusa I3 FDM 3D Printer

- FDM Style Printer
- Build volume 25 x 21 x 20 cm
- Up to 0,05 mm layer height
- Materials: PLA, PETG
- Other materials on request

PLA

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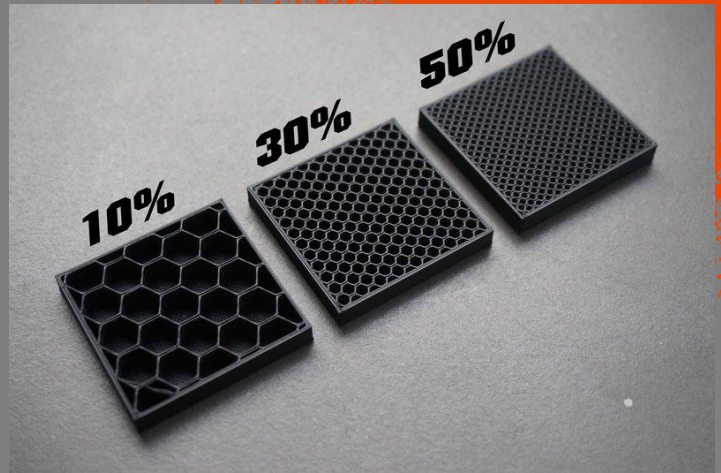*Sensitive to temperature*

PETG

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

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*
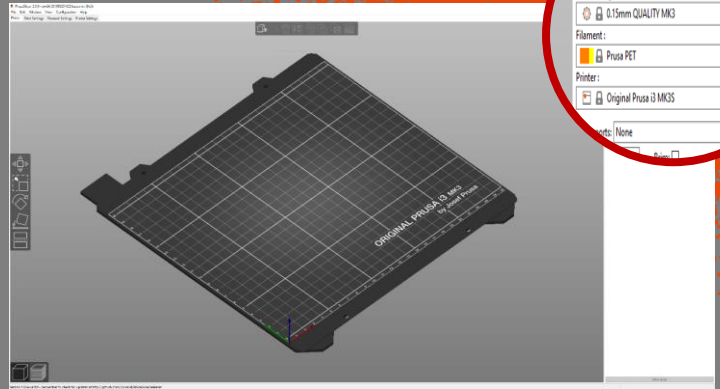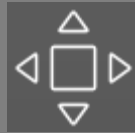- *Prevents lifting of the part from the print bed*

PrusaSlicer

Base view

- *Open PrusaSlicer*
- *Make sure the Simple mode is activated*
- *Drag and drop your .stl on the print bed*

Transformations

Move

Use this to move your part around on 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 the x,y and z axis

Place on face

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

Cut

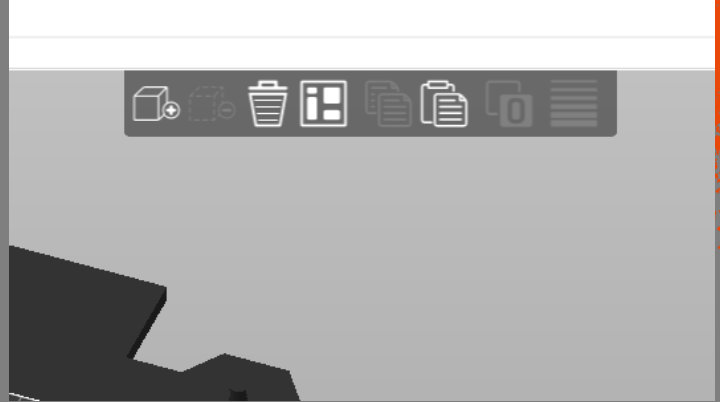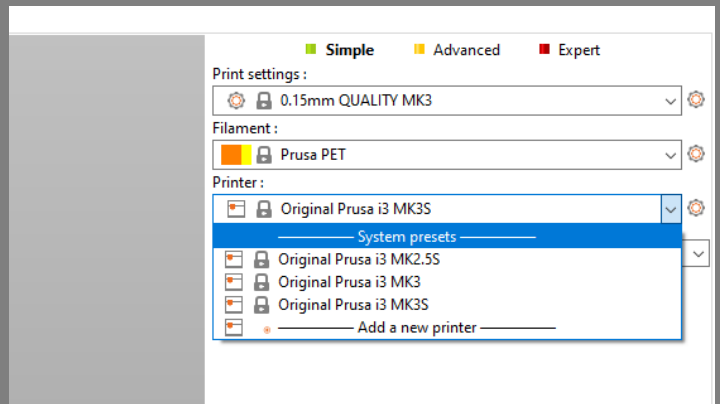*This tool lets you cut your part into two*

PrusaSlicer

Add more

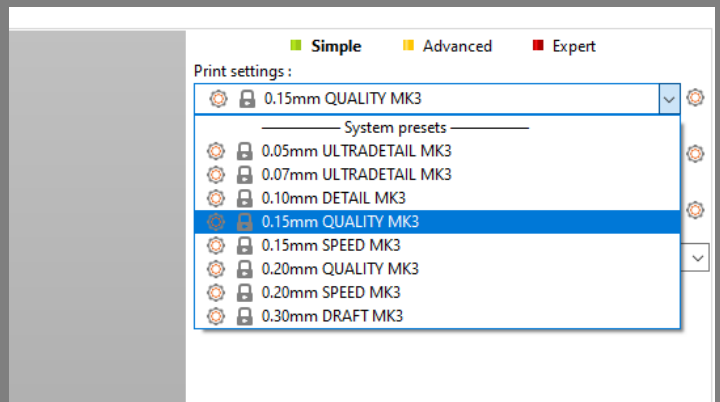- *Use the buttons on the top to add, copy, paste and auto arrange parts*

Machine

- *First you have to select the correct machine*
- *The machine type can be found on the red label attached to the machine*

Print settings :
0.15mm QUALITY MK3

Filament :
Prusa PET

Printer :
Original Prusa i3 MK3S
— System presets —
Original Prusa i3 MK2.5S
Original Prusa i3 MK3
Original Prusa i3 MK3S
— Add a new printer —

Layer Height

- *Next select the desired layer height/print quality*
- *Smaller layer heights take longer to print*

Print settings :
0.15mm QUALITY MK3
— System presets —
0.05mm ULTRADETAIL MK3
0.07mm ULTRADETAIL MK3
0.10mm DETAIL MK3
0.15mm QUALITY MK3
0.15mm SPEED MK3
0.20mm QUALITY MK3
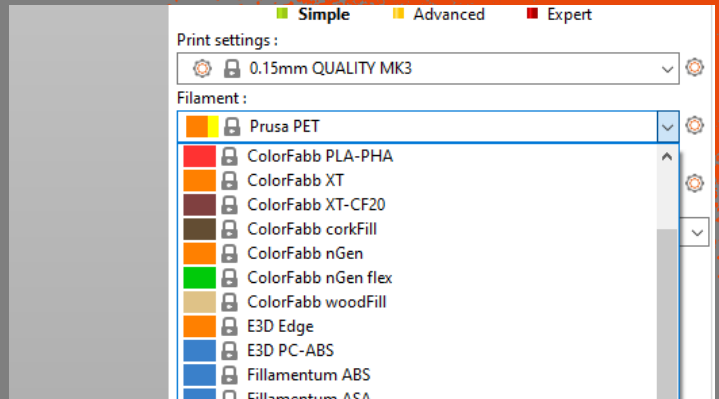0.20mm SPEED MK3
0.30mm DRAFT MK3
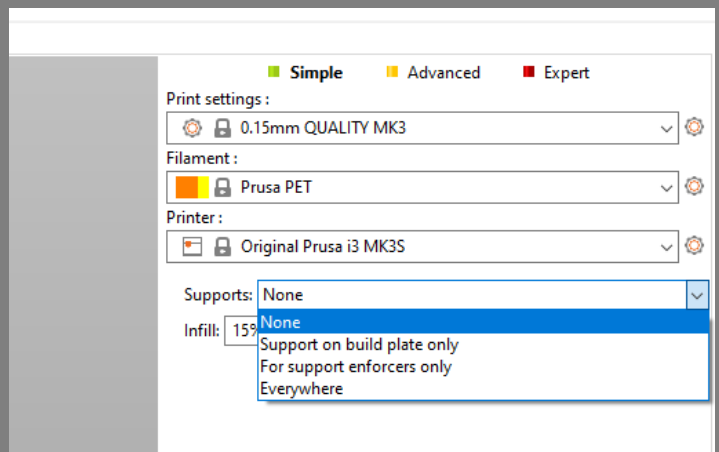
PrusaSlicer

Material

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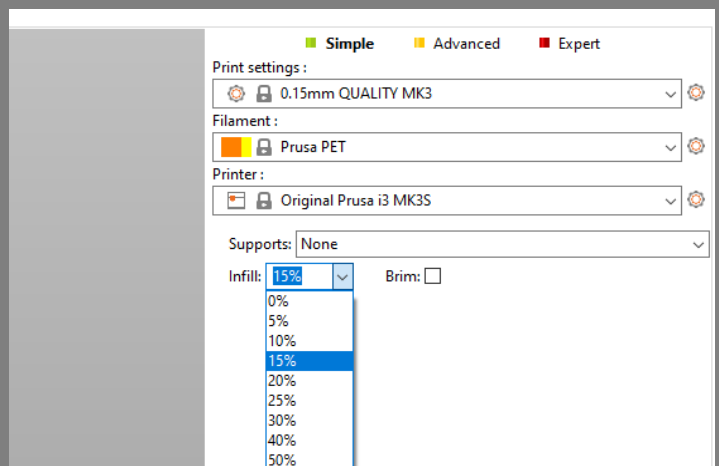- *Please use Prusa PLA/ABS/PET*

Supports

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

Infill

- *Set your infill percentage*
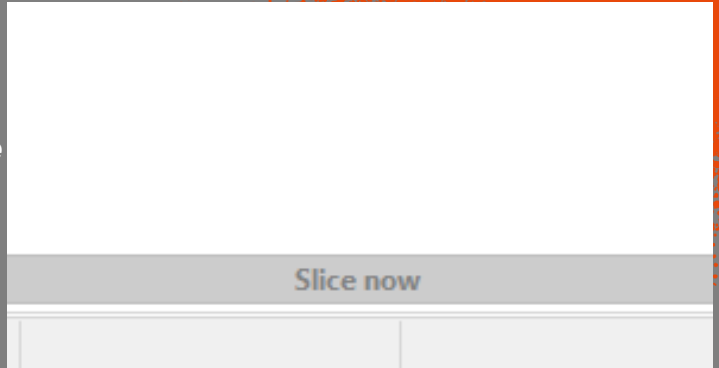- *More infill means more weight, longer print times and higher costs*
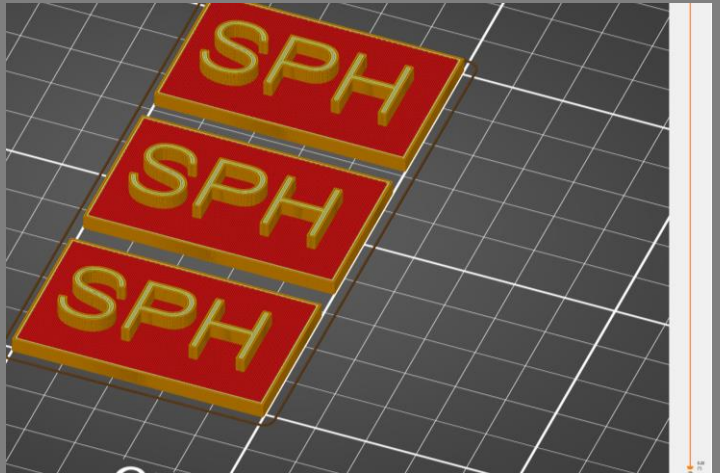- *For most parts 15% is fine*

PrusaSlicer

Slice

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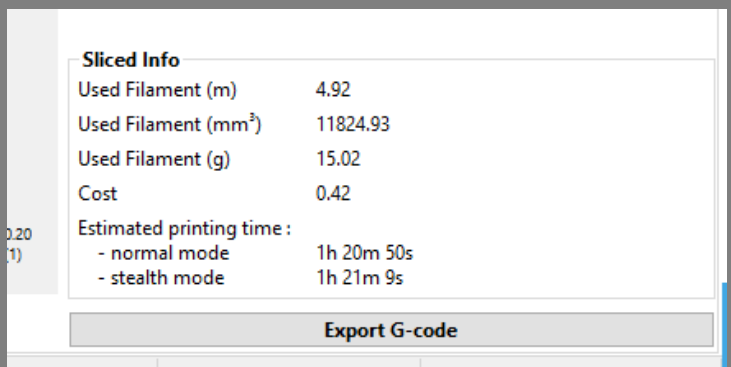
Slice now

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 (m)	4.92
Used Filament (mm^3)	11824.93
Used Filament (g)	15.02
Cost	0.42
Estimated printing time :	
- normal mode	1h 20m 50s
- stealth mode	1h 21m 9s

Export G-code

At the Machine

Step 1
- *Turn on the Machine at one of the terminals*

Step 2
- *Make sure the correct and enough filament is loaded*

Step 3
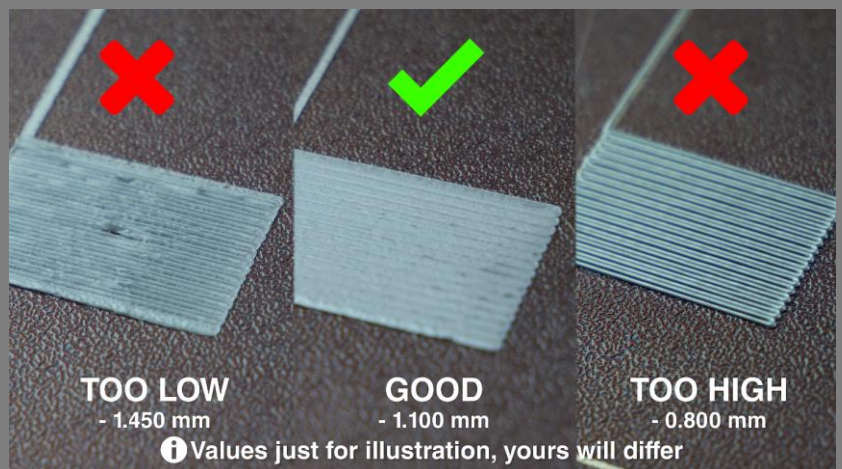- *Press the encoder wheel on the printer once – navigate to "Print from SD"*

Step 4
- *Choose your part (usually the first one in the list) and press the wheel once more to start the print*

Step 5
- *Check the height of your first layer*
- *When the layer is too high/low please contact a Makerspace Manager*

TOO LOW
- 1.450 mm

GOOD
- 1.100 mm

TOO HIGH
- 0.800 mm

Values just for illustration, yours will differ

ETH STUDENT PROJECT HOUSE

How to change the filament

Step 1

- *Press the encoder wheel on the printer once – navigate to "Preheat"*
- *Choose the temperature according to the filament with the higher melting temperature*

Step 2

- *As soon as the Nozzle is up to temperature, hold on to the filament and press the encoder wheel on the printer once – navigate to "Unload filament"*

Step 3

- *Press the encoder wheel again to select "Unload filament", now pull firmly on the filament until it is all the way out of the printer*
- *Secure the end of the filament on the filament roll!*

Step 4

- *Cut the new filament at an angle and insert it into the extruder*
- *Press the encoder wheel on the printer once – navigate to "Load filament" and press the wheel again. Now you should feel how the filament gets pulled in*

Step 5

- *If the color of the filament is clear, choose "Yes" in the upcoming prompt if not, choose "No", more filament gets extruded. The printer will ask you again until you choose "Yes"*