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, ABS
- Other materials on request

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

ABS
LEGO
Hotend: 215 – 250 °C
Heatbed: 100 – 120 °C
Tons of warping
Hard to print
Not 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 midair
• 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
• Never print ABS without a Brim
Open PrusaSlicer

Make sure the Simple mode is activated

Drag and drop your .stl on the printbed

**Baseview**

**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 plate

**Cut**
This tool lets you cut your part into two
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

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

**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
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

Values just for illustration, yours will differ