Printing guide (FDM)

The following tips are intended for **PLA** with a **0.4 mm** nozzle and are based on Prusasclicer. Every other slicer has similar names for the settings but keep in mind that there may be slight differences.

Key settings:

Layer height: at least 0.2 mm
Perimeters: at least 3
Infill: at least 20%
Bottom layers: at least 4
Top layers: at least 5
Support: disabled
Bridging: enabled
Lift Z: 0.5 mm

Strong first layer adhesion is crucial for a successful print!
When in doubt helper discs are always a good idea!

BRIDGING WILL TAKE CARE OF THE UNSUPPORTED AREA