

## 3D printing guide

# Fillamentum ABS Extrafill

(acrylonitrile butadiene styrene )

Printing temperature: 220–250 °C

Heated bed temperature: 80–105 °C

Speed: 30–50 mm/s

Part cooling fan: 0–20 % (5 % is good start)

Heated bed surface: PEI, mirror/glass

Adhesive: Dimafix, PVA glue, 3Dlac, Magigoo

Raft/skirt/brim: Brim 5–10 mm

Heated chamber/enclosure: recommended

**Cooling** - It is not recommended to use more than 20 % of fan speed, as too much cooling could lead to improper layer bonding and warping. We recommend using no cooling for standard objects, which leads to stronger parts. For bridges and big overhangs, it is possible to go up to 30–40 % for the desired layer.

**Adhesion** - Brim 5–10 mm. The best adhesion was achieved with Dimafix on clean glass or mirror. On the PEI we recommend using wider brim for large objects.

**Printing speed** – ABS is better to print little bit slower. Material has more time to bond properly preventing the layer separation and cracks between layers.

**Overhangs** - for the overhangs higher than 45° we recommend using lower layer heights and also enough time to cool, otherwise, overhangs could slightly warp.