

# How I 3d Print - Slicer settings

this is ment to be more some kind of a collection of settings that have worked for me in the past. I use this because i don't use my 3d printer on a daily basis but instead have rather long pauses between usage periods. So I tend to forget what worked the last time and then have to start from scratch again:

All the settings below are for printing on a Tevo Tarantula 3D Printer using Simplify 3d as my slicer software. I will note the changes from the defaults in Simplify 3d.

## Printbed setup

I have put a 21x21cm sized piece of glass (actually cut out of some scrap of a spy mirror) on top of the heated bed and spray just a little bit of hairspray on top of it to make it matte, this seems to help adhesion well enough and can be cleaned off with some water.

## PLA

- Extruder - Retraction: 3mm (1 is default but leads to "hairy" prints)
- Support: 10% infill with 4mm offset worked fine for a 1.5x6.5cm area.
- Temperature: Heated bed: Layer 1-3 @ 50°C, 4+ @ 60°C, Extruder: 190°C
- Cooling - Fan Speeds: Layer 1-2 @ 0%, 3-9 @ 60%, 10+ @ 100% -> this was to prevent warping of the print, combined with the first cold then warmer bed.

## ABS / ASA

- The following settins where used for a Layer thickness of 0.4mm
- Extruder - Retraction: 3mm
- Support: 8% infill, dense infill @ 30%, Max overhang 45°, support pillar resuoltuin: 1mm,
- Temperature: Heated bed: 110°C, Extruder 260
- Cooling: Fan: off

From:

<http://wiki.psuter.ch/> - **pswiki**

Permanent link:

[http://wiki.psuter.ch/doku.php?id=how\\_i\\_3d\\_print\\_-\\_slicer\\_settings](http://wiki.psuter.ch/doku.php?id=how_i_3d_print_-_slicer_settings)

Last update: **23.02.2020 12:07**

