

# PRINTING GUIDE

FOR FDM PRINTERS - LAST UPDATED 9TH OF JANUARY 2021

Shown are three sample profiles, depending on the qualities you wish for in a print. You can easily pick and choose from these values to create a printing profile that suits your needs.

## FAST

- 0.3mm layer height
- 7% Infill
- 2 walls/shells
- 3 top/bottom layers

Default Printing Speed:  
80mm/s

Note: These settings are "pushed to their limits" to speed things up.

## STURDY

- 0.2mm layer height
- 20% Infill
- 3-4 walls/shells
- 5-6 top/bottom layers

Default Printing Speed:  
50mm/s

Note: Creates a very dense print that will both be strong and feel heavier.

## DETAILED

- 0.12mm layer height
- 10% Infill
- 2-3 walls/shells
- 4-5 top/bottom layers

Default Printing Speed:  
35mm/s

Note: Especially the printing speed and layer height will enhance the quality, but it takes a longer time.

2 walls is usually enough to make a print strong. However some types of filament can allow the infill to be seen through the finished print (especially with filaments that has a bit of translucency). If that's the case, it's better to print with 3-4 walls.

Printing speed: The maximum speed varies from printer to printer. 50mm/s is usually a safe standard to print at, but check your printer's model to find its maximum speed. Printing at a high speed may reduce the overall details of a print.



The amount of details on the Z-axis is determined by the layer height. The amount of details on the XY-axis is determined by the nozzle size.

The standard nozzle is 0.4mm. Try a 0.3mm or 0.2mm for even finer details, or try a 0.6 or 0.8mm nozzle for an exceptionally fast print.

If you experience very visible layer lines, it may be worth performing an extruder calibration. It helps assure that your printer extrudes the exact amount as it's told to.

Some mugs can have an area with a slanting angle (like the base of the Bard and the Paladin). Printing with 0.2mm can make the layer lines very visible. Some slicer programs are able to change the settings of a specific region. A 0.08mm layer height can be used to give it a smoother surface in those specific areas.

You can also find pre-made profiles around the internet. Fat Dragon Games are for example very popular for their Cura-profiles for Creality printers.



For slicer software we recommend either Cura or PrusaSlicer.

HAPPY PRINTING!

