

1- Zetamix General guidelines Tool steel H13

Zetamix filaments are on a fine powder (5-20 µm) and a thermoplastic binder system for the FDM process. Green parts need a binder removal before being sintered. In the debinding process the binder is thermally removed. These general guidelines are based on the processing of test parts with a wall thickness of 2 to 4 mm.

The recommendations are considered to work as a standard guideline and must be adapted to individual wall-thickness and part-design.

Typical material properties	
<i>Product</i>	Filament for FDM process
<i>Binder basis</i>	Polyolefinebased binder system
<i>Appearance</i>	Dark grey filament

Typical processing properties	
<i>Printing temperature</i>	180°C
<i>Plate temperature</i>	30 °C
<i>Nozzle size</i>	0.4 mm to 1 mm
<i>Layer thickness</i>	0.20 mm (possibility to go from 0.1 to 1mm, need to ajust printing speed)
<i>Printing speed</i>	20 mm/s (recommended)
<i>Debinding process</i>	Thermal debinding From 50 °C to 650°C at 10°C/h in an argon mixture with 2.5% hydrogen atmosphere
<i>Sintering process</i>	In a high temperature furnace Up to 1350°C at 50°C/h, holding time 2h in an argon mixture with 2.5% hydrogen atmosphere.
<i>Sintering shrinkage rate</i>	In xy direction: 16.5% (ie 10 mm → 8.5 mm after sintering) In z direction: 17.0% (ie 10 mm → 8.5 mm after sintering)
<i>Oversize factor</i>	In xy direction: 119.7% (ie 11.97 mm → 10 mm after sintering) In z direction: 120.5% (ie 12.05 mm → 10 mm after sintering)

Printing instructions:

The filament is softer than plastic filaments. Therefore, it is preferable to use a driving gear which is not too much aggressive and will not crush the filament (ideally a grooved driving gear). The filament can be grinded by the extruder, that's why it should be cleaned before a long print. To make sure that the printer is ready we recommend preheating the system and start extruding some material. If nothing come out of the nozzle there might be a clog. Therefore, the nozzle must be replaced or cleaned.

We recommend the use of a wear resistant nozzle, for instance with a ruby or ceramic tip. In order to avoid filament grinding, we recommend to load the filament without any pressure at all and then delicately increase it until the filament just gets pushed through the PTFE tube.

We recommend printing the piece on flexible plate. The part can be detached by binding it.

Printing parameters:

Printing speed: from 15 to 50 mm/s depending on the shape of the part

Layer height: from 0.3mm down to 0.1mm

Retraction: no retraction

Fan speed: 100% (the higher the better the print quality)

Wall line count: 3 recommended (at least two)

Infill: any 2D pattern (triangles, grid, honeycomb, rectilinear)

Infill density: from 100% down to 20% (the top surfaces above the infill depends on the pattern infill density)

Top/bottom surface number:

- for a 0.1mm layer height: 10
- for a 0.2mm layer height: 5

Supports structure: Can be printed using the same material or a soluble material via a dual extrusion system. In order to have a great surface quality at least 3 dense top layers should be printed between the support structure and the part.