Sample Slic3r settings for Proto-pasta HTPLA on Prusa MK3

Alex Dick
Protoplant
12/12/18
Layer height

Layer height: 0.2 mm
First layer height: 0.2 mm or %

Vertical shells

Perimeters: 1 (minimum)
Spiral vase:

Recommended object thin wall thickness for layer height 0.20 and 2 lines: 0.86 mm

Horizontal shells

Solid layers: Top: 4 Bottom: 4

Quality (slower slicing)

Extra perimeters if needed: √
Ensure vertical shell thickness: √
Avoid crossing perimeters: √
Detect thin walls: False
Detect bridging perimeters: √

Advanced

Seam position: Nearest
External perimeters first: √
### Extrusion width

- **Default extrusion width:** 0.7 mm or % (leave 0 for auto)
- **First layer:** 0 mm or % (leave 0 for default)
- **Perimeters:** 0 mm or % (leave 0 for default)
- **External perimeters:** 0 mm or % (leave 0 for default)
- **Infill:** 0 mm or % (leave 0 for default)
- **Solid infill:** 0 mm or % (leave 0 for default)
- **Top solid infill:** 0 mm or % (leave 0 for default)
- **Support material:** 0 mm or % (leave 0 for default)

### Overlap

- **Infill/perimeters overlap:** 25% mm or %

### Flow

- **Bridge flow ratio:** 1

### Other

- **Clip multi-part objects:**
- **Elephant foot compensation:** 0 mm
- **XY Size Compensation:** 0 mm
- **Resolution:** 0 mm
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Color</td>
<td>Red</td>
</tr>
<tr>
<td>Diameter</td>
<td>1.75 mm</td>
</tr>
<tr>
<td>Extrusion multiplier</td>
<td>1</td>
</tr>
<tr>
<td>Density</td>
<td>1.24 g/cm³</td>
</tr>
<tr>
<td>Cost</td>
<td>50 money/kg</td>
</tr>
</tbody>
</table>

**Temperature °C**

<table>
<thead>
<tr>
<th>Source</th>
<th>First layer</th>
<th>Other layers</th>
</tr>
</thead>
<tbody>
<tr>
<td>Extruder</td>
<td>240</td>
<td>240</td>
</tr>
<tr>
<td>Bed</td>
<td>60</td>
<td>60</td>
</tr>
</tbody>
</table>
Enable

Keep fan always on: ☑️
Enable auto cooling: ☑️

If estimated layer time is below ~5s, fan will run at 100% and print speed will be reduced so that no less than 5s are spent on that layer (however, speed will never be reduced below 10mm/s). During the other layers, fan will always run at 100% except for the first 2 layers.

Fan settings

Fan speed: Min: 100  % Max: 100  
Bridges fan speed: 100  
Disable fan for the first: 2  

Cooling thresholds

Enable fan if layer print time is below: 5  approximate seconds
Slow down if layer print time is below: 5  approximate seconds
Min print speed: 10  mm/s