

Printing Guide and Suggestions - Ars Moriendi 3D

These are the settings we suggest for a sturdy mug with a nice finish.

- 3 walls/shells
- 4 top and bottom layers
- 10-12% infill
- 0.2mm layer height
- No support
- Default printing speed between 50 and 80mm/s (Sometimes a higher speed can decrease the prints quality depending on the printer, this is a setting to experiment with!)

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. Most slicer programs are able to change settings for a specific region, and a 0.08mm layer height can be used to give it a smoother surface in those specific parts.

Should you wish for a flipped/mirrored version of the mug, most slicer programs are able to mirror the mug. Remember to mirror the lid, in case you plan to use it.

If it's the intention to paint the print, a thin coat of filler primer, followed by a gentle sanding can be recommended, before applying the final choice of primer.

www.ArsMoriendi3D.com