

Print and Finishing Info

Recommended Print Settings:

Filament choice is not important.

Use recommended temperature settings for filament chosen.

Support is optional depending on your overhang performance based on filament chosen.

Infill - 20%

Layer Height - 0.1 - 0.3mm

Orientation - lay flat on back.

Glue together after printing or after finishing.

Finishing:

Remove all support material.

Sand down peice. Wipe down part after sanding.

Apply primer (I prefer sandable auto primer).

Sand primer lightly. Wipe down part after sanding.

Apply 2 coats of desired paint color.

Apply clear coat.