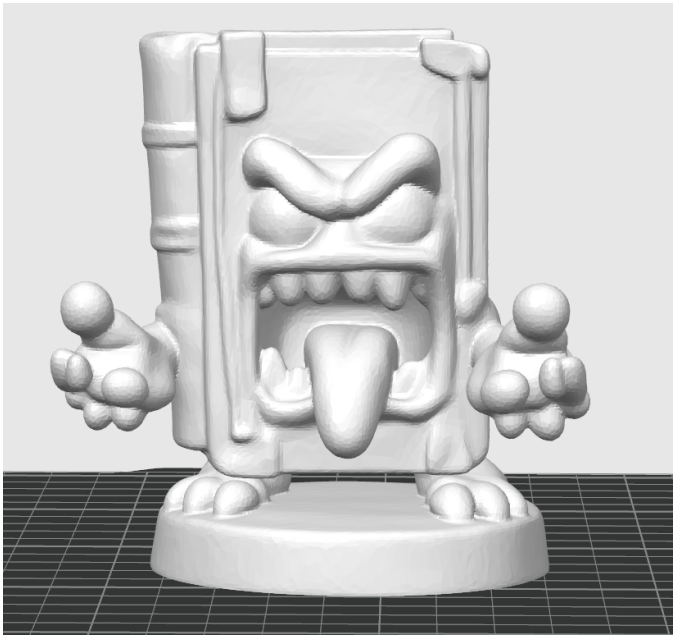
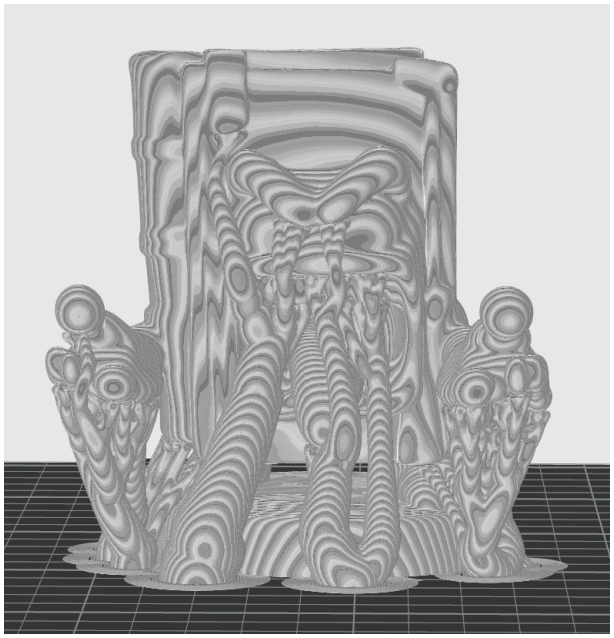
BOOK OF DOOM

100 mm collectible | FDM printing | PLA+



Front view
Unpainted model preview



Supplied sliced preview
Tree supports surround the face and hands.

Example slice: time and material

Estimate	Bambu Lab A1 / Bambu Studio estimate
Total time	5 hours 54 minutes
Model printing time	5 hours 30 minutes
Preparation / timelapse	6 minutes 25 seconds / 18 minutes 6 seconds
Total filament	91.02 g / 30.27 m
Model filament	83.00 g / 27.60 m
Support filament	8.02 g / 2.67 m
Filament changes	0
Displayed material cost	2.09 - currency not shown; based on your slicer filament price

Estimates transcribed from your supplied slicing screenshot, not measured print results. Timelapse is included in this slice. Actual time and usage can vary.

For all compatible FDM printers. The example settings and estimates use a Bambu Lab A1 with a 0.4 mm nozzle. Select your own printer, plate and filament profile, then slice again for your setup.

PRINT SETTINGS

100 mm collectible | FDM printing | PLA+

A bad-tempered spellbook, ready for your paint scheme.

File: book of Doom update cut.3mf
Dimensions: 97.2 × 80.1 × 100.0 mm (X × Y × Z)
Assembly: one connected model with integral pedestal.

01 Reference print settings

Setting	Shared collection profile
Printer / nozzle	Bambu Lab A1 / 0.4 mm
Filament preset	eSUN PLA+ @BBL A1
Build plate	Textured PEI Plate
Layer height	0.15 mm; first layer 0.20 mm
Walls / infill	3 wall loops / 15% gyroid
Top / bottom shells	Top: 5 layers, 1.0 mm minimum thickness; bottom: 4 layers
Nozzle / bed temperature	220°C first and later layers / 65°C first and later layers
Wall speeds	Outer 80 mm/s / inner 150 mm/s
First-layer speed	50 mm/s
Part-cooling fan range	60-80% in saved filament profile; retain its automatic rules
Seam / adhesion	Aligned seam / brim ears, 5 mm width, 0.1 mm gap

02 Set up before slicing

1. Import the model into your slicer and select your own printer, nozzle, plate and filament profile. The supplied 3MF contains an A1 example setup; do not send it unchanged to another printer.
2. Keep the supplied upright orientation, pedestal on the bed. The inspected base is flat at Z = 0; no further cut is needed.
3. Use a clean plate and confirm the filament preset matches your spool. Other PLA+ brands may need their own profile.

Inspection status: closed, consistently wound mesh; one connected component. Flat underside approximately 4,207 mm². Geometry checked in the previously supplied cut 3MF. These new screenshots provide the slice estimates; the shared settings are supplied by the creator. A screenshot cannot verify every toolpath or establish that the mesh is unchanged.

PRINT, CLEAN & PAINT

03 Supports and preview checks

Support setting	Saved value
Type / placement	Tree (auto), default style / on build plate only
Threshold angle	30°
Top / bottom Z gap	0.20 mm / 0.20 mm
Support-to-object XY gap	0.35 mm
Interface	2 top and 2 bottom layers; top spacing 0.50 mm

Before pressing Print: slice and inspect the first layer for a continuous pedestal footprint. Check underneath the reaching hands, fingers, mouth/teeth, tongue and projecting cover edges. Build-plate-only trees may not reach every area above the pedestal. If coverage is missing, use local support painting or allow supports from the model, then recheck removal access. Avoid packing the mouth with unnecessary support.

Your support preview: substantial trees cover the front of the model. Cut them away in small sections, especially around the mouth and hands. It illustrates the supplied slice, but cannot confirm all contact points or removal clearances.

04 Remove supports and prepare for paint

1. Let the plate and model cool before removal. Handle the pedestal and body rather than the fingers.
2. Remove trees in small sections with cutters. Support each delicate feature while trimming; do not pull a large tree against a hand or tooth. Wear eye protection when clipping.
3. Trim remaining nubs and gently sand accessible marks. Keep sanding light around facial details; control dust and follow abrasive/primer safety instructions.
4. Remove dust, allow the model to dry, then apply thin coats of a plastic-compatible primer. Paint with thin acrylic layers; washes and drybrushing can bring out the spine bands, cover edges and page block. Finish with a compatible clear coat if desired.

05 Troubleshooting and optional refinements

What you see	What to try next
Regular visible layer lines	Use the supplied 0.15 mm first. Optional 0.12 mm layers can reduce stepping, with a longer print time.
Rough undersides	Inspect support coverage and overhangs. Smaller layers alone will not remove support scars or sagging.
Uneven pedestal ridges	Check sliced wall outlines against the mesh. If they are smooth, investigate flow/speed consistency; a flat bottom does not fix sidewall ridges.
Marks on the face	Move or paint the seam toward the rear; this is an optional change from the saved aligned seam.
Uneven outer surfaces	Optional test: outer walls at 50-60 mm/s. Change one setting at a time and compare.
Base lifting / poor contact	Check plate cleanliness, selected plate and first-layer preview before adjusting adhesion settings.

For other printers: retain the model dimensions, but select your own machine and filament presets and regenerate supports. This is an A1 starting profile, not a universal printer profile. Keep temperature and cooling matched to your material.