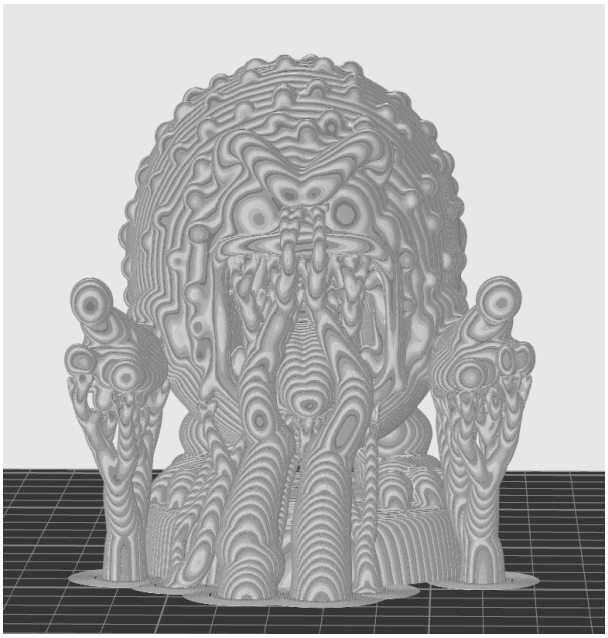
BUNGHOUL

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	6 hours 16 minutes
Model printing time	5 hours 51 minutes
Preparation / timelapse	6 minutes 25 seconds / 17 minutes 43 seconds
Total filament	95.29 g / 31.69 m
Model filament	87.95 g / 29.25 m
Support filament	7.34 g / 2.44 m
Filament changes	0
Displayed material cost	2.19 - 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 fearsome bun with plenty of bite, ready for paint.

Model: Bunghoul
Target height: 100 mm (shared collection size)
Design: character on an integral pedestal.
Confirm dimensions in your slicer; no model file inspected.

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. These reference settings use an A1 example setup; adapt them to your own printer.
2. Keep the supplied upright orientation, pedestal on the bed. Check that the underside sits flat at Z = 0 and that the first-layer preview forms a continuous footprint.
3. Use a clean plate and confirm the filament preset matches your spool. Other PLA+ brands may need their own profile.

Review status: based on creator-supplied images, slice estimates and shared collection settings. No Bunghoul mesh or 3MF was supplied for inspection, so base flatness, mesh integrity, exact dimensions and complete toolpaths remain unverified.

PRINT, CLEAN & PAINT

03 Supports and preview checks

Support setting	Shared collection profile
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 rounded belly and lower mouth 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, hands and beneath the body. 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 or teeth.
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 raised bun texture, teeth and facial features. 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.