



Hook



Common Mediante

[VIEW IN BROWSER](#)

updated 12. 5. 2025 | published 12. 5. 2025

Summary

Designed for style and strength, it's perfect for hanging coats, bags, towels, or keys, helmet etc.

[Household](#) > [Garage](#)

Tags: [wall](#) [holder](#) [hook](#) [helmet](#) [wallmount](#) [modern](#)
[garage](#) [wallmounted](#) [useful](#) [motorcycle](#) [useful3dprints](#)
[hooks](#)

Cura slicer

Settings for Strong PETG Prints

Printer & Material Basics

Nozzle Temp: 240–250 °C (start at 245 °C and adjust if needed)

Bed Temp: 75–85 °C

Cooling Fan: Off or max 20% (layer fan can weaken layer adhesion)

Enclosure: Recommended for consistent temps (not mandatory for PETG, but helps)

Layer & Wall Settings

Layer Height: 0.2 mm (or 0.28 mm for stronger/thicker layers)

Wall Line Count: 3

Wall Thickness: At least 2 – 3 mm

Top/Bottom Layers: 5-6

Top/Bottom Thickness: 1.6-2.5 mm

Infill

Infill Density: 90 - 100%

Print Speed

Print Speed: 30-45 mm/s (slower = stronger)

Wall Speed: 25-30 mm/s

Initial Layer Speed: 15-20 mm/s

Other Key Settings

Initial Layer Height: 0.24 mm (good bed adhesion)

Z Hop When Retracted: On

Retraction Speed: 25-35 mm/s

Generate support : Enable

Support structure : Tree

Pro Tips:

Dry your filament – PETG absorbs moisture and it **greatly reduces strength**.

Use a **brim or raft** if warping is an issue.

For critical strength parts: consider **increasing wall count and infill** rather than relying on 100% infill alone.

Turn off "**Print Cooling**" or keep it minimal for layer bonding.

Model files



hook.stl

License

This work is licensed under a
[Creative Commons \(4.0 International License\)](#)



Attribution—Noncommercial—No Derivatives

-
- ✖ | Sharing without ATTRIBUTION
 - ✖ | Remix Culture allowed
 - ✖ | Commercial Use
 - ✖ | Free Cultural Works
 - ✖ | Meets Open Definition