ZINC 3D Printing Guide

Version 1.2 - 3/22/2021

118 Design

NOTICE:

Introduction

- The files for all 3D printed parts can be found at: https://github.com/118design/ZINC/
- Please send any questions or comments to: support@118.design
- Hardware kits may be purchased at: www.118.design

Required tools & materials

All parts can be printed on a Prusa MK3s or an Ender 3. In order to fit these common printers the frame has been divided into 2 pieces and must be glued together. Files for single-piece frames are included in the repository but will require careful support placement and a much larger printer. I highly recommend sticking with the 2 piece frame.

Additionally there are 2 frame variations to choose from: one with a rail and one without. They are label FRAME_TOP_RAIL and FRAME_TOP_COMPACT respectively.

All pieces should be printed from a high-quality PETG or a filament with similar characteristics. Printing with other materials may change dimensional accuracy/durability. Do so at your own discretion.

PrusaSlicer is the recommended slicer.

Visually inspect each part after printing and do not use any parts that show discoloration, layer separation, or warping.

Parts List - blaster

Recommended Groupings

- FRAME_TOP_COMPACT **OR** FRAME_TOP_RAIL
- FRAME_FRONT_COMPACT **OR** FRAME_FRONT_RAIL
- FRAME_BOTTOM

PRIMARY COLOR
(0.2mm layer height)

- SLIDE_FRONT
- SLIDE_REAR **OR** SLIDE_REAR_TPULL **AND** TPULL

ACCENT COLOR 1
(0.15mm layer height)

- TRIGGER_LEFT
- TRIGGER_RIGHT
- MAG_RELEASE
- SEAR
- CATCH

ACCENT COLOR 2
(0.15mm layer height)

- MUZZLE_COMPACT **OR** MUZZLE_EXTENDED

HIGH-VIS ORANGE

Parts List - magazine

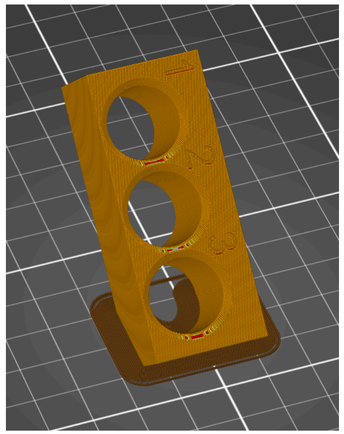
- MAG_BODY (any length)
- MAG_FOLLOWER
- MAG_BASE

3D Printing Guide

How to select your files

Print a calibration cube. Verify your printer is dimensionally accurate.

Print the TOLERANCE_GAUGE (0.2mm layer height)



Check your barrel fit in each hole. The correct fit is very tight but should not require sanding. You may need pliers to remove the barrel if it is fully seated in the frame.

For reference with PrusaSlicer and an MK3s I use the UNDERSIZE file set.

1: UNDERSIZE

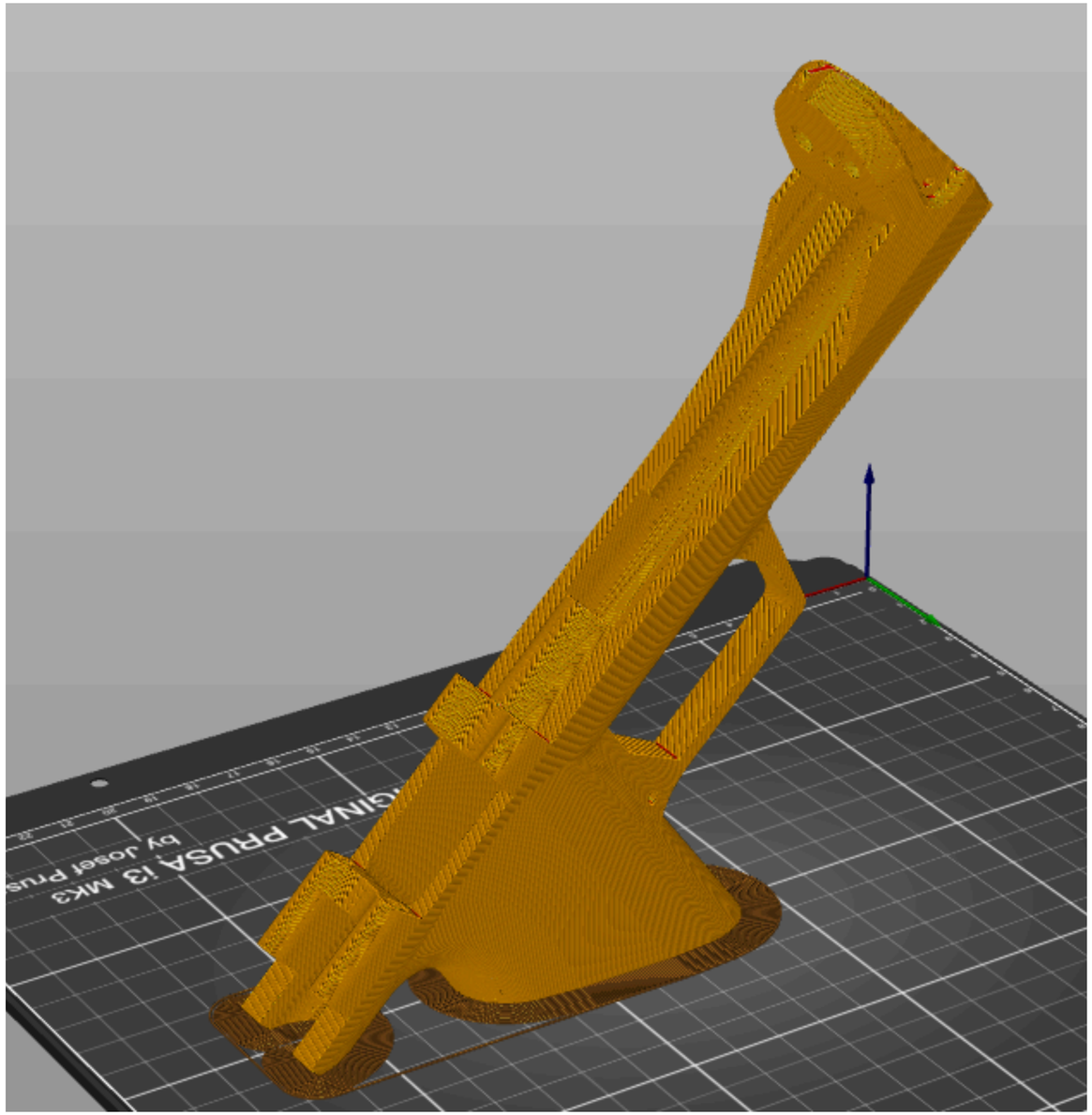
2: STANDARD

3: OVERSIZED

Select the branch that matches your test. Print all files from that branch and then all files in the main folder.

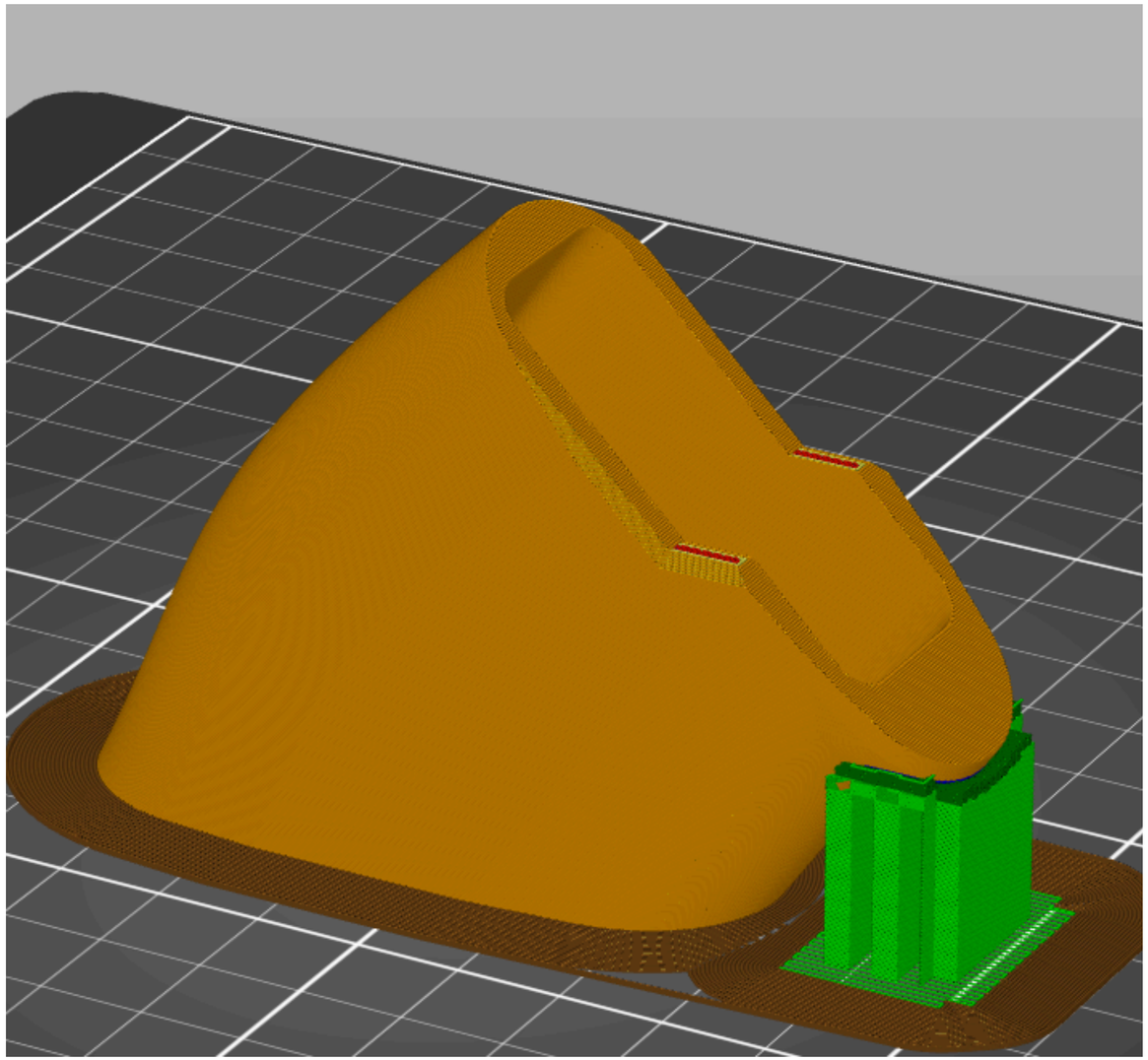
FRAME_TOP_(any)

- **3 sizes: UNDERSIZE, STANDARD, OVERSIZE**
- 2 variations: FRAME_TOP_RAIL or FRAME_TOP_COMPACT
- 0.2mm layer height
- At least 6 perimeters, 75-100% infill. Use your own discretion.
- No support
- Brim
- Ensure there is excellent bed adhesion so the frame can be properly glued
- Depending on the orientation of your cooling fan shroud you may need to experiment with the direction the print faces.
- Inspect after printing. DO NOT USE if there are any signs of shifted layers, under extrusion, or excessive moisture.



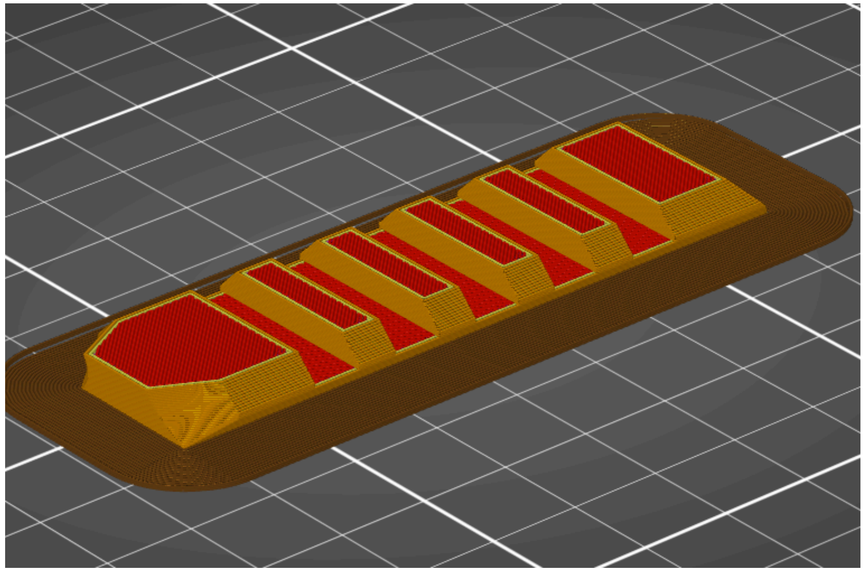
FRAME_BOTTOM

- 0.2mm layer height
- At least 6 perimeters, 100% infill
- Support underneath the overhang
- Brim
- Ensure there is excellent bed adhesion so the frame can be properly glued

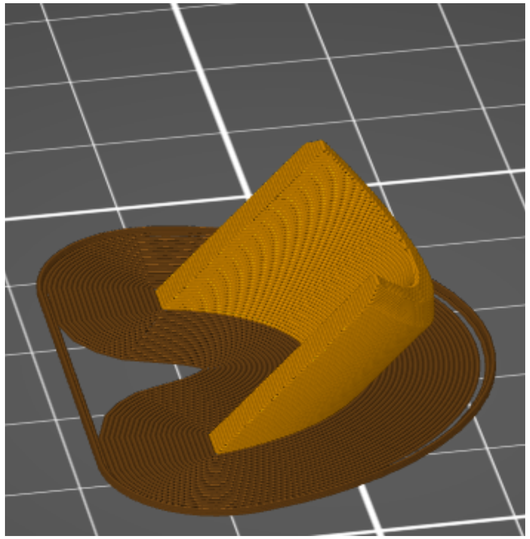


FRAME_FRONT_(any)

- 2 variations: FRAME_FRONT_RAIL or FRAME_FRONT_COMPACT (Depreciated)
- 0.2mm layer height
- At least 4 perimeters, 100% infill

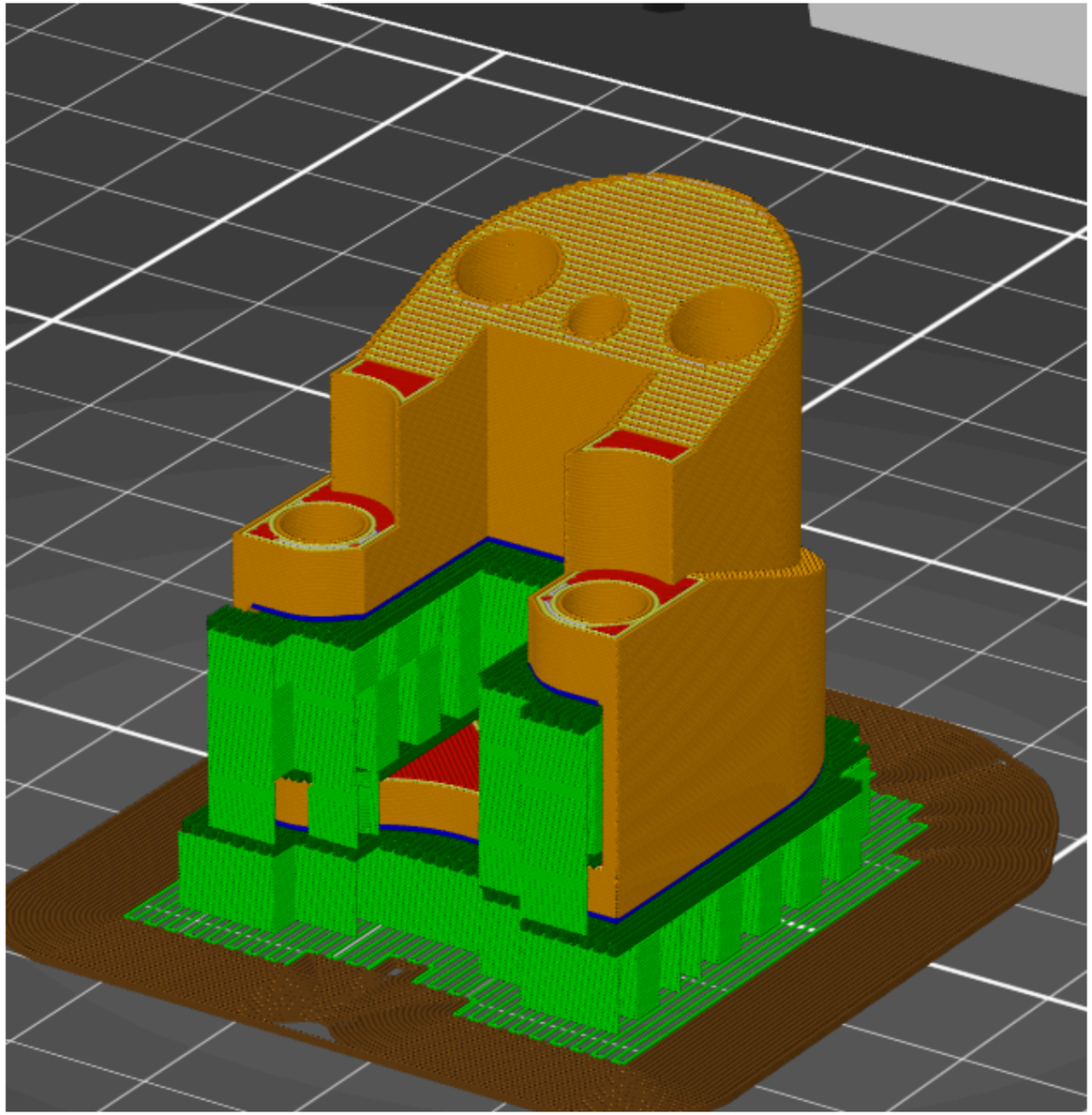


- No support
- Brim for COMPACT only
- Ensure there is excellent bed adhesion so the frame can be properly glued



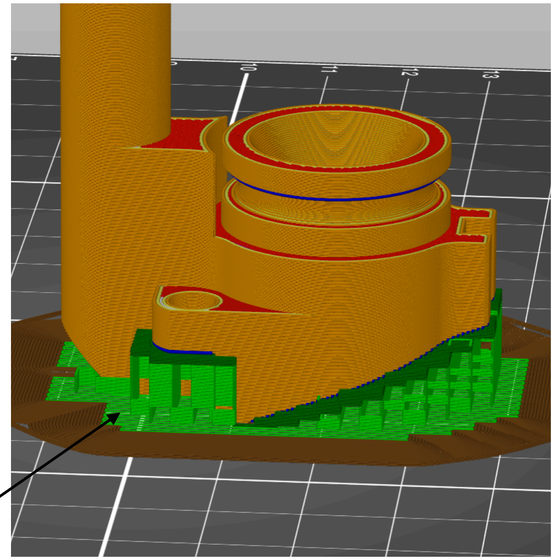
SLIDE_FRONT

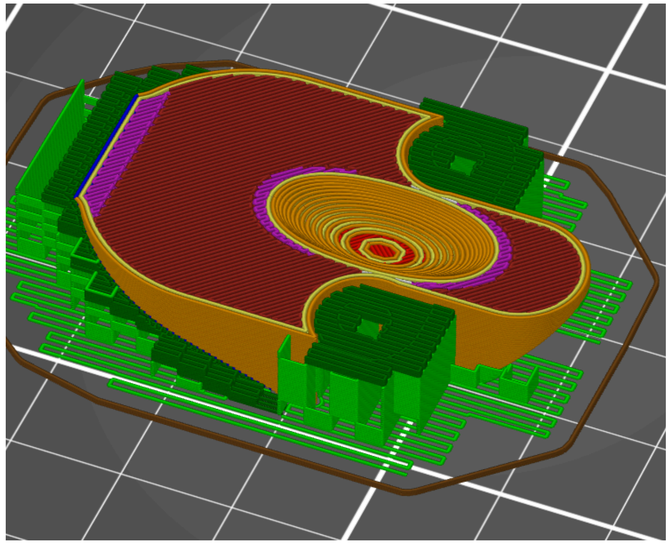
- 0.15mm layer height
- At least 6 perimeters, 100% infill
- Support everywhere
- This part will require cleanup to function properly. Take your time removing the support material

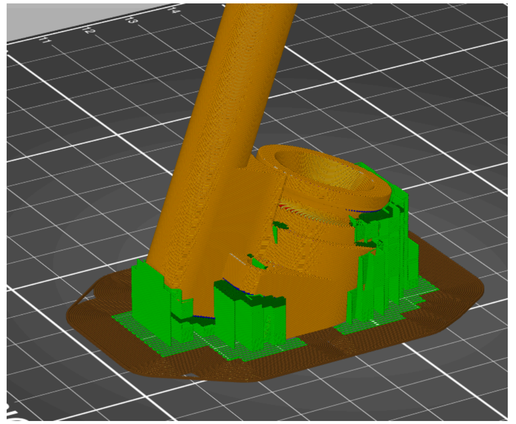


SLIDE_REAR_(any)

- 0.15mm layer height
- At least 6 perimeters, 100% infill
- Support for the overhangs but NOT in the internal turnaround as it will be impossible to remove
- Depending on the overhang capacity of your printer you can try this alternative orientation

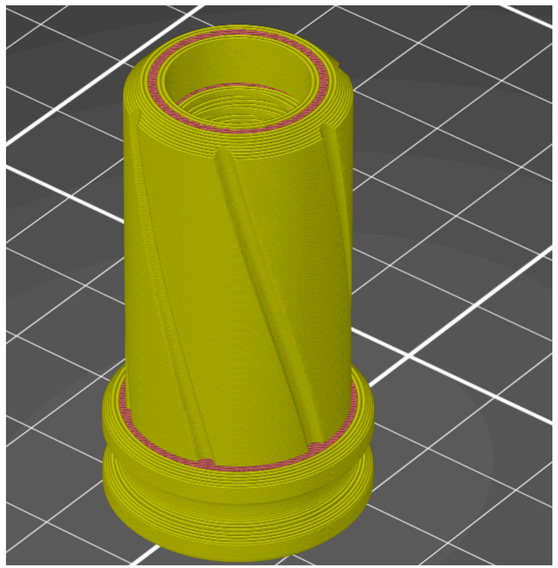


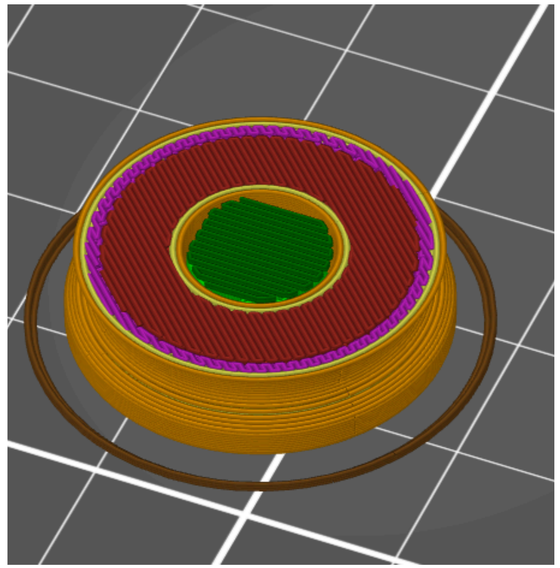




PLUNGER

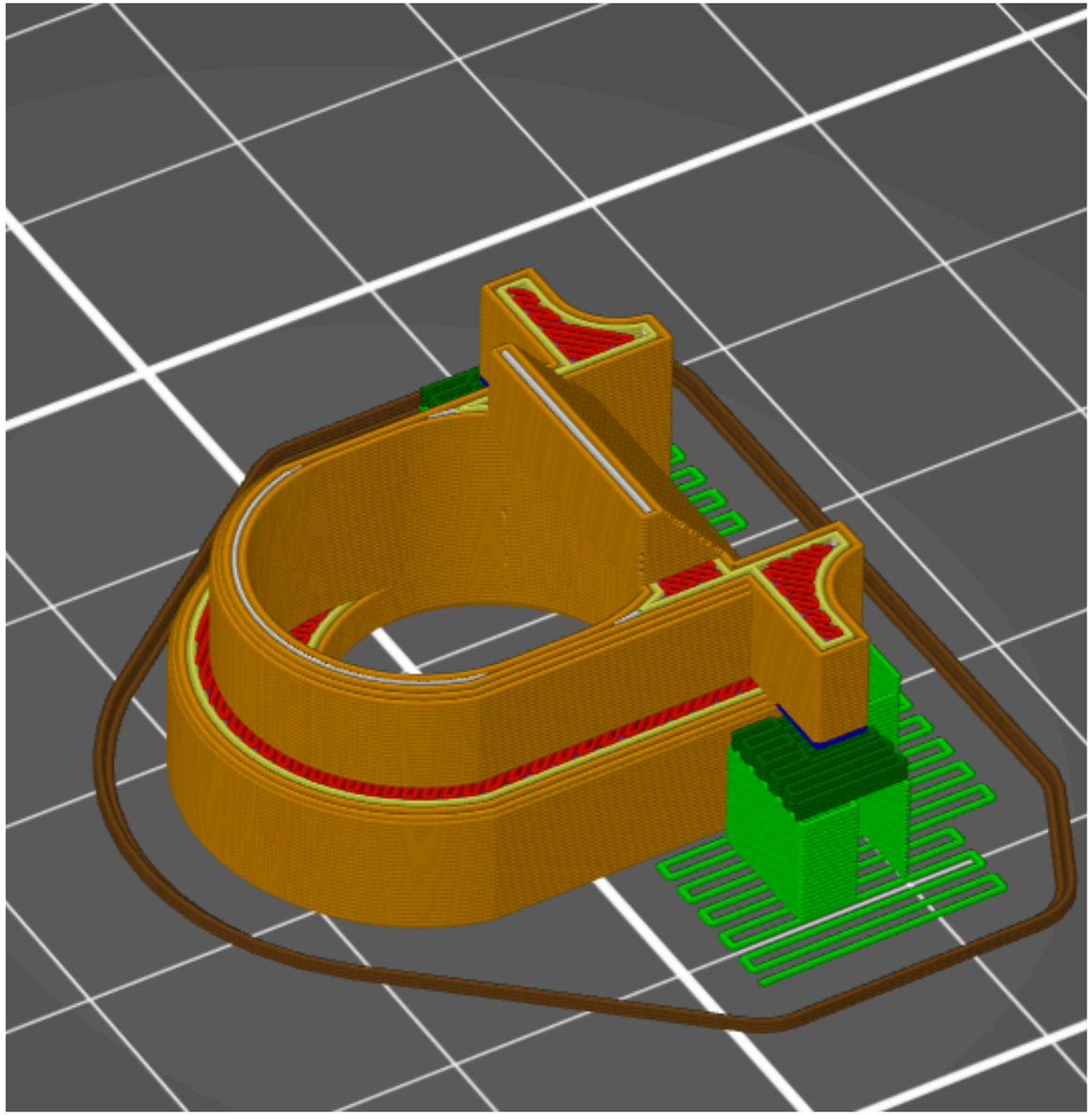
- 0.15mm layer height
- Support for the overhang at the base only
- 6 perimeters, 100% infill





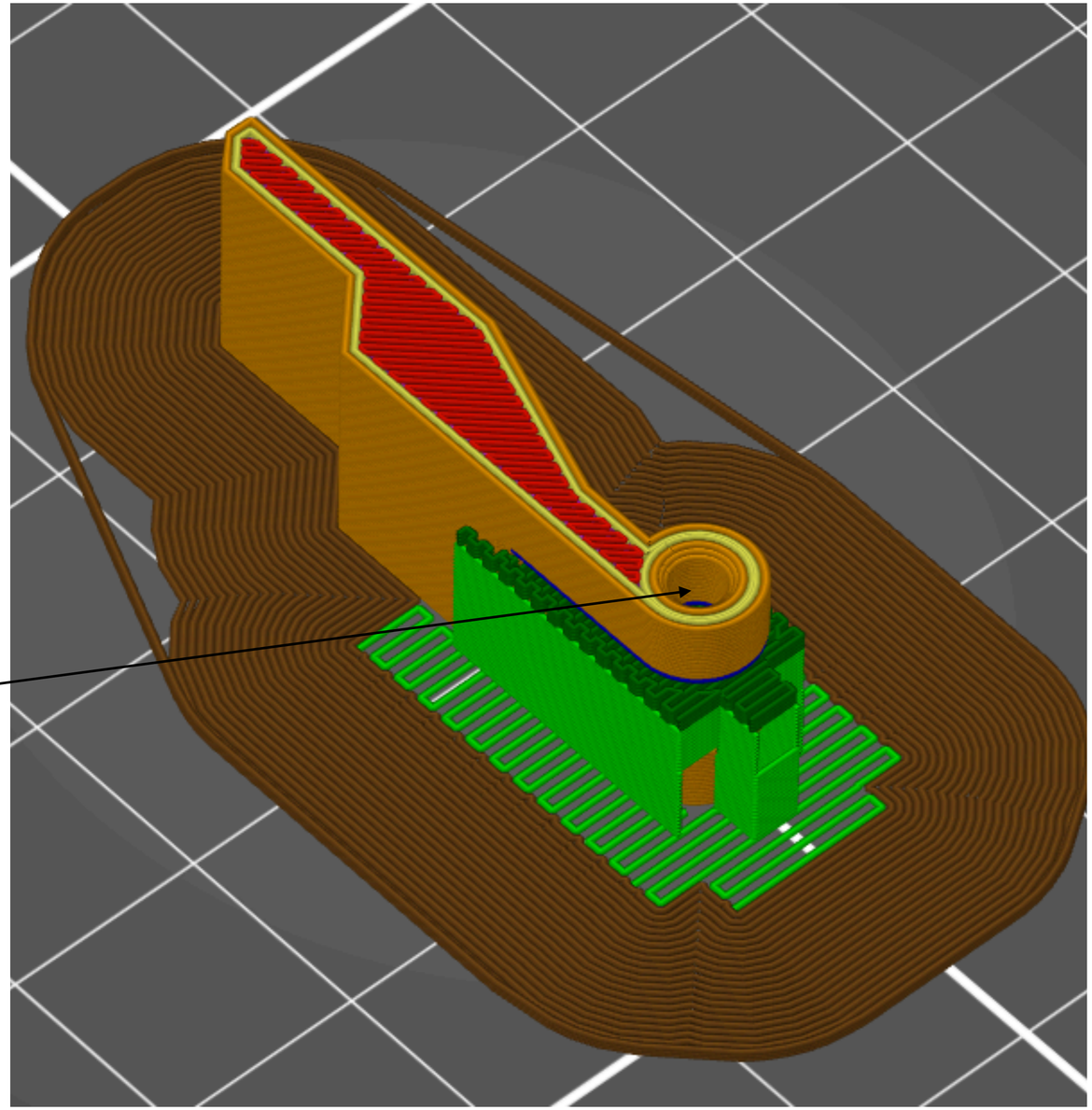
CATCH

- 0.2mm layer height
- 4 perimeters, 100% infill
- Ensure excellent bed adhesion
- Support everywhere
- You may need to file/sand/trim this part if your printer produces an “elephants foot” at the base of prints
- This is a wear part. Print extras and replace periodically.



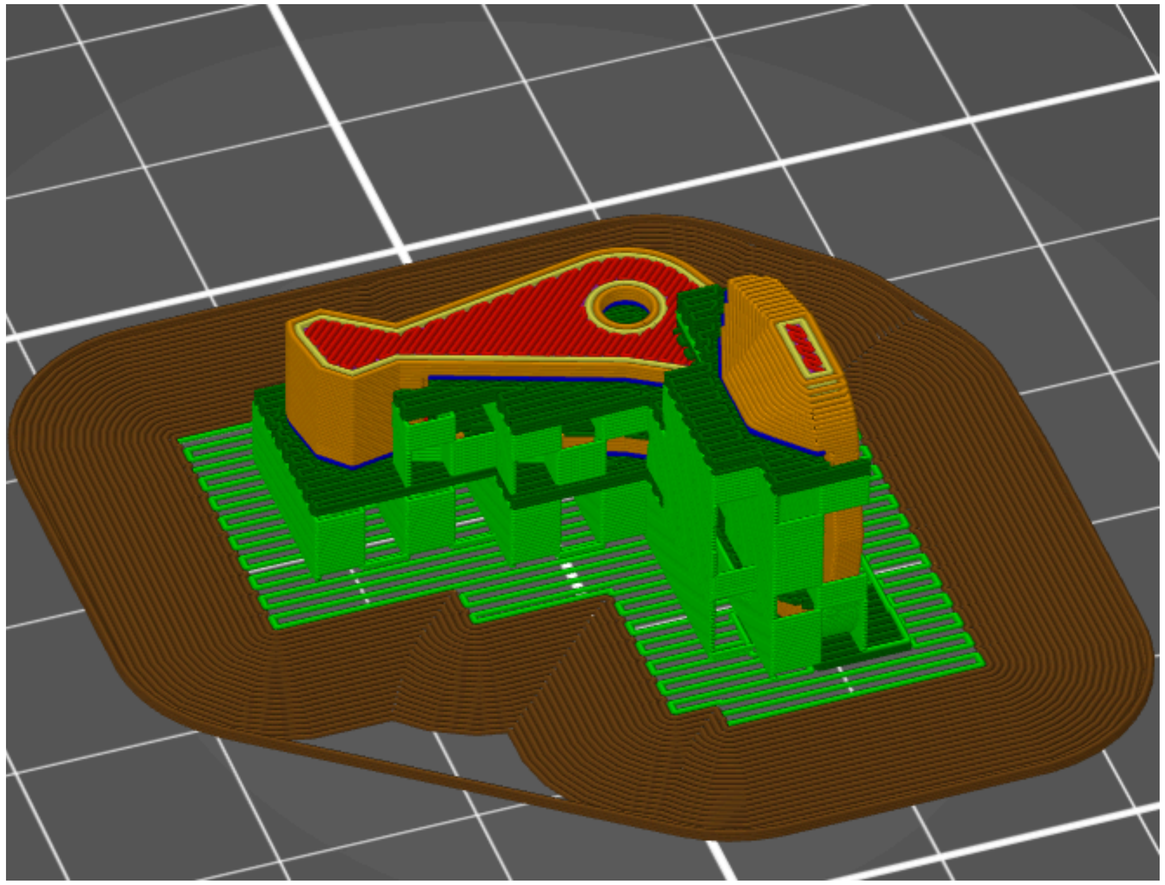
SEAR

- 0.15mm layer height
- 4 perimeters, 100% infill
- Support everywhere
- Ensure the chamfer is printed facing up



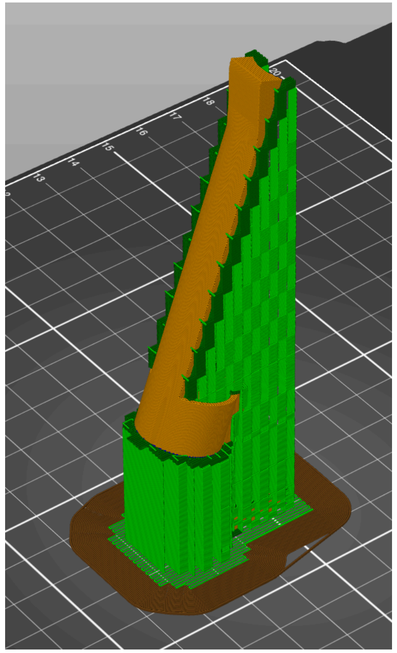
MAG_RELEASE

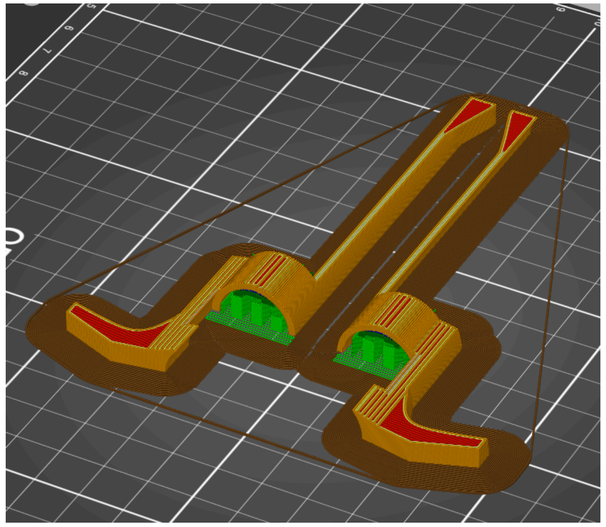
- 0.15mm layer height
- 100% infill
- Support everywhere



TRIGGER

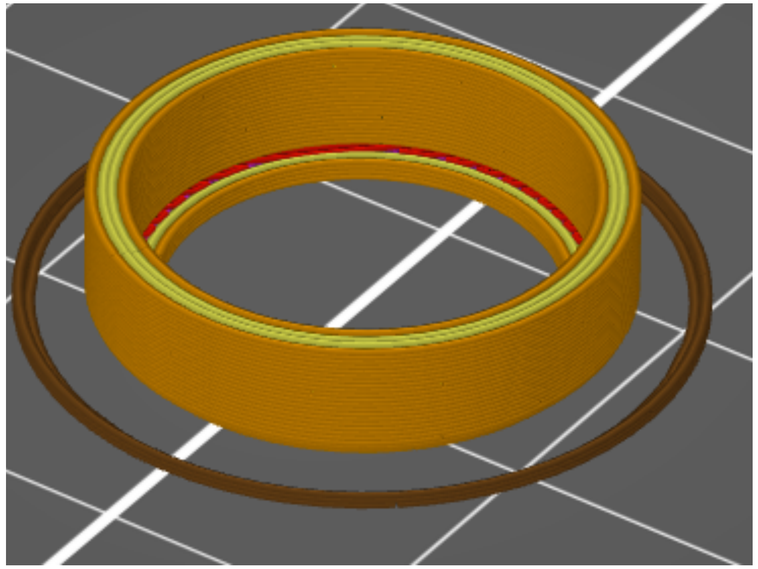
- 0.15mm layer height
- 4 perimeters, 100& infill
- Brim
- Supports under the overhang if your printer requires it
- Print the 1-piece version first. If you have durability issues then try the 2-piece version
- For the 2-piece version ensure there is excellent bed adhesion as these parts must be glued together
- These parts are delicate before they are glued. Be careful removing supports





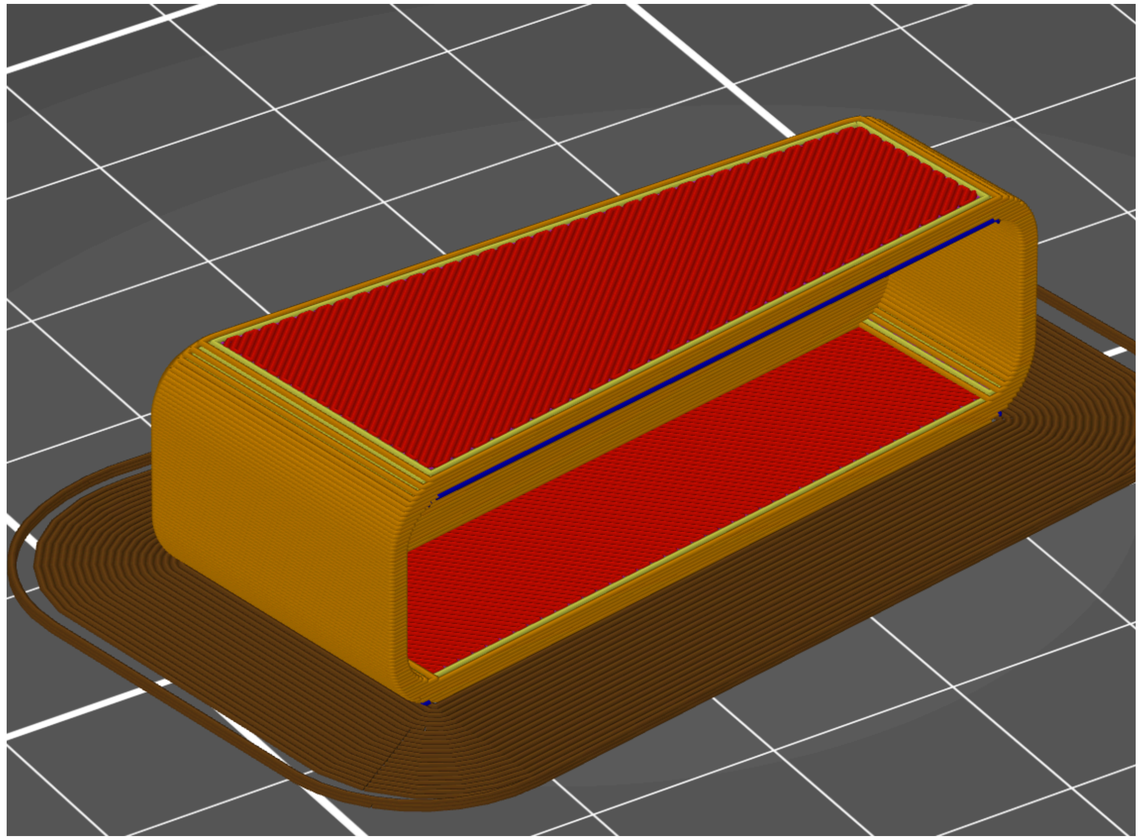
MUZZLE_(any)

- Print in a HIGH-VIS filament!



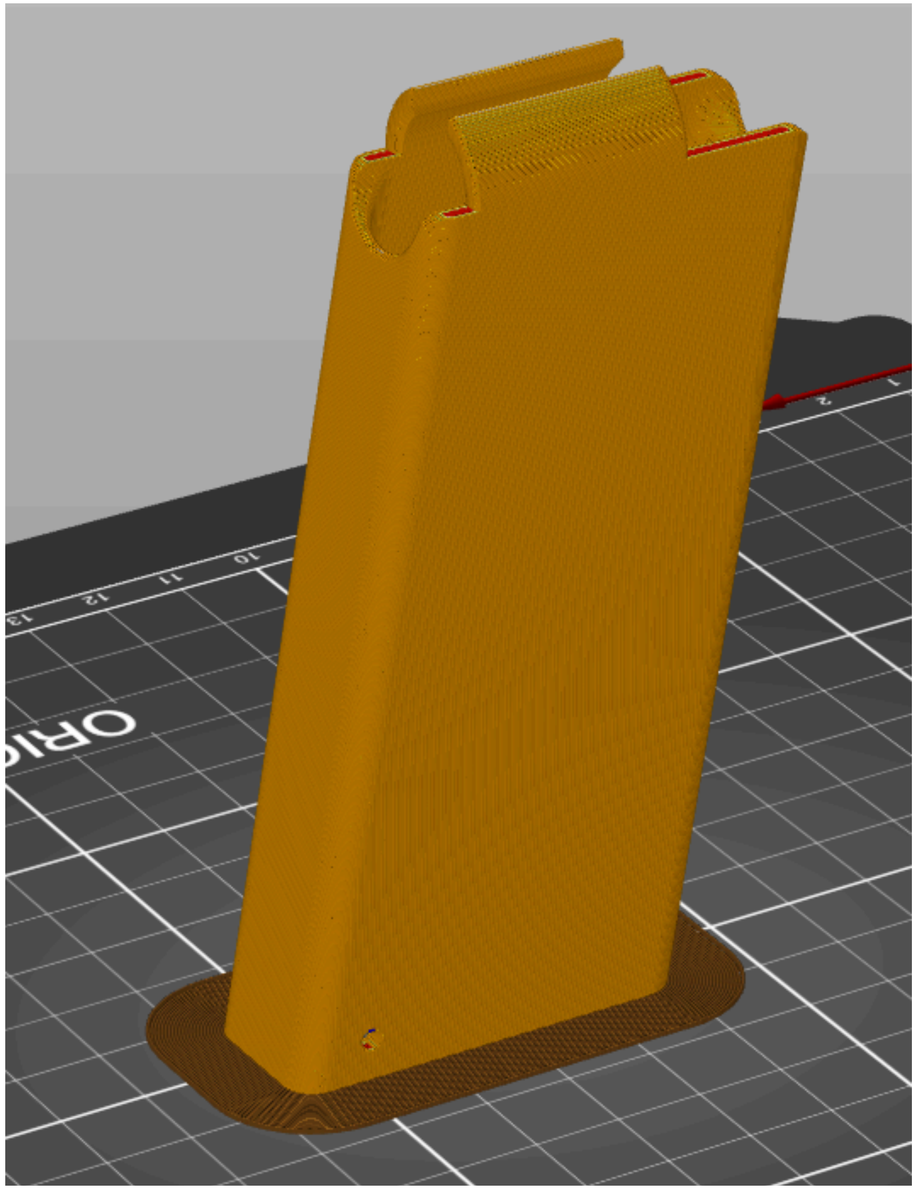
MAG_FOLLOWER

- 0.15m player height
- 4 perimeters, 100% infill
- No support needed if you’re printer is dialed in for bridging
- Print in this orientation to prevent binding against the magazine body, sand as needed



MAG_BODY

- **2 Sizes: STANDARD & UNDERSIZE (gravity drop)**
- 0.15mm layer height
- 4 perimeters, 100% infill
- Brim



MAG_BOTTOM

- **2 Sizes: STANDARD & UNDERSIZE (gravity drop)**
- 0.15mm layer height
- 100% infill