

# **ZINC 3D Printing Guide**

**Version 1.1 - 2/22/2021**

**118 Design**

# **NOTICE:**

**All files licensed under Creative Commons CC BY-NC-SA.  
Please visit: <https://creativecommons.org/licenses/by-nc-sa/4.0/> for more details.**

# 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](mailto:support@118.design)
- Hardware kits may be purchased at: [www.118.design](http://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 labeled 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

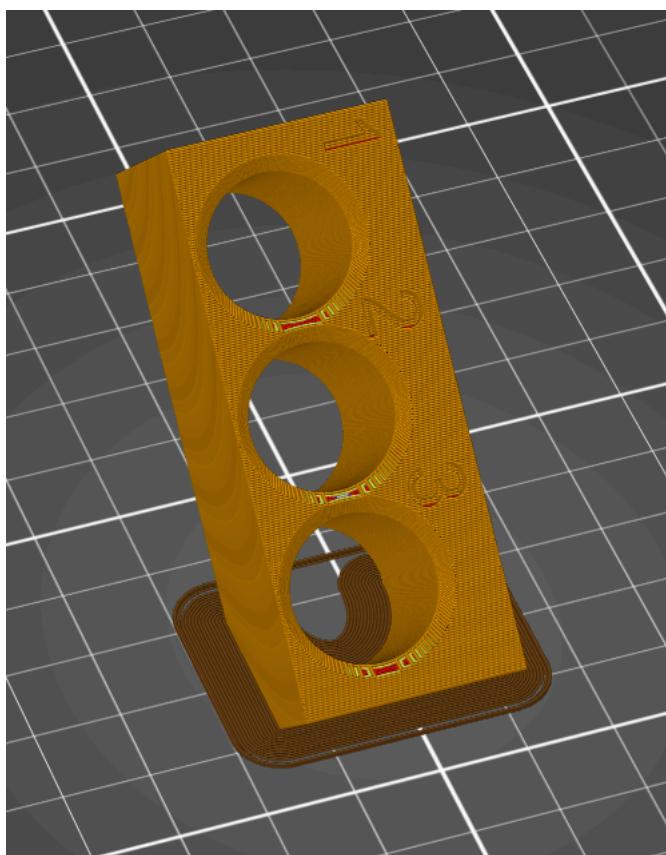
# 3D Printing Guide

## 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.

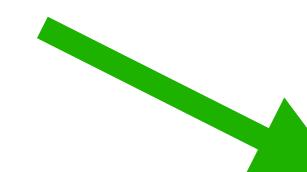
*For reference with PrusaSlicer and an MK3s I use the **UNDERSIZE** file set.*



**1: UNDERSIZE**



**2: STANDARD**

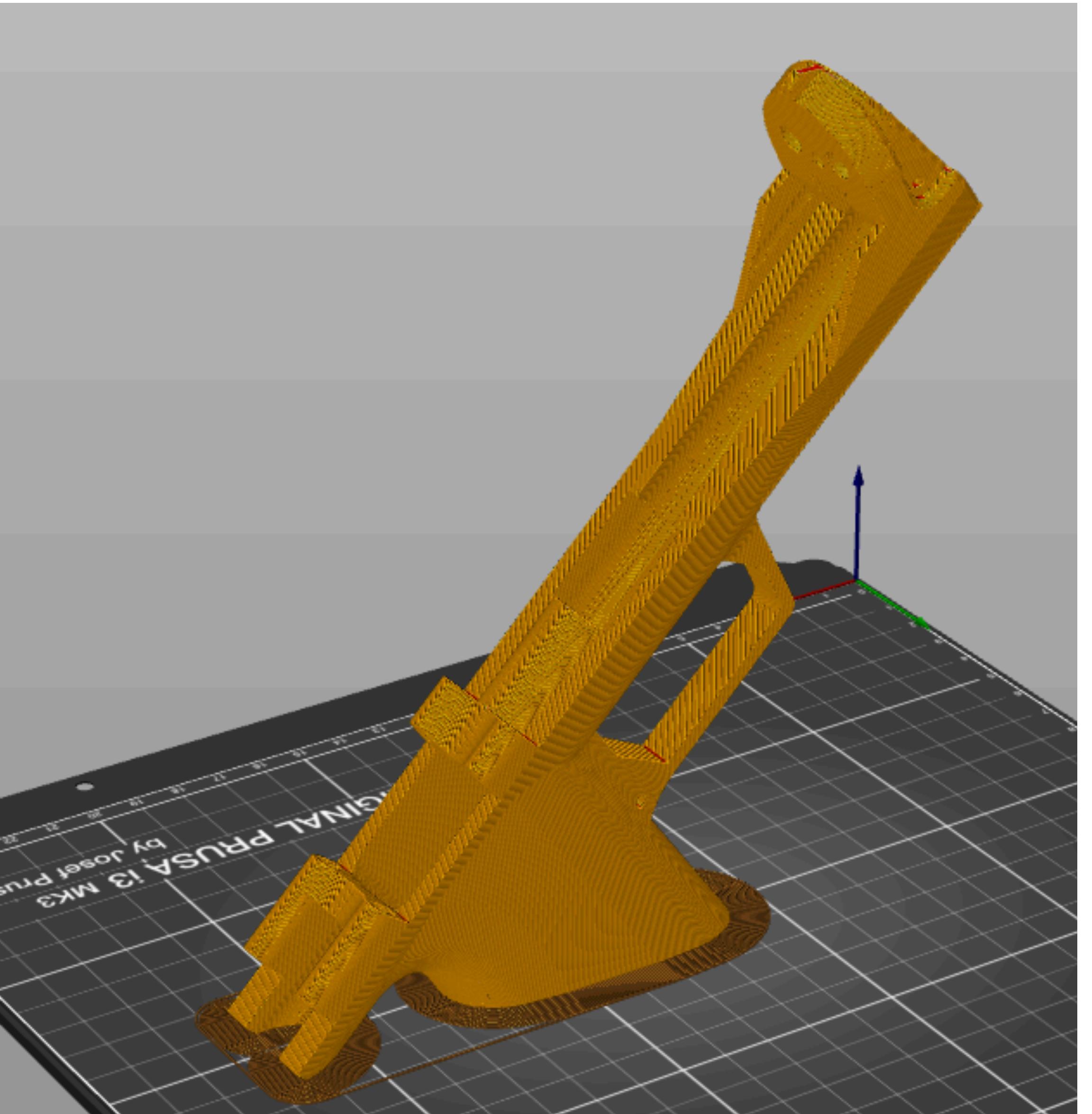


**3: OVERSIZED**

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

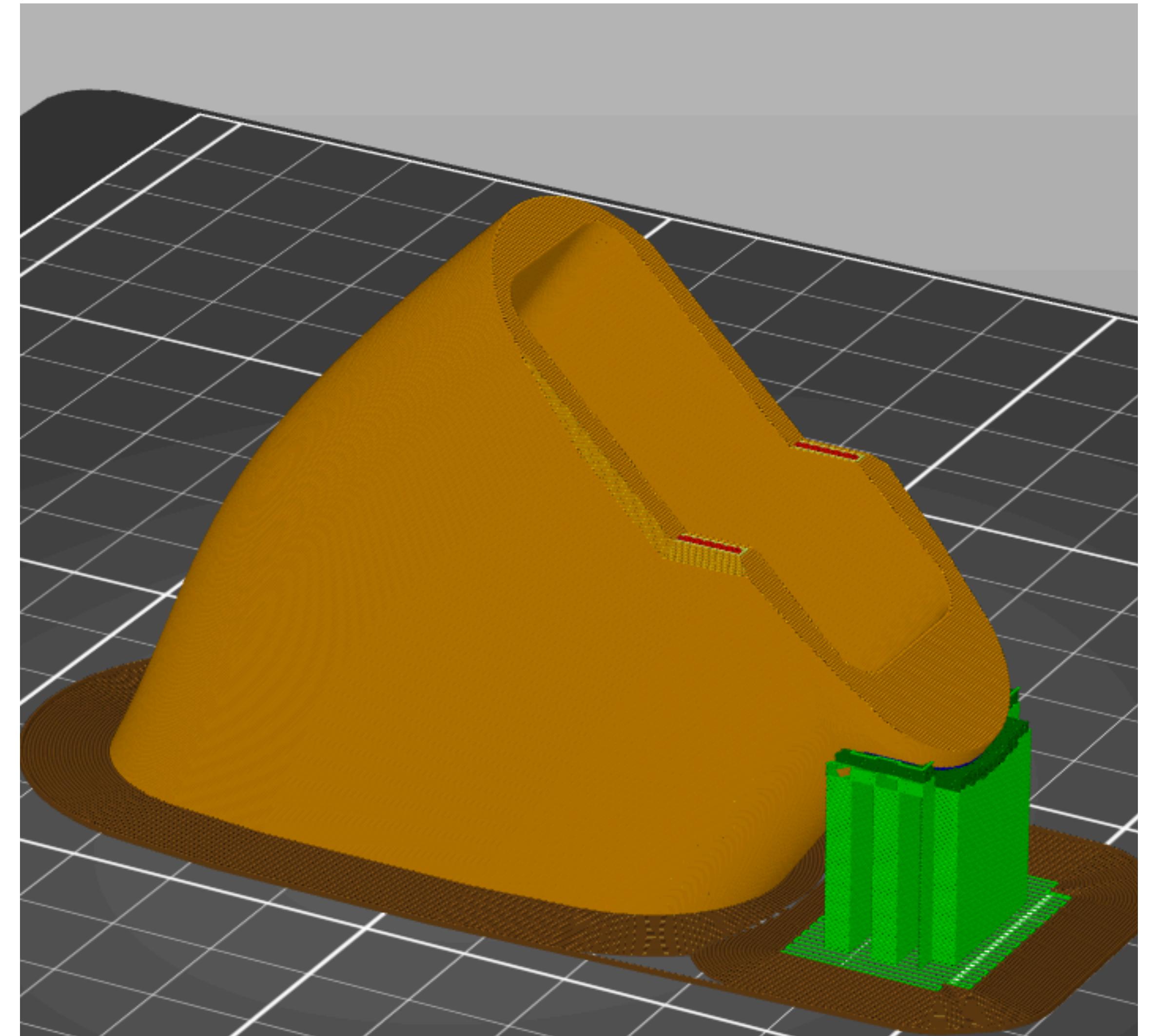
# FRAME\_TOP\_(any)

- **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
- Inspect after printing. DO NOT USE if there are any signs of shifted layers, under extrusion, or excessive moisture.



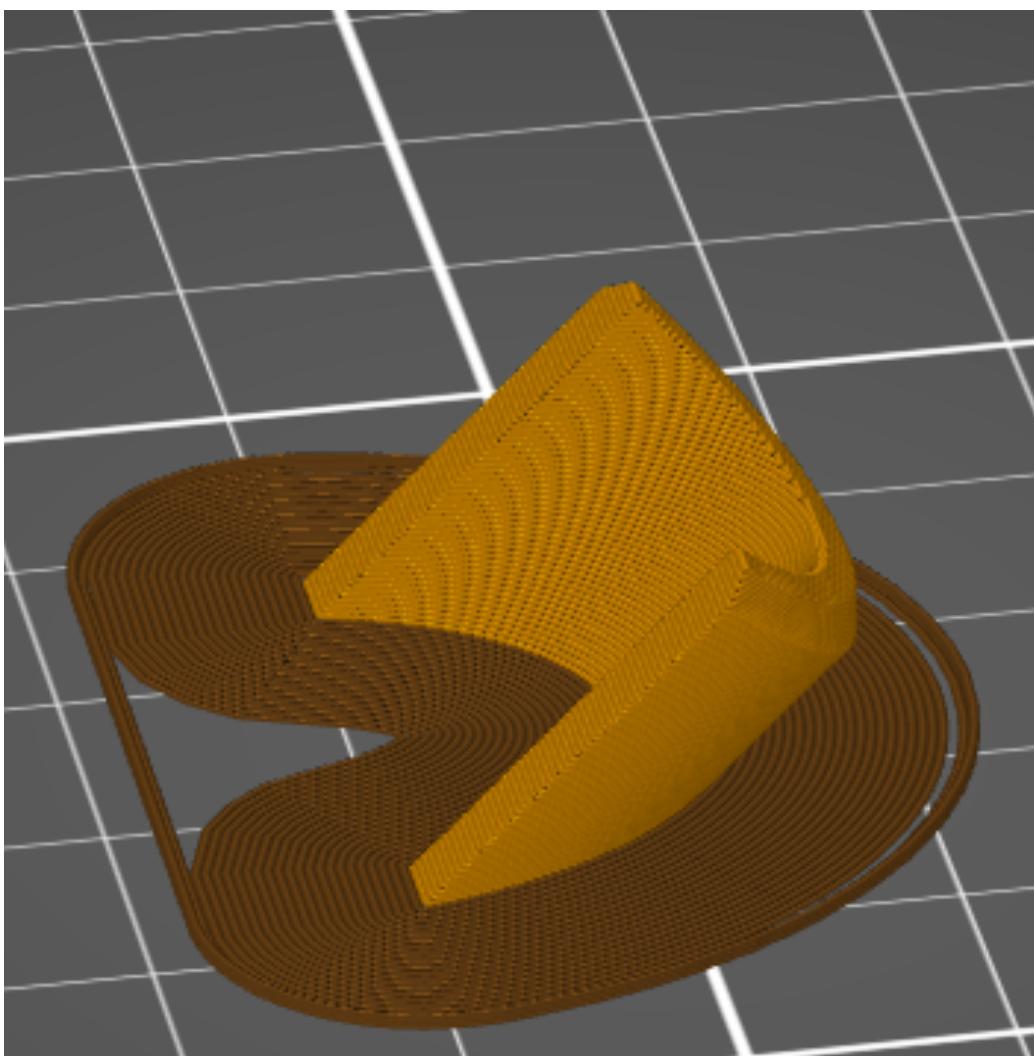
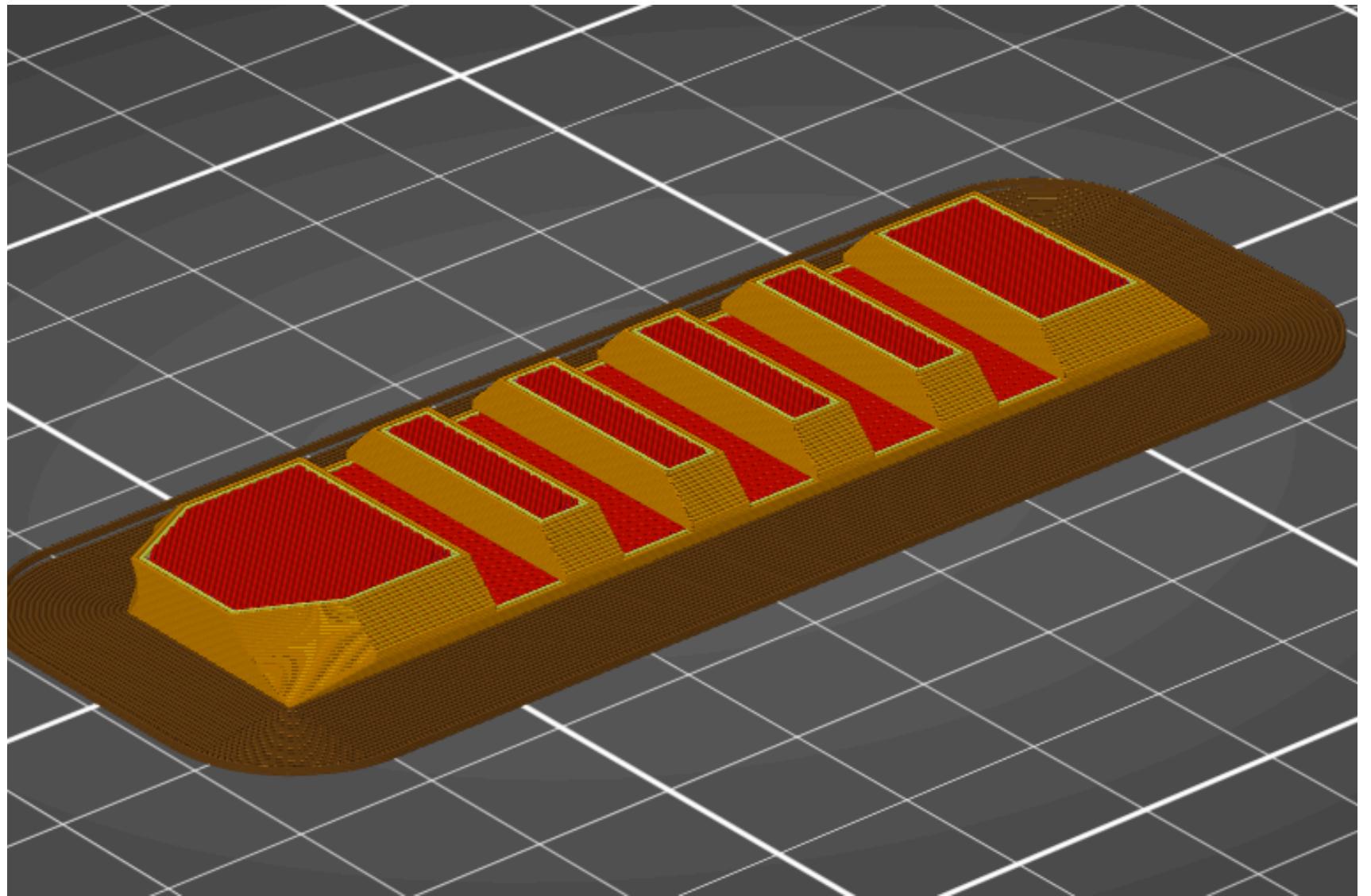
# FRAME\_BOTTOM

- 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



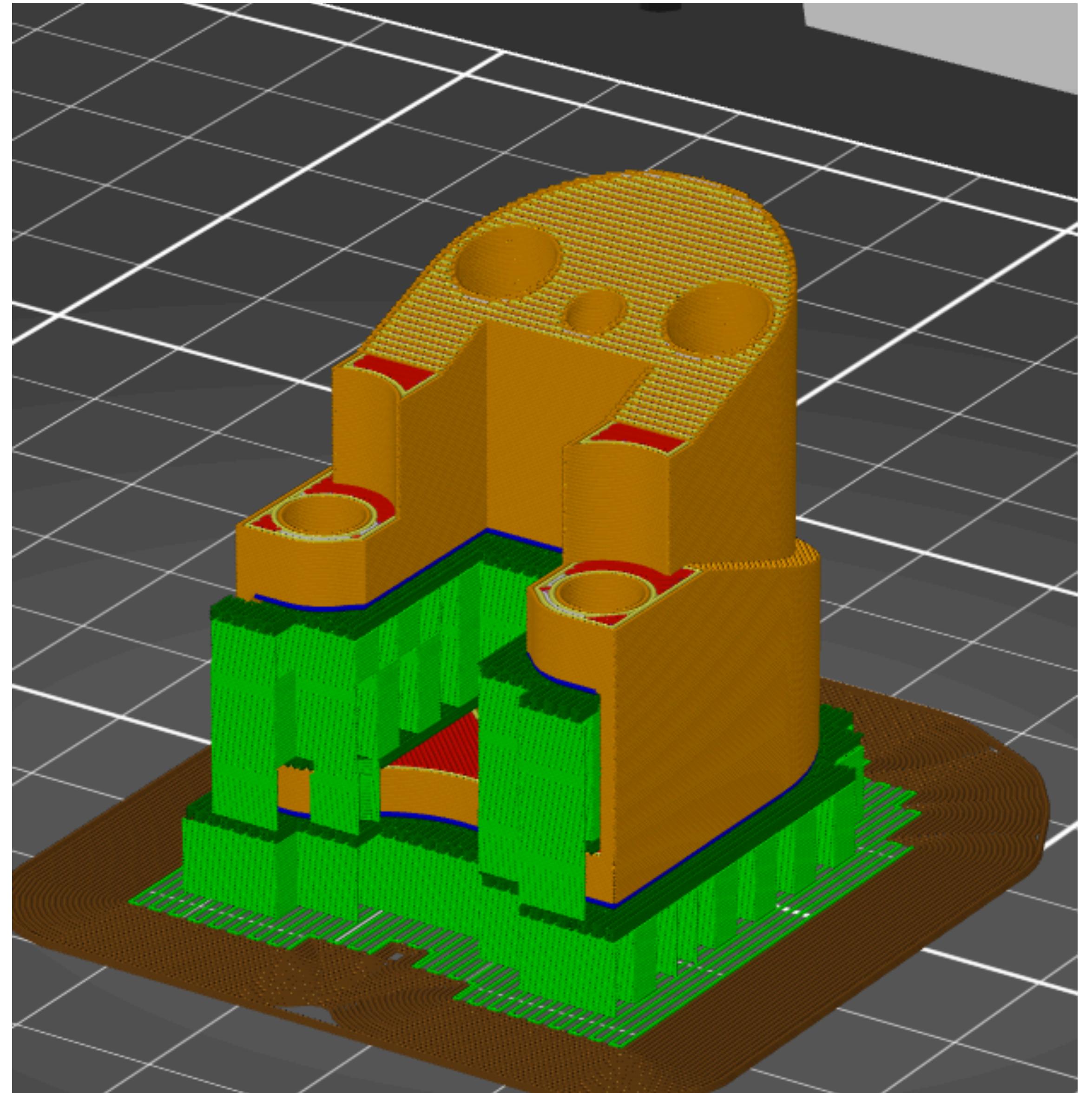
# FRAME\_FRONT\_(any)

- 2 variations: FRAME\_FRONT\_RAIL or FRAME\_FRONT\_COMPACT  
(Deprecated)
- 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



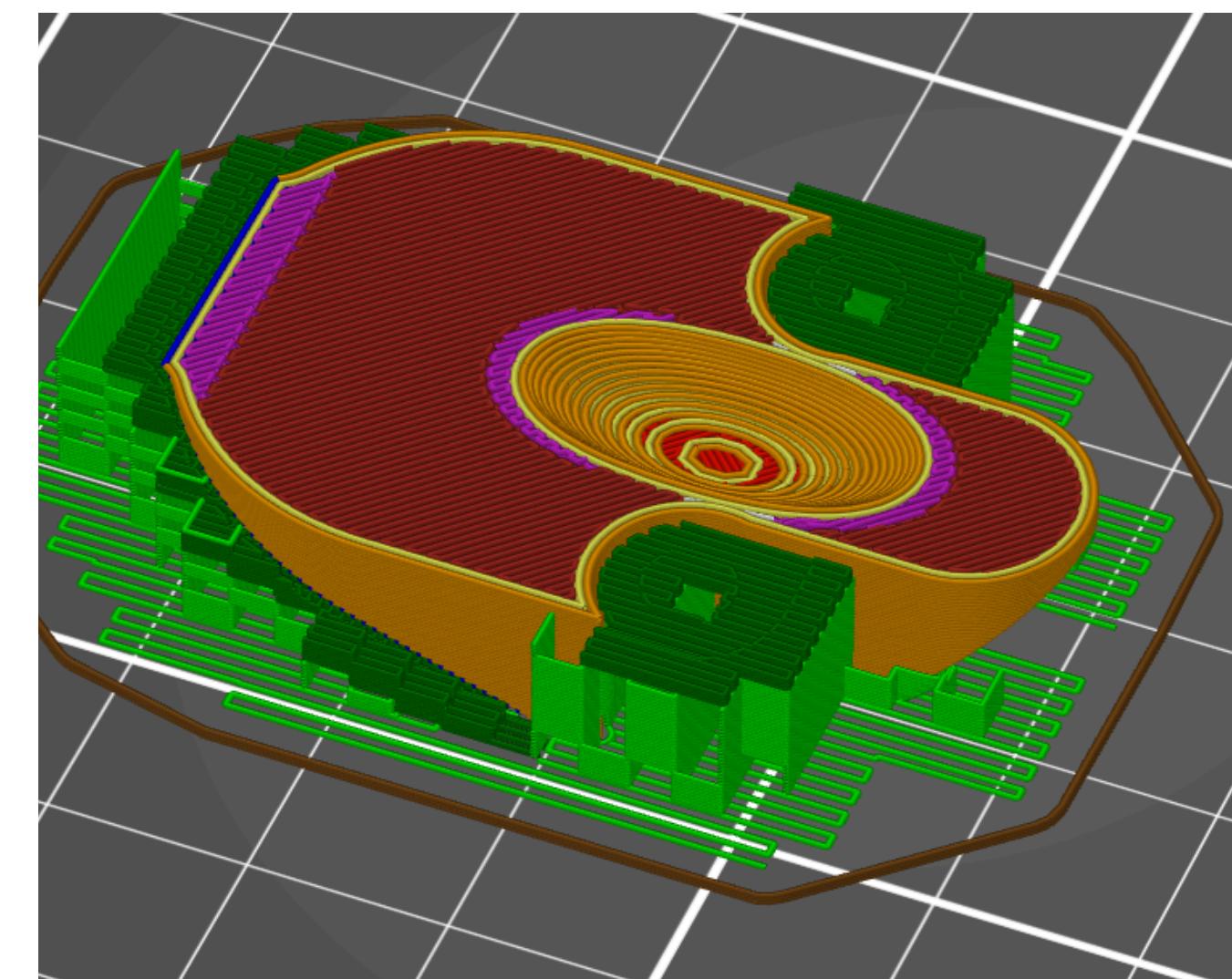
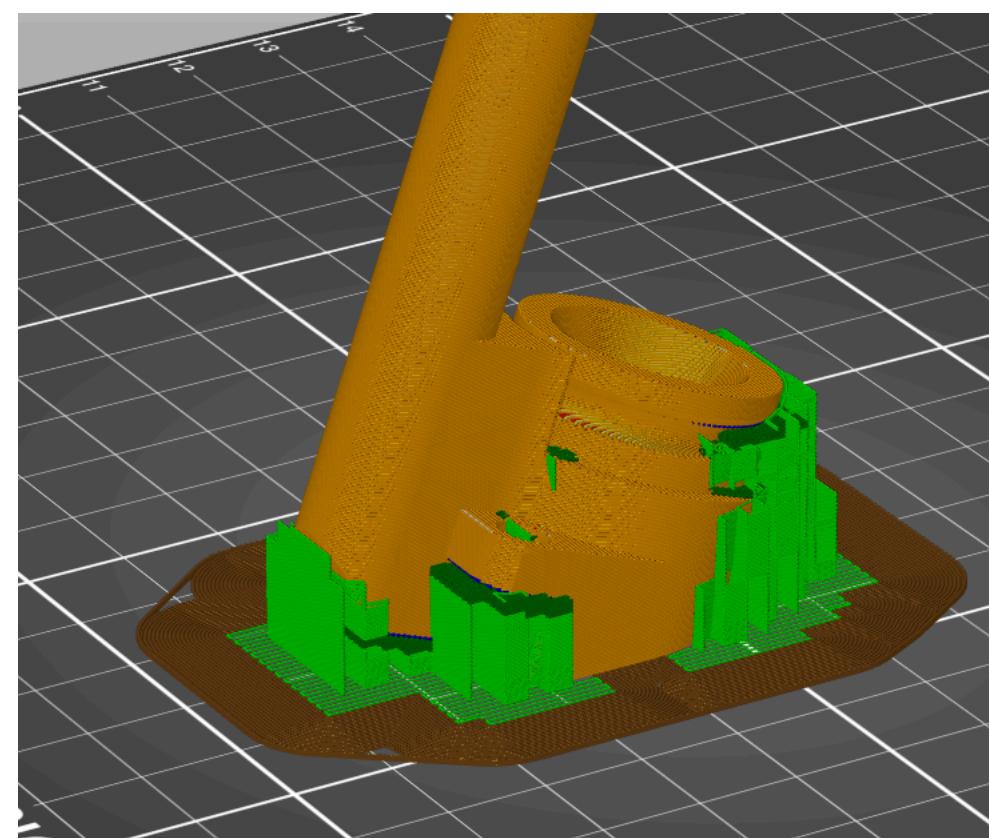
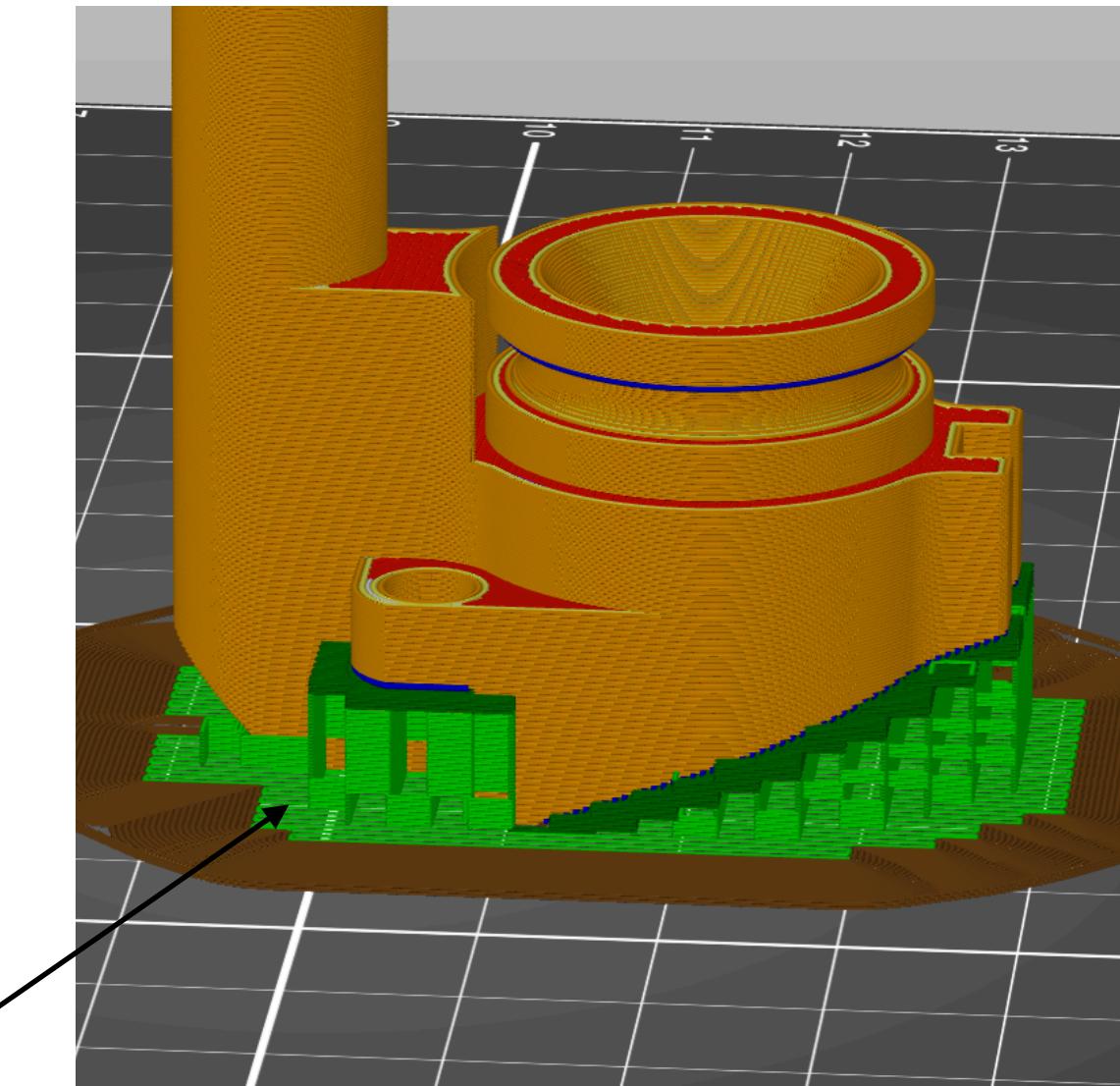
# SLIDE\_FRONT

- 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



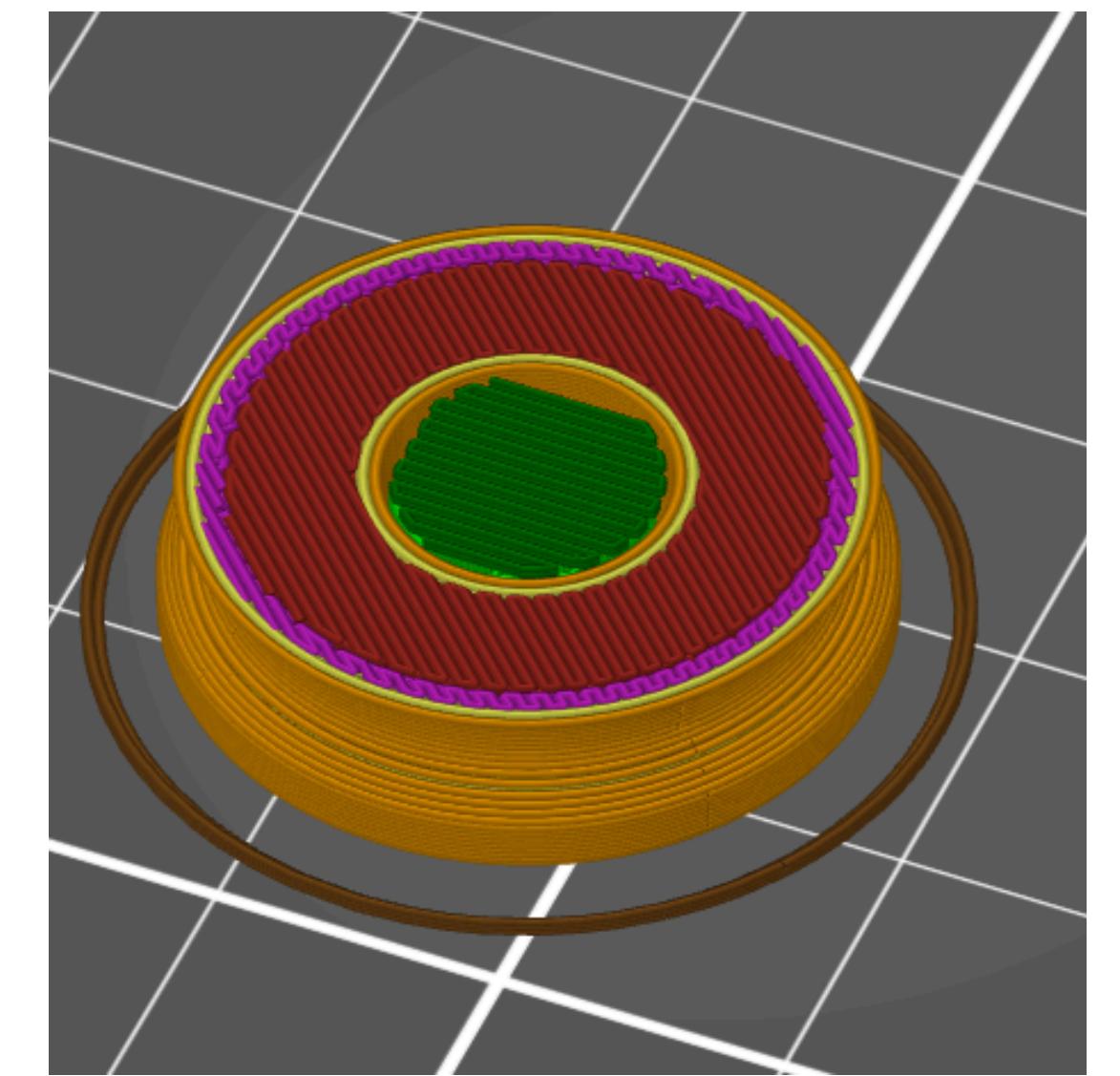
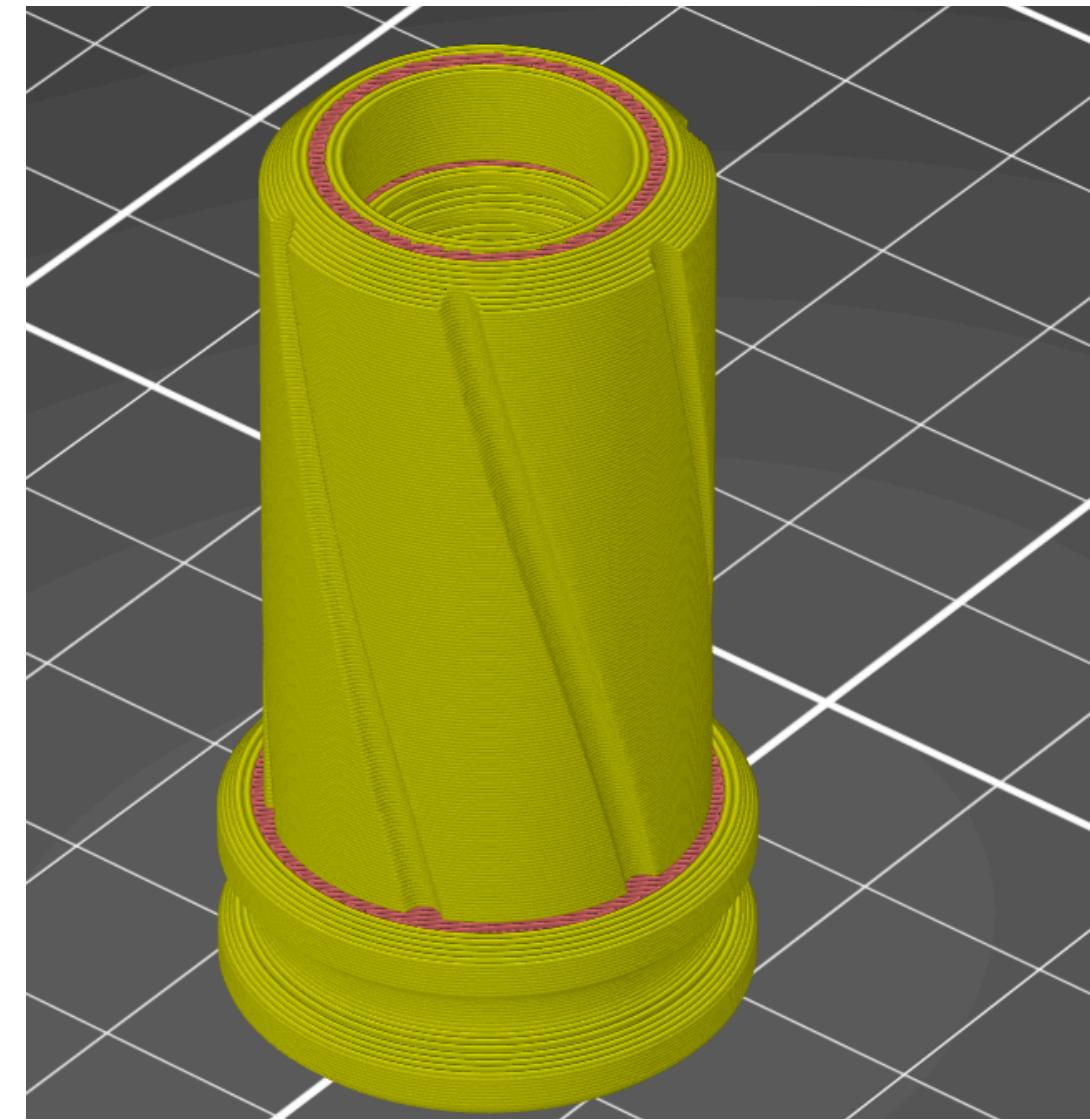
# SLIDE\_REAR\_(any)

- 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 →



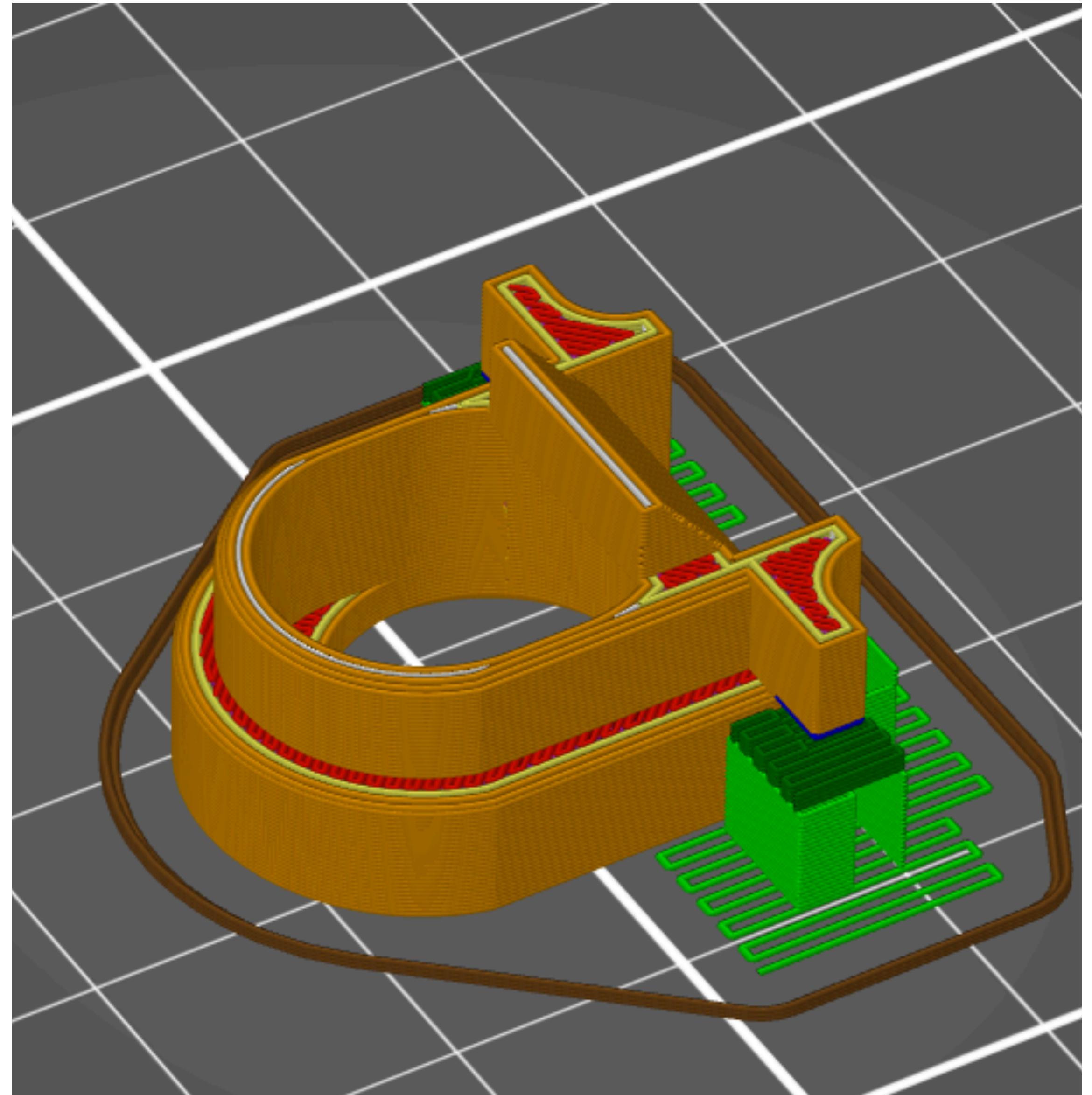
# PLUNGER

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



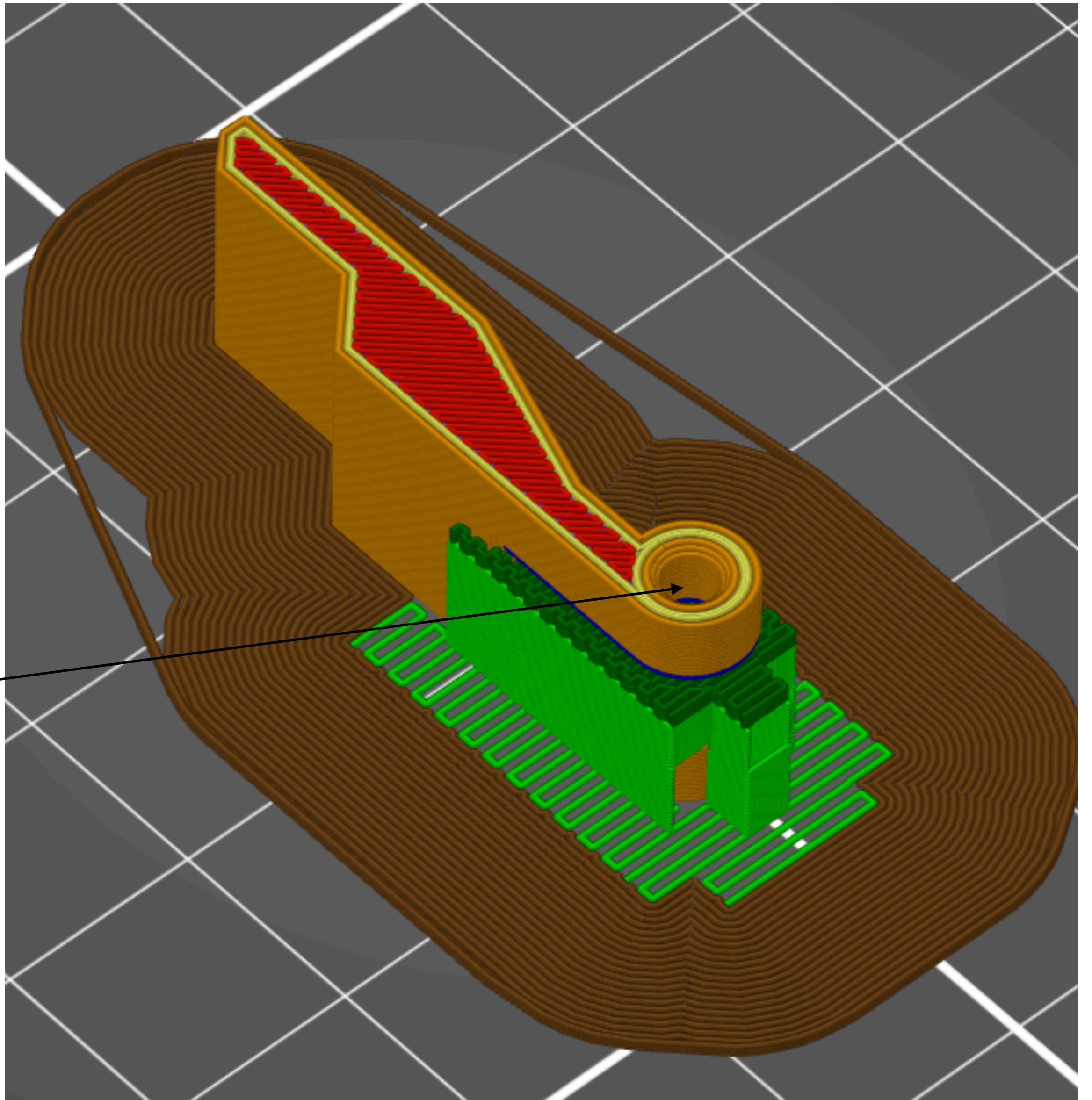
# CATCH

- 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 “elephant’s foot” at the base of prints
- This is a wear part. Print extras and replace periodically.



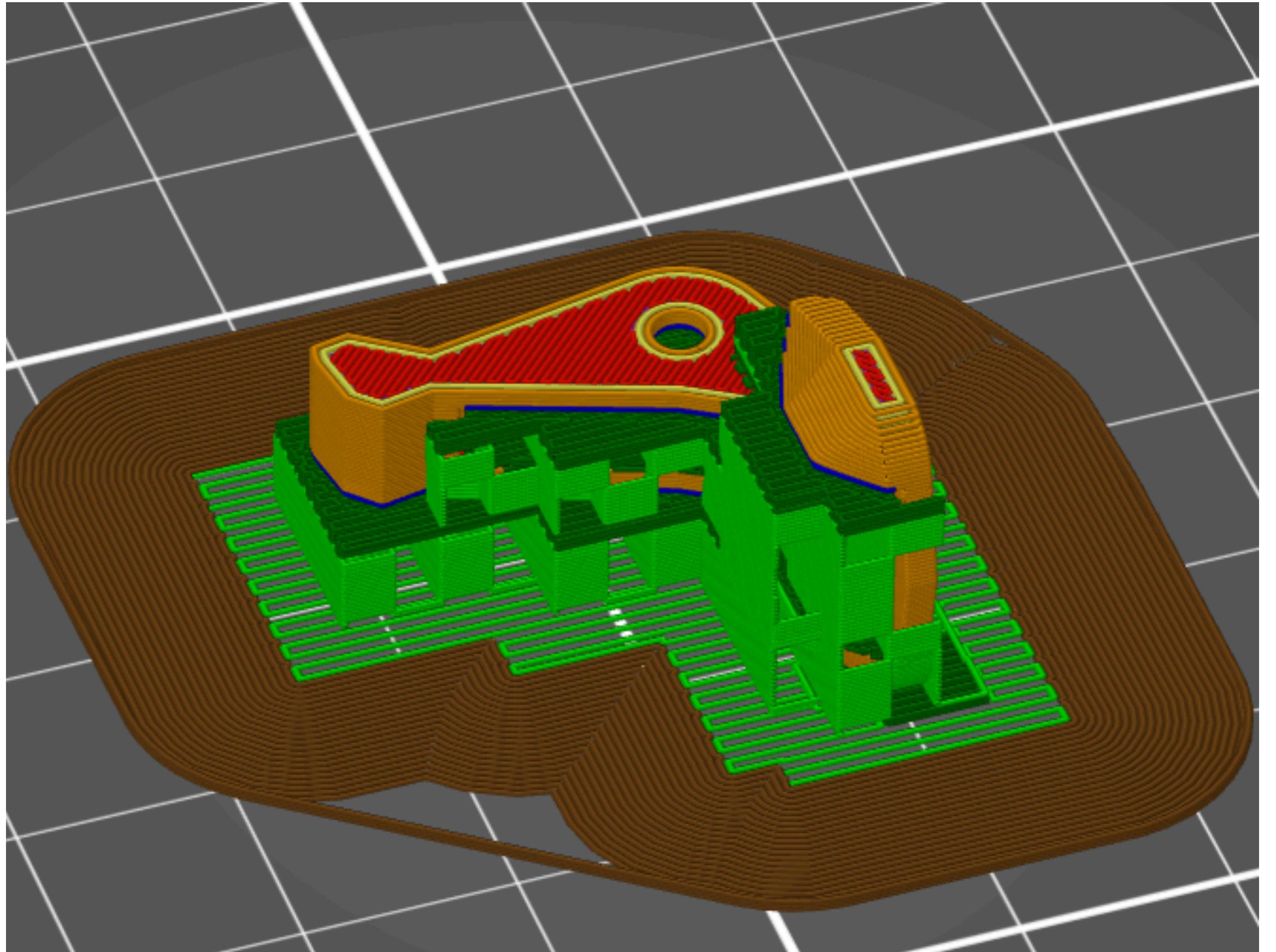
# SEAR

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



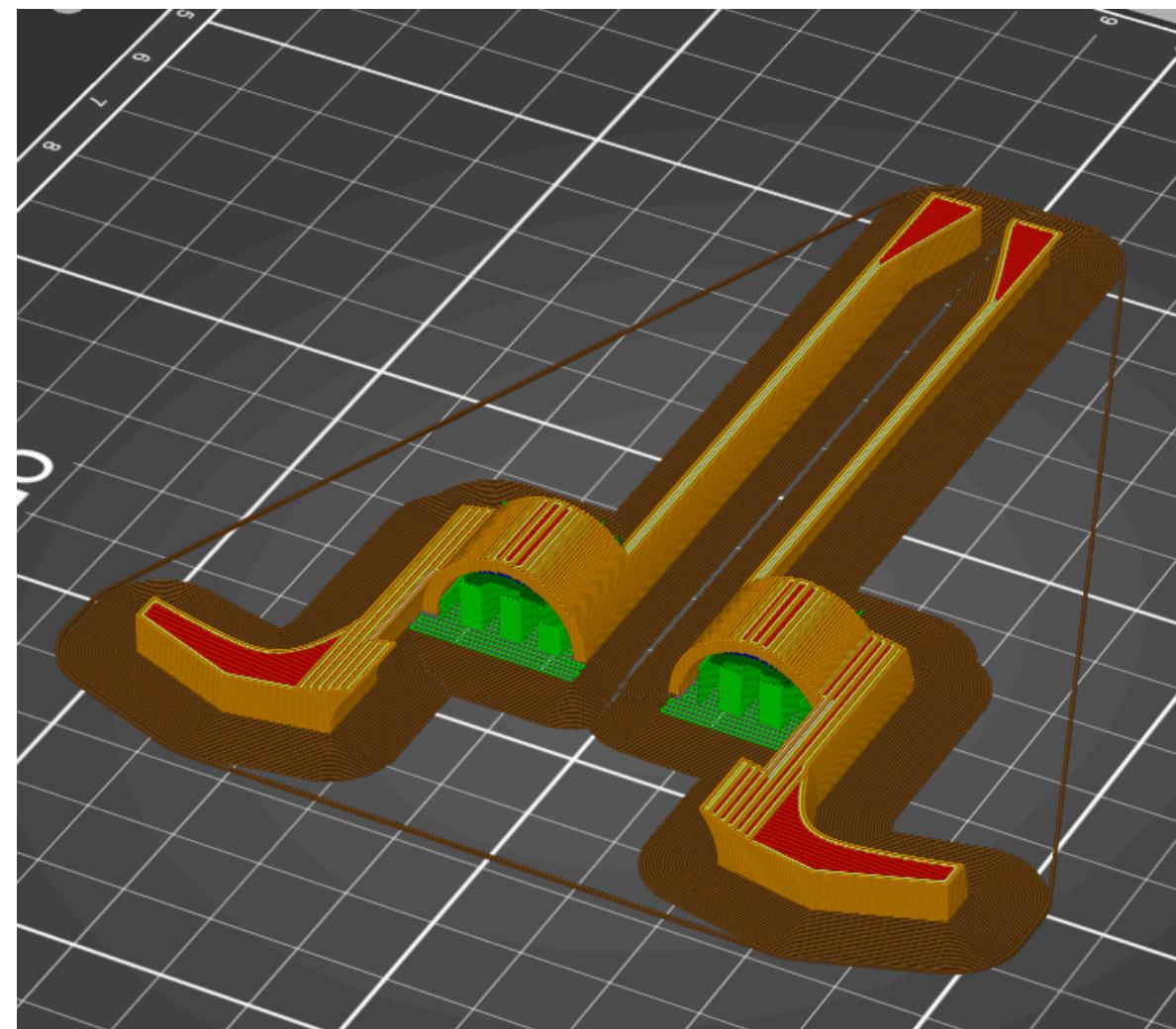
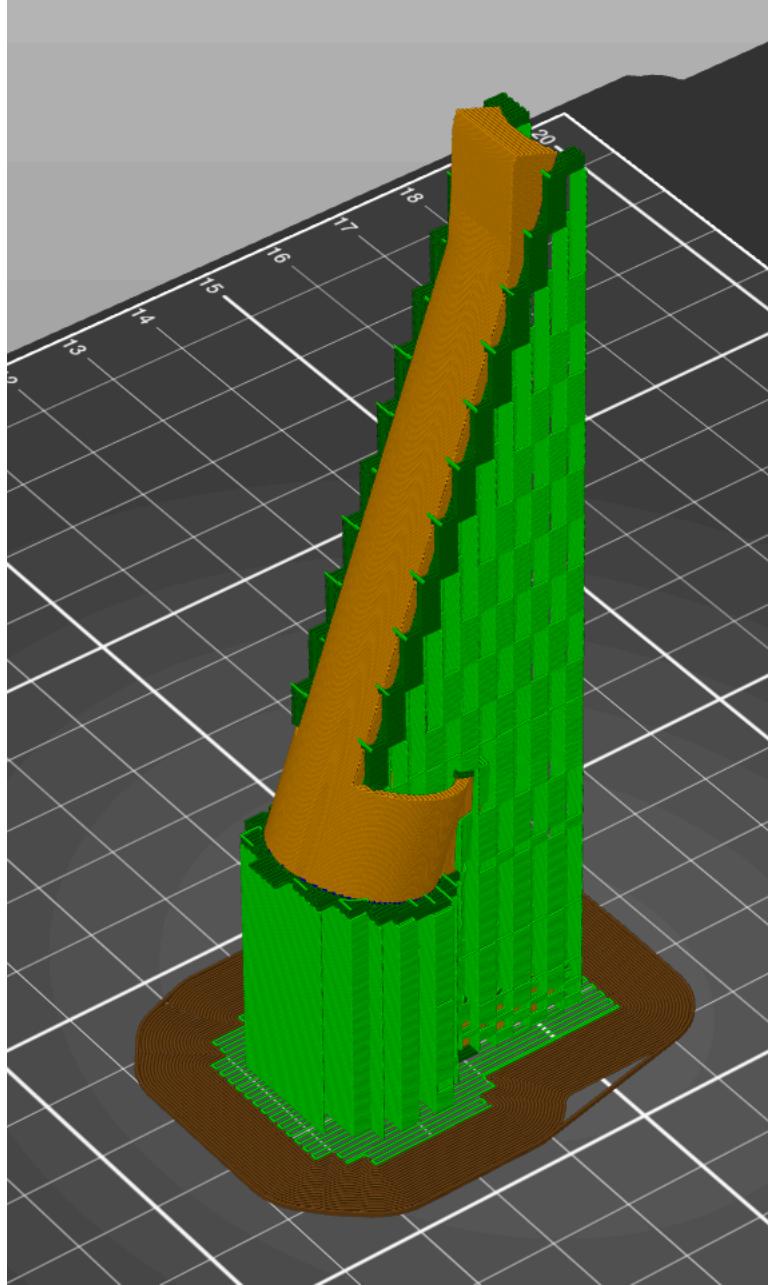
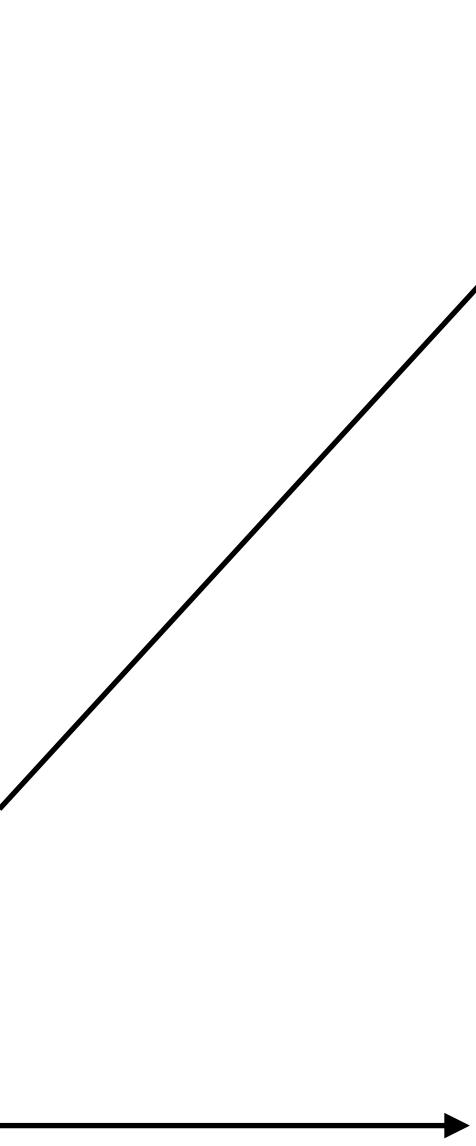
# MAG\_RELEASE

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



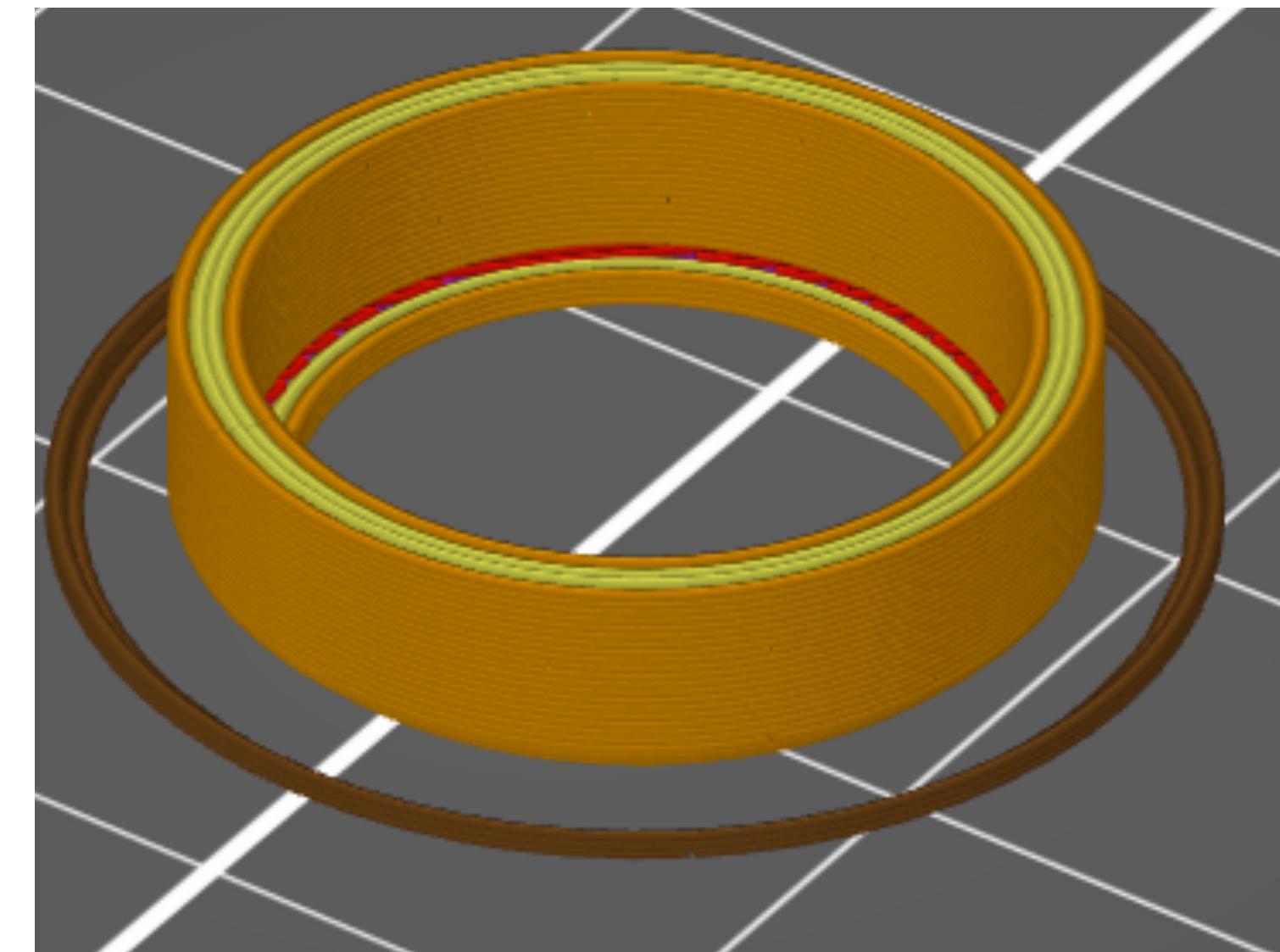
# TRIGGER

- 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



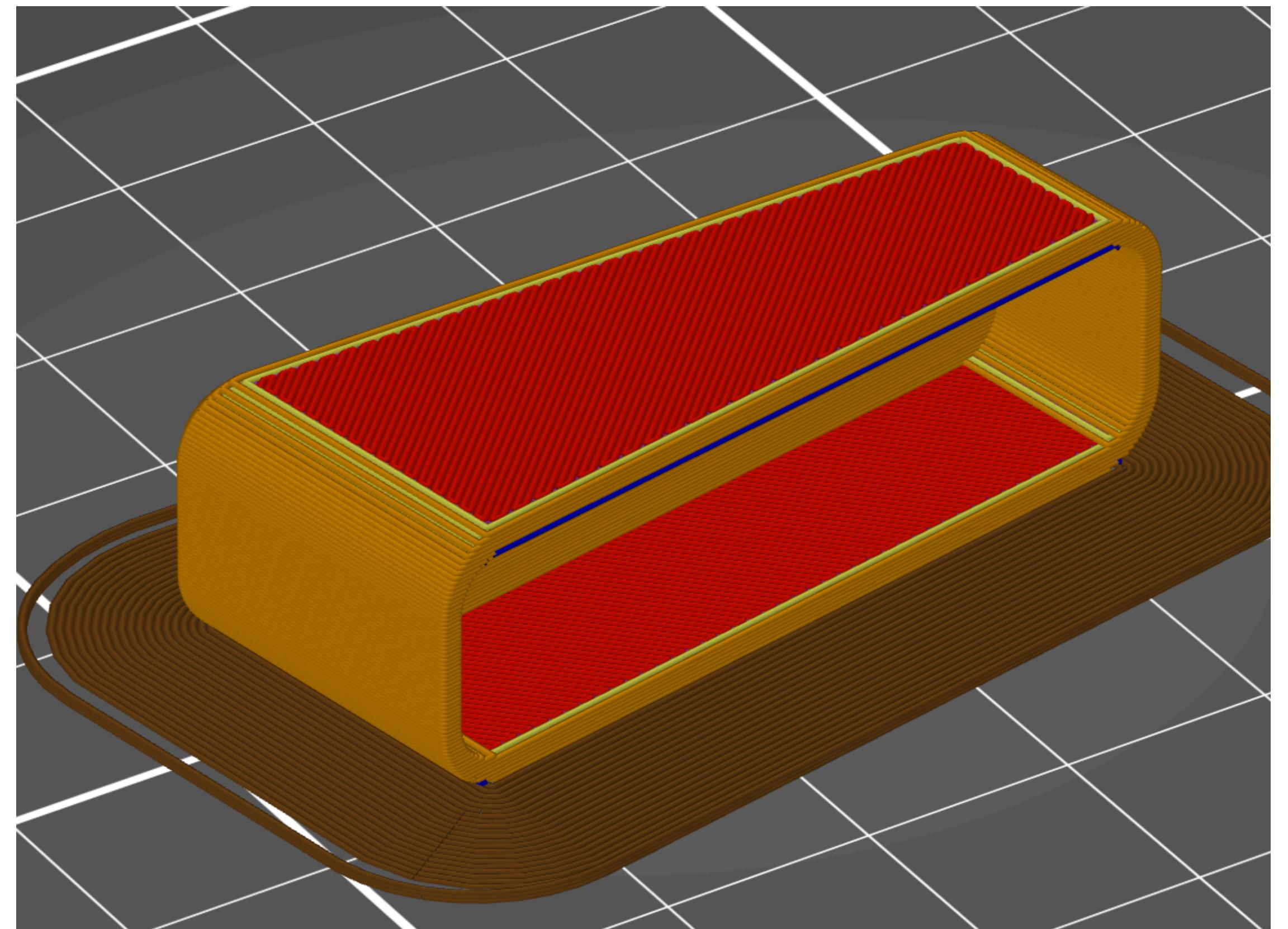
# MUZZLE\_(any)

- Print in a HIGH-VIS filament!



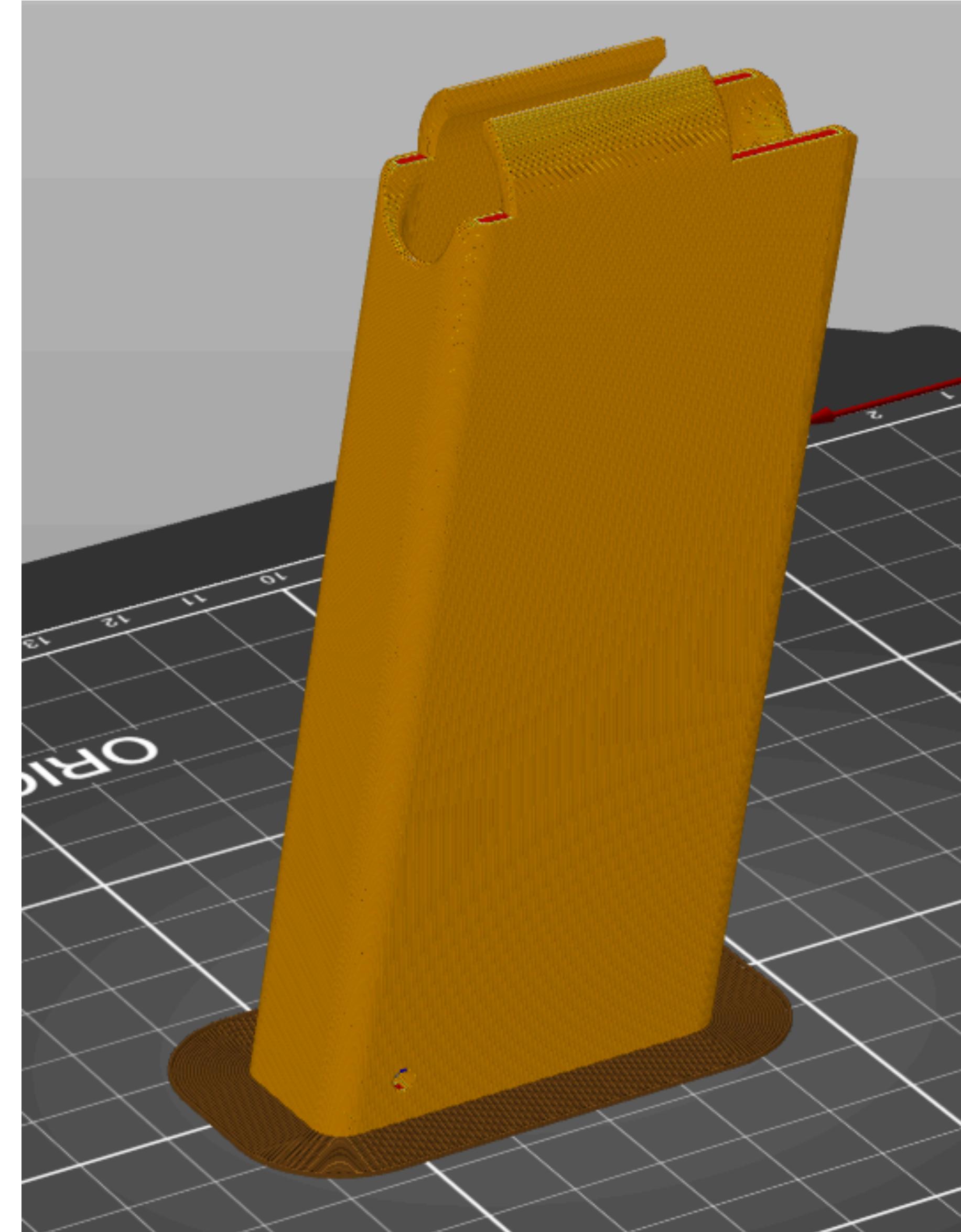
# MAG\_FOLLOWER

- 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

