

# **ZINC 2.0 Print Guide**

**Updated 8/8/2021**

**118 Design**

# **DON'T RUIN THIS FOR EVERYONE.**

Like any toy blaster this one, depending on its appearance & presentation, has the potential to be mistaken for a real firearm. This blaster must be printed in bright colors and used responsibly so that it cannot under ANY circumstances be misconstrued for a deadly weapon. If you fail to build & use this blaster with this in mind you will jeopardize not only your own life, but also the future of this hobby.

# **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.**

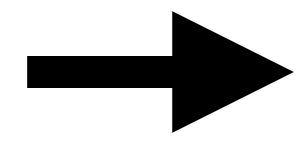
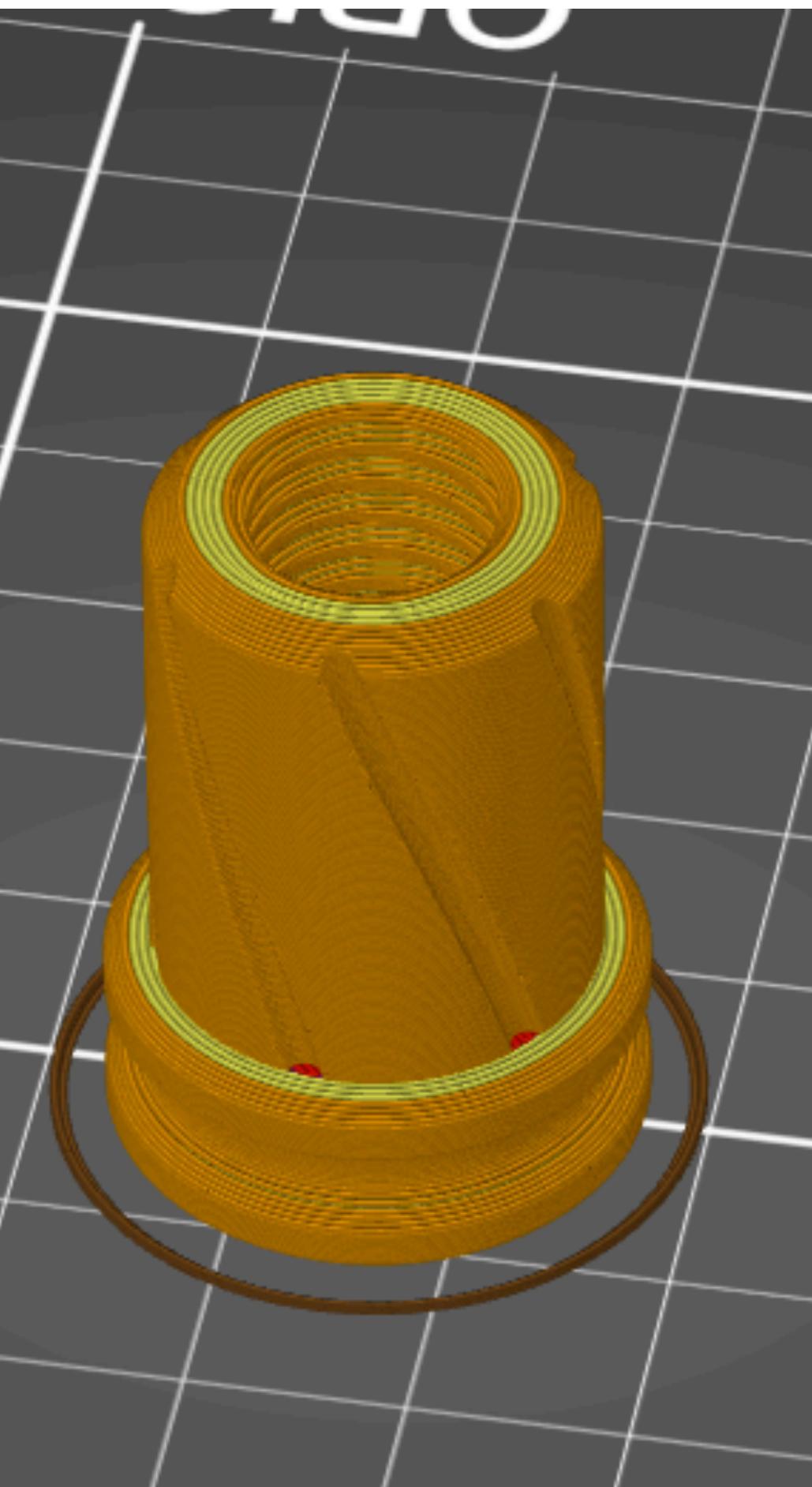
# Introduction

- 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)



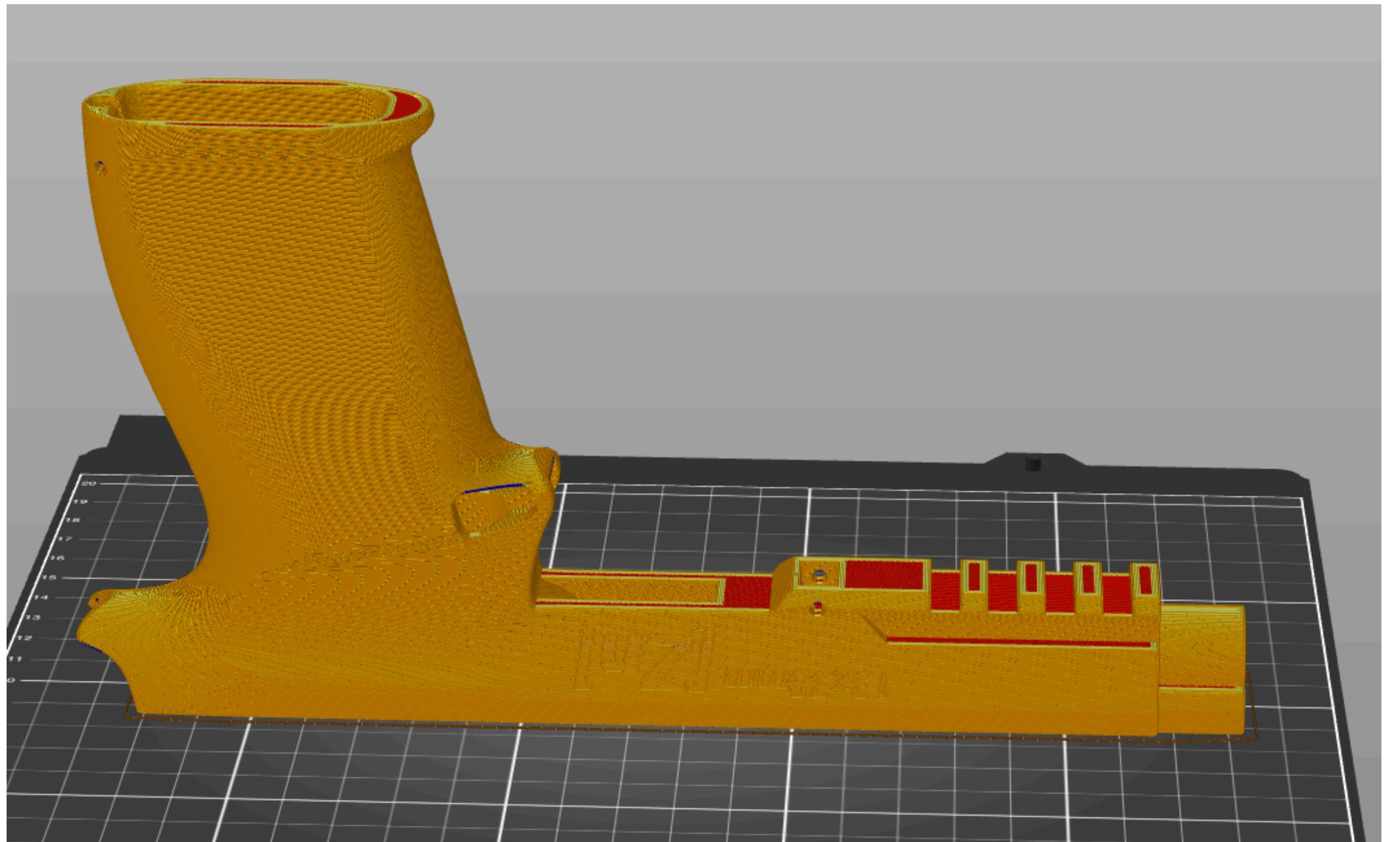
# Calibration

- Begin by printing the plunger (0.15mm layer height, 6 perimeters, no support)
- Verify that you can thread the aluminum plunger core into the printed part. You may need to use a wrench but it should go in smoothly and fully. (*You will need to remove the core and insert an o-ring later! Do this just for calibration, not as a final assembly*)
- If the core doesn't go all the way in, or is excessively tight, then do not print any more parts until you recalibrate. Print a dimensional cube and verify your e-steps
- Do not size up your print to make the core fit! This will destroy the dimensional accuracy of the rest of the parts



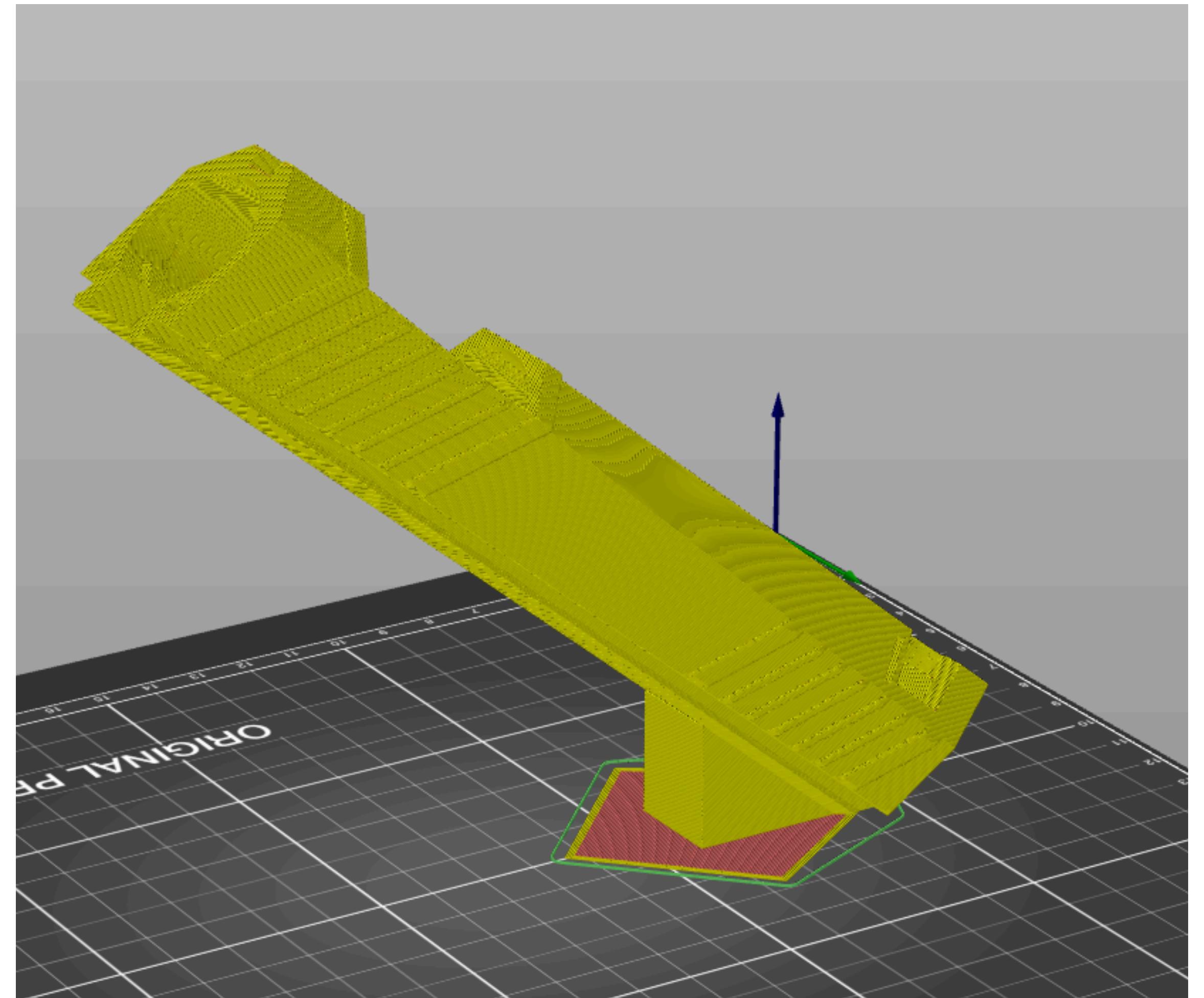
# FRAME

- 0.2mm Layer Height
- 6 Perimeters
- 5 Top, 5 Bottom
- Full infill is ideal
- NO SUPPORT
- NO BRIM (Use one if absolutely necessary but understand it will require careful clean up with an Exacto knife)
- Ensure excellent bed adhesion, lift from the bed will ruin the part. Use an adhesive if necessary
- PETG or PLA+
- Avoid “elephants foot”. If it is present you will need to clean up the rails with an Exacto knife.



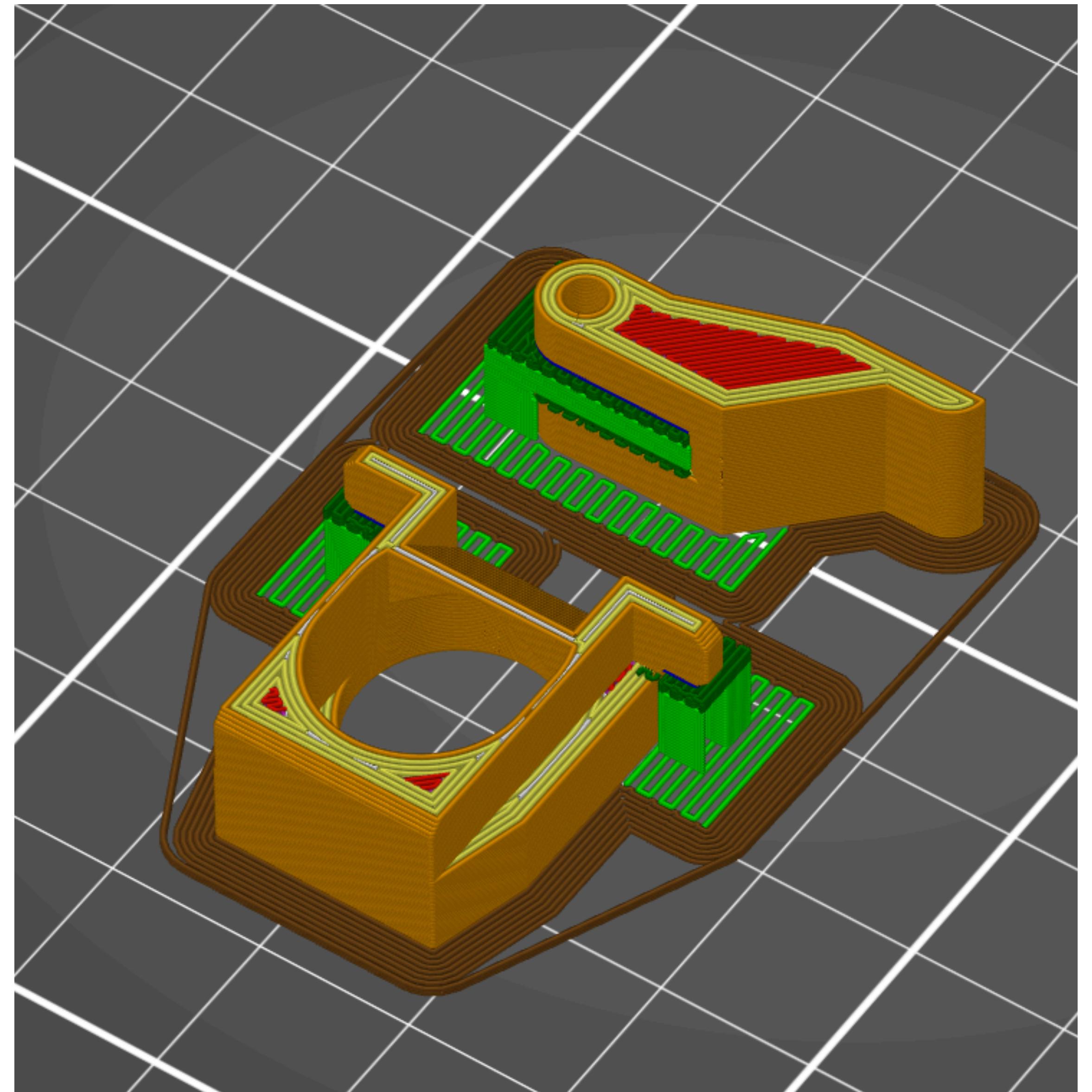
# SLIDE

- 0.2mm Layer Height
- 6 Perimeters
- 5 Top, 5 Bottom
- 100% infill
- No Supports
- PETG recommended. This part needs to flex to install the plunger tube.
- Ensure there is no warping: *print slowly & be mindful of how your fan shroud is oriented.*



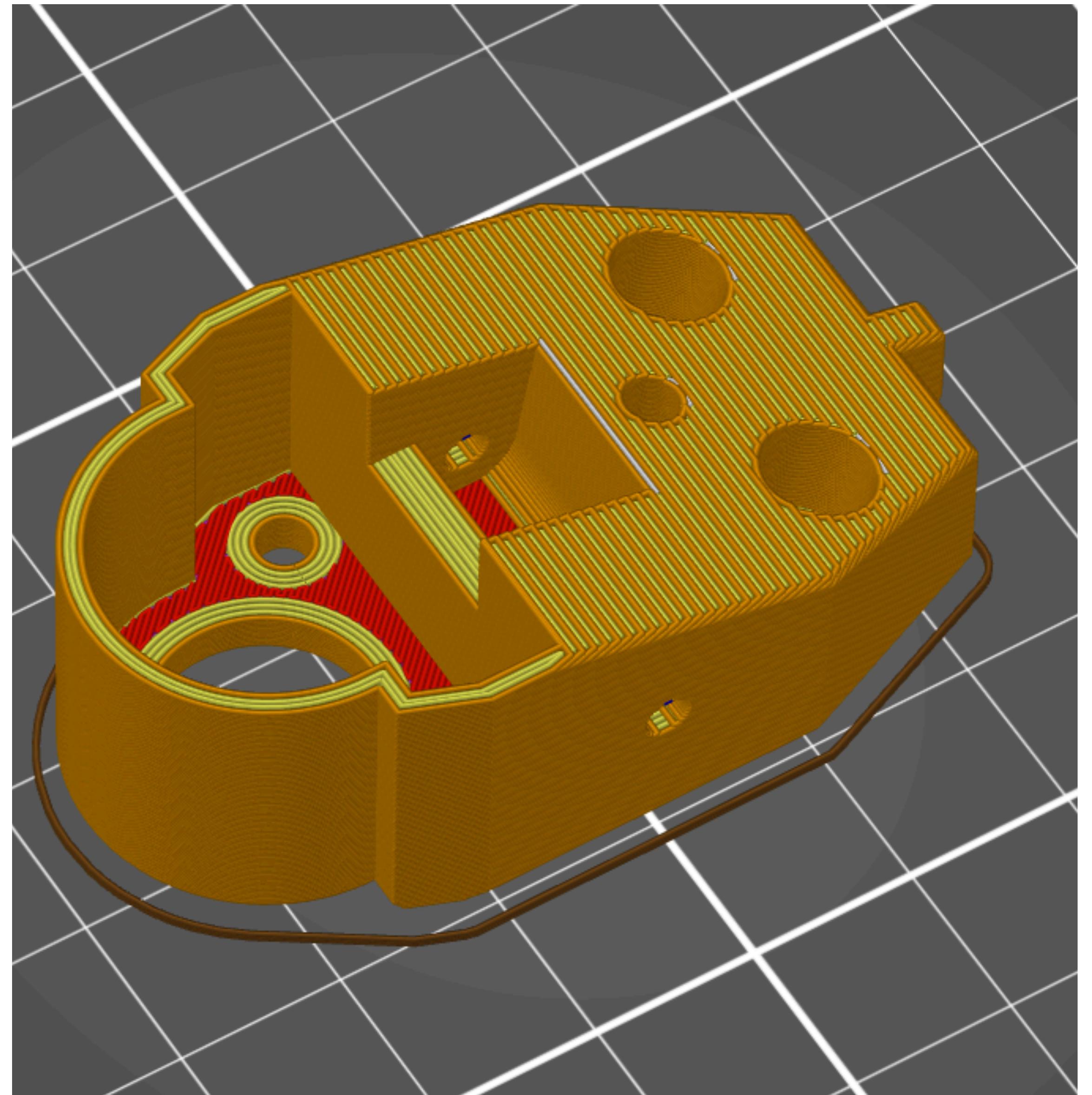
# CATCH & SEAR

- 0.2mm Layer height
- 4 perimeters
- 100% infill
- Print with support as shown.
- PLA+ or PETG



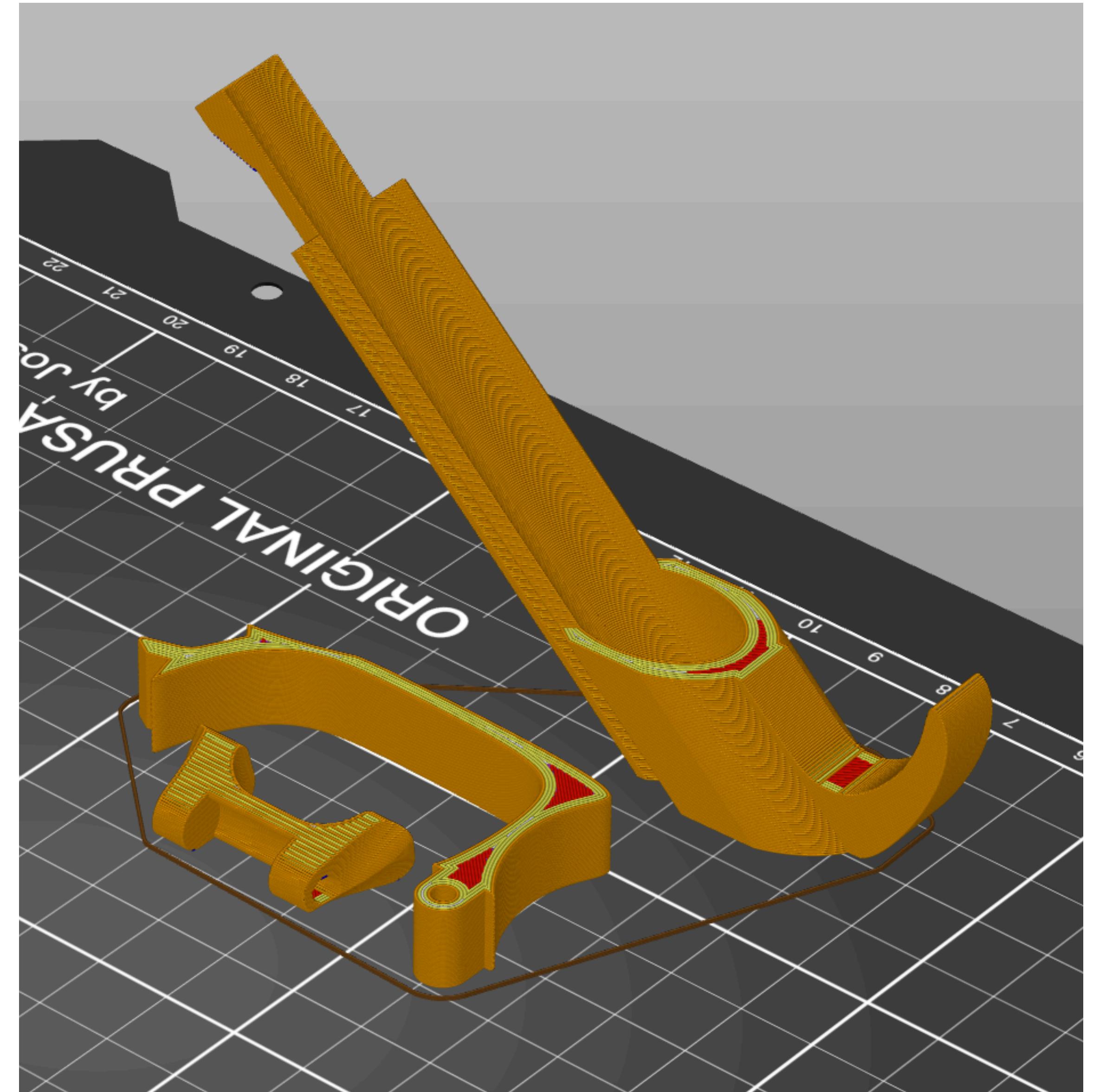
# FRONT

- PRINT IN HIGH-VIS ORANGE.
- 0.2mm layer height
- 4 perimeters
- No supports
- 5 top, 5 bottom



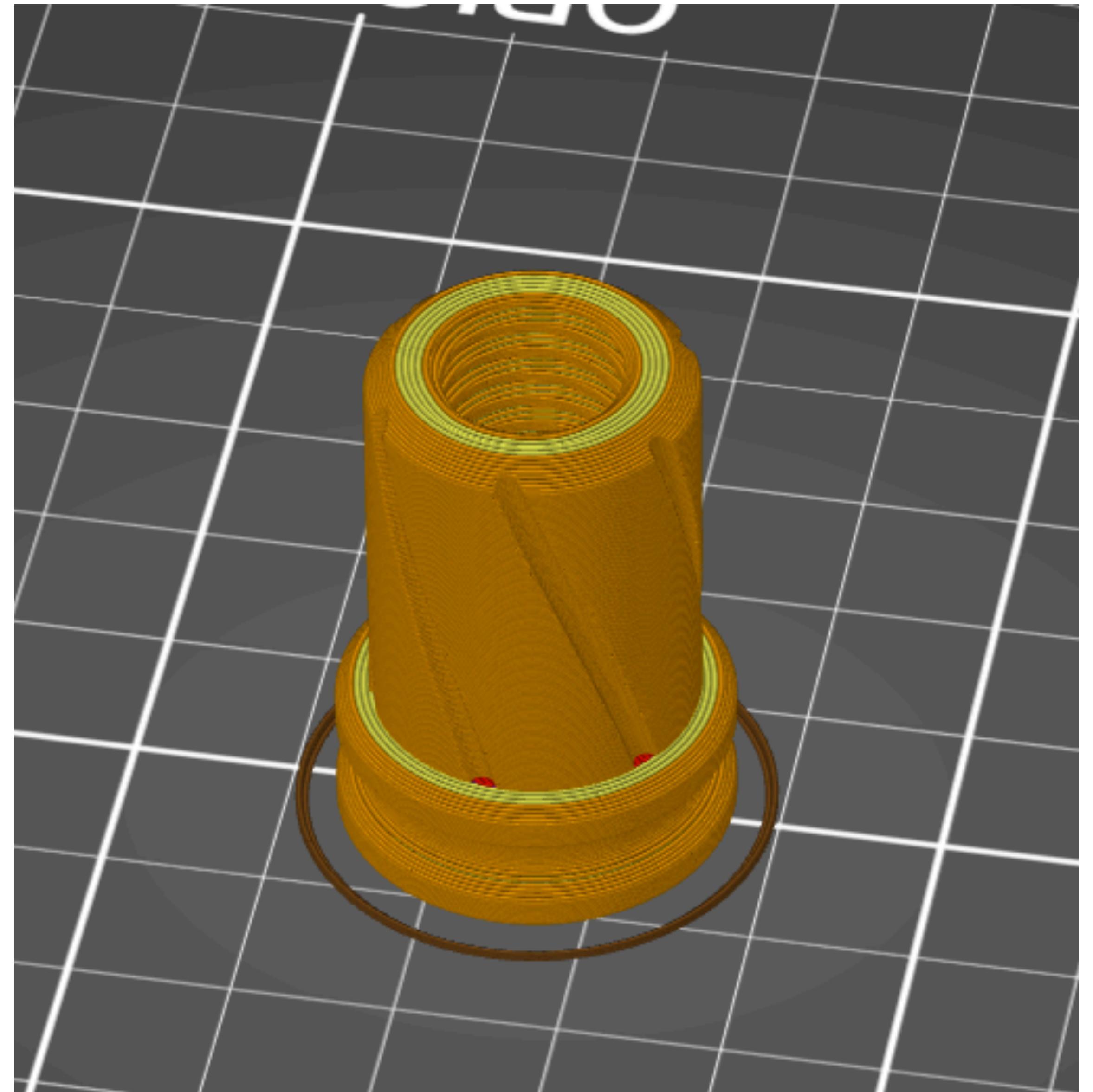
# TRIGGER, RELEASE, GUARD

- Print the “RIGHT” variations of GUARD and RELEASE if you are right handed. Same for left handed.
- 0.2mm layer height (print RELEASE at 0.1mm layer height for best resolution)
- 6 perimeters
- Try to do this without a brim. There will be a lot of cleanup if you need to use one.



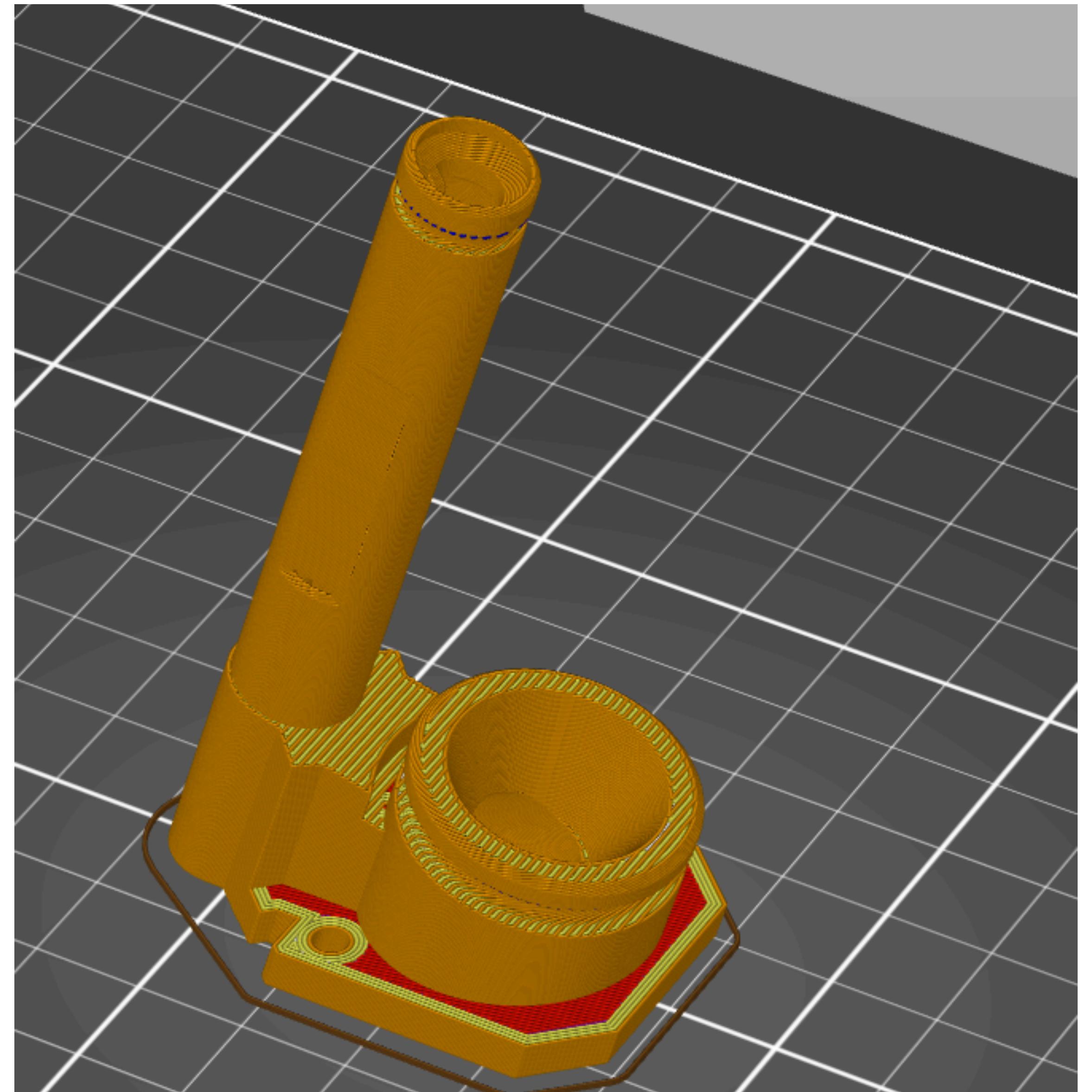
# PLUNGER

- 2 variations, high & low power.
- 0.15mm layer height
- No supports
- 6 perimeters
- If you can't thread the aluminum plunger core into the printed part you need to recalibrate your printer.



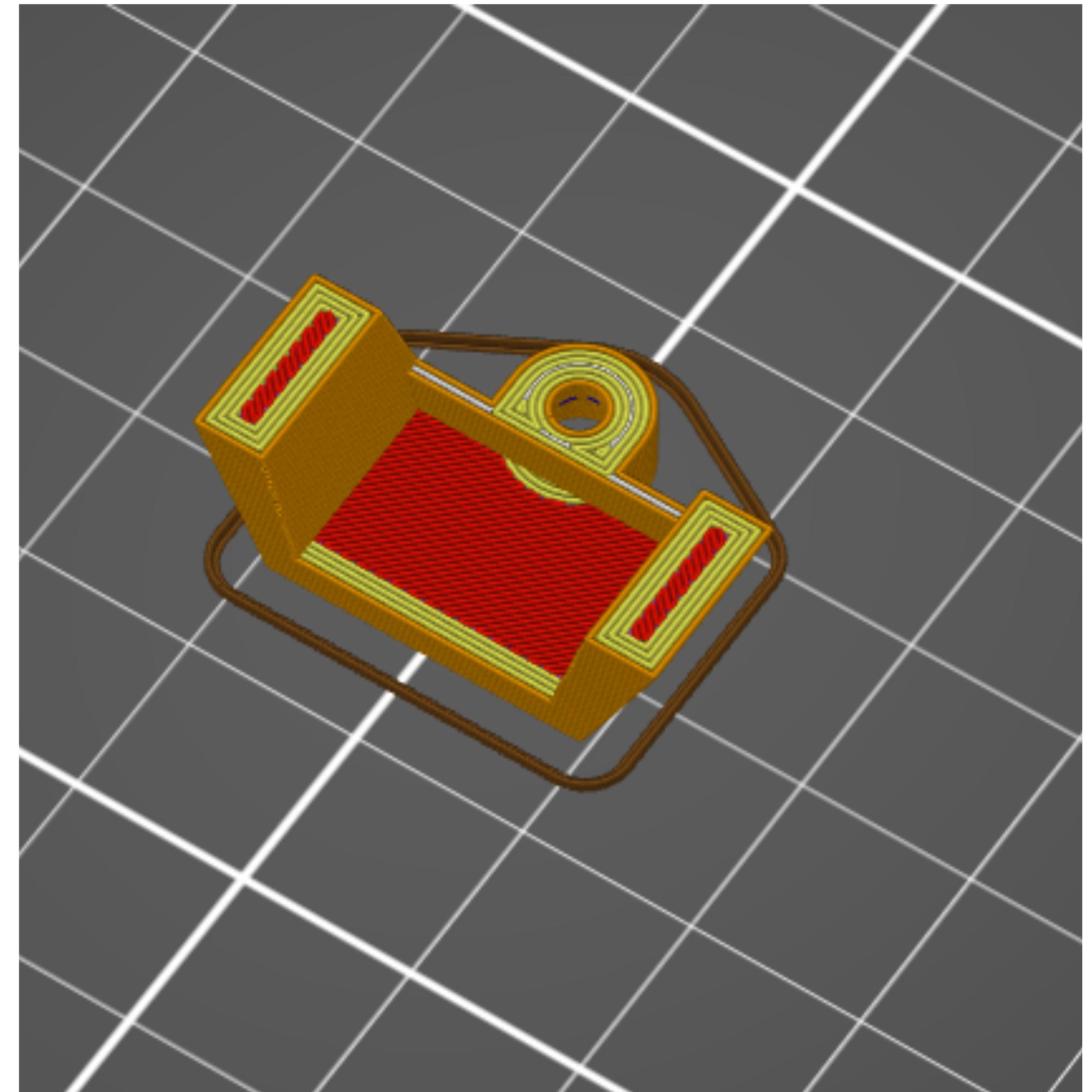
# REAR

- 0.2mm Layer height
- 6 perimeters
- No supports



# COVER

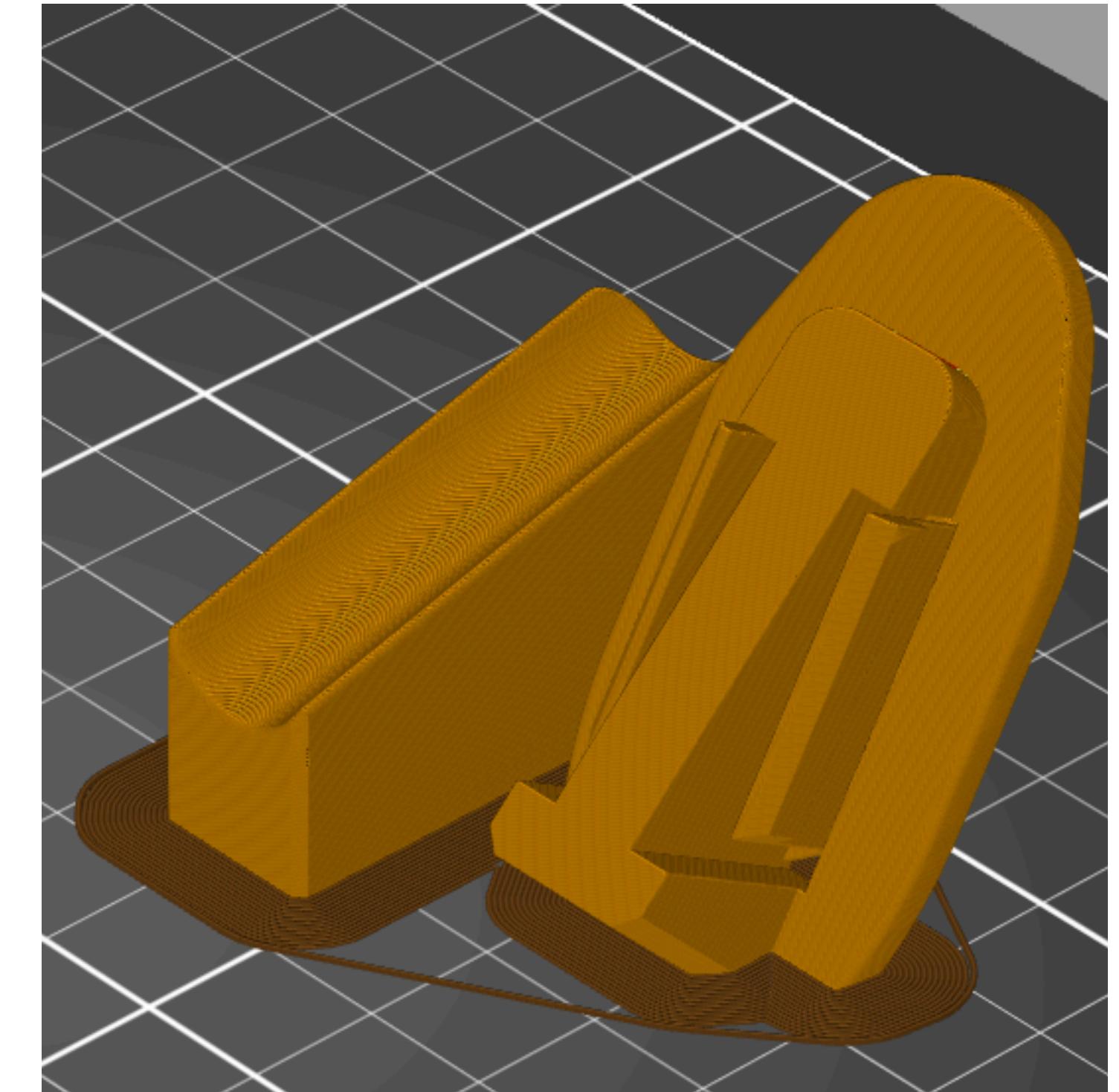
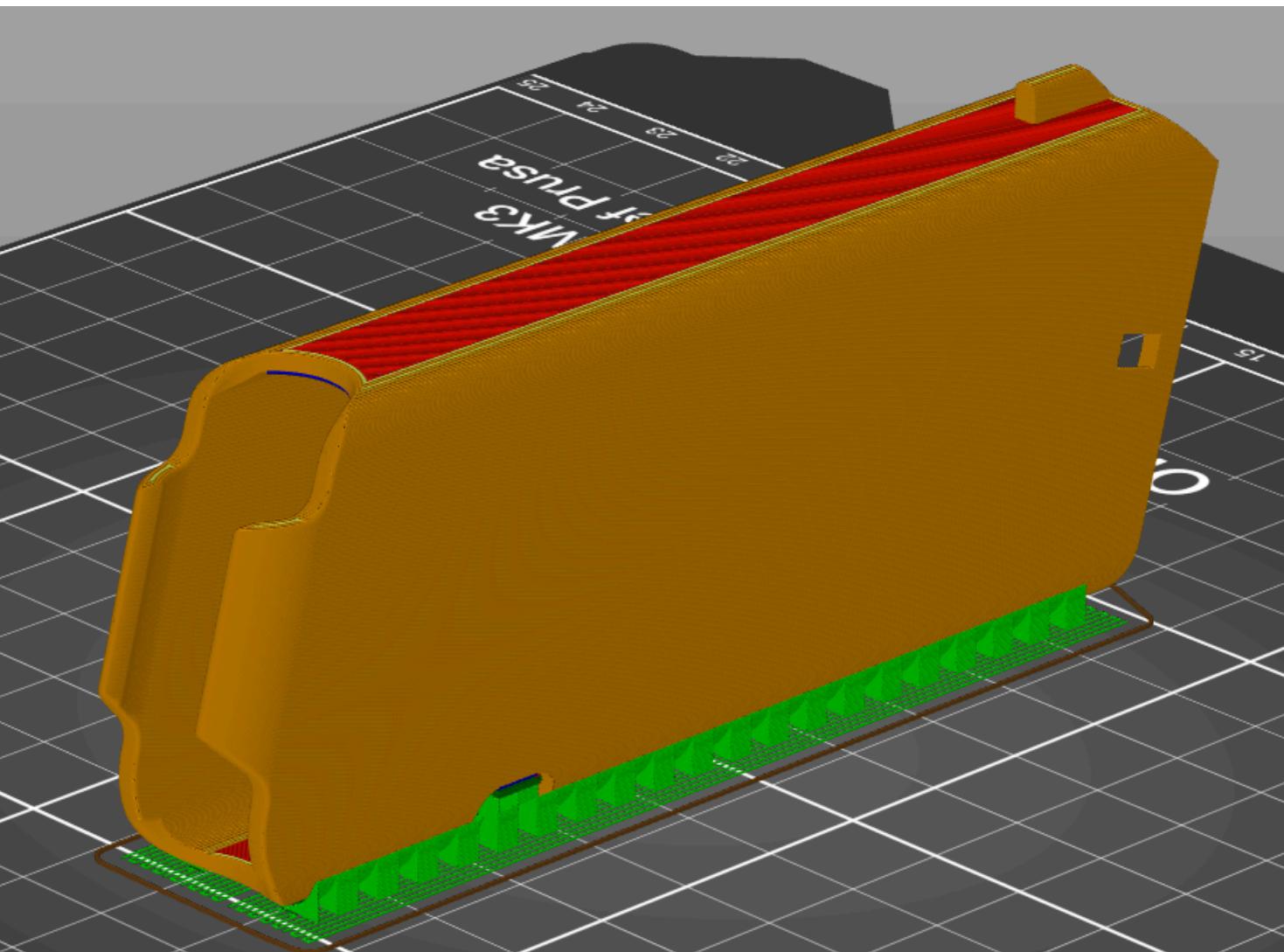
- Just print it.



# MAGAZINE

## Option A

- 0.1mm Layer Height
  - 3 Perimeters
  - Brim for adhesion
- 
- This orientation can be challenging to print. Make sure your bridging and overhangs are dialed in. Go to the next page for an alternative option.



# MAGAZINE

## Option B

- 0.1mm Layer Height
  - 3 Perimeters
  - Brim for adhesion
- 
- This orientation will require you to sand the magazine well to get rid of the noise.

