

# HarrysAquatics FragStation™ Print Instructions:

## General Print Settings:

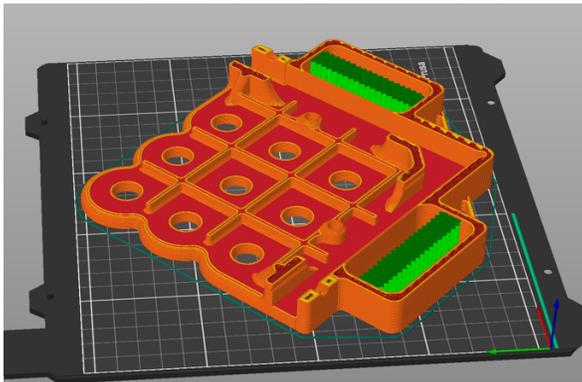
- 0.2mm or 0.3mm Layer Height
- PLA or PETG recommended.
- .4 mm or .6 mm nozzle.
- 2 perimeters
- 30% general infill
- Do **NOT** enable general support.

## Postprocessing:

- Snap out support blocks with screwdriver.
- Use M6x1.00mm Tap to create smooth threads.

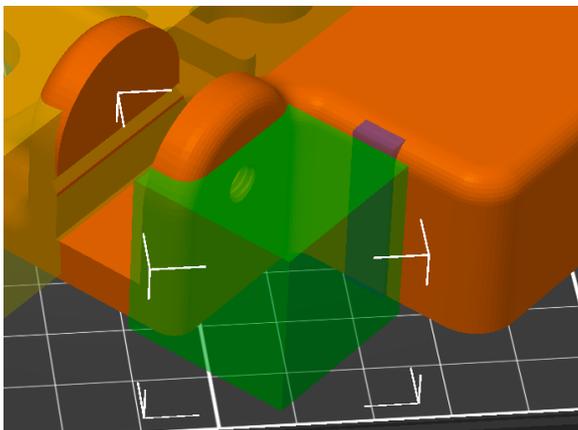
## Custom Support Area:

Use .25" wide **Custom Support Block** in center of glue pocket. Use GRID style support for easy 1 piece removal.



## Custom Infill/Perimeters Area:

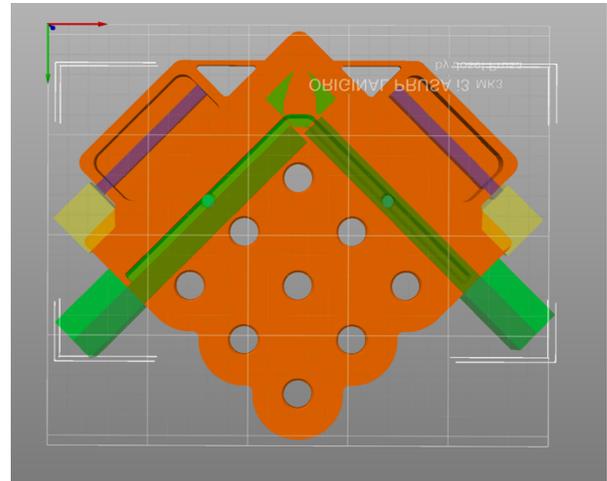
Reinforce **both** screw tabs with 1x1x1 modifier of 80% infill, 4 perimeters. This prevents the screw from snapping the tab (without wasting 80% infill on the entire model).



## Custom Bridging Directions:

The water drainage slot requires small amount of bridging. In many slicers it does not default to the shortest, perpendicular bridging layout.

Example (Green) Modifier Blocks (45° Bridging, 90° Bridging, 135° Bridging).



Resulting Bridging Angle from correct modifier blocks:

