

HEAD (AMAME)

V2 Slicing Guide

v1.0 July 7/2023



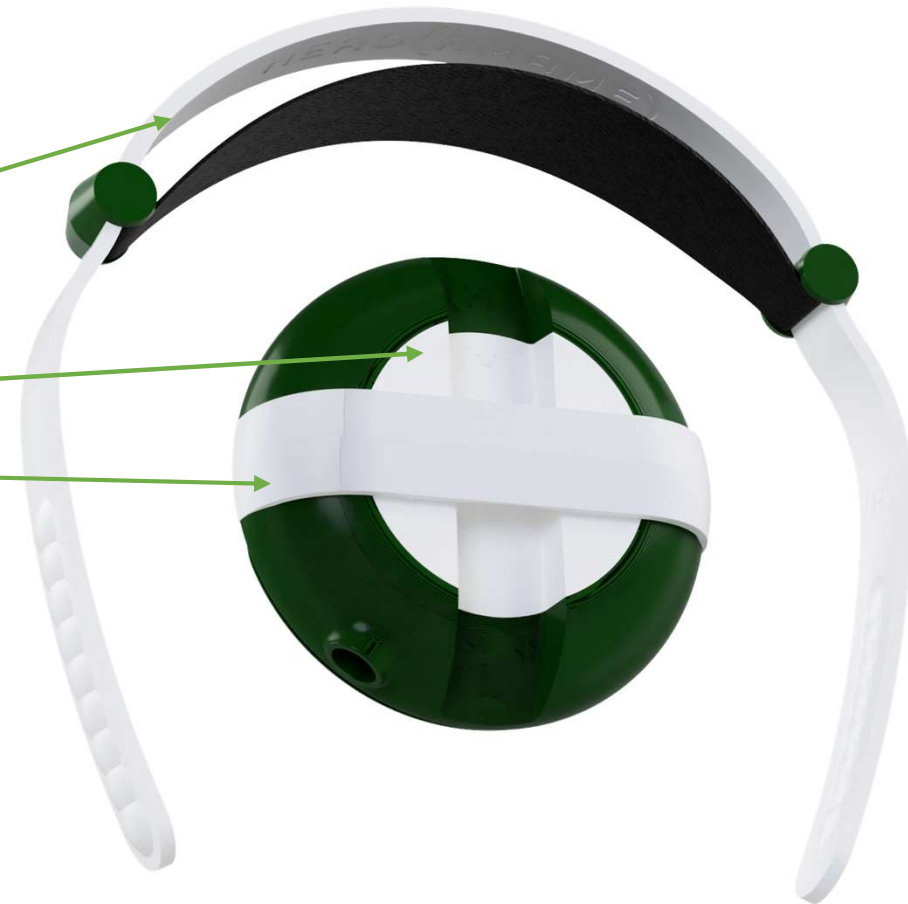
(H) General Notes

Some head(amame) parts need to be stronger than others.

Headband Spring
Headband Mount
Spring Clamp

White parts should be printed in a stronger filament such as PETG, ABS, or ASA to keep them from “relaxing” over time.

The rest of the parts can be printed in nearly any filament!

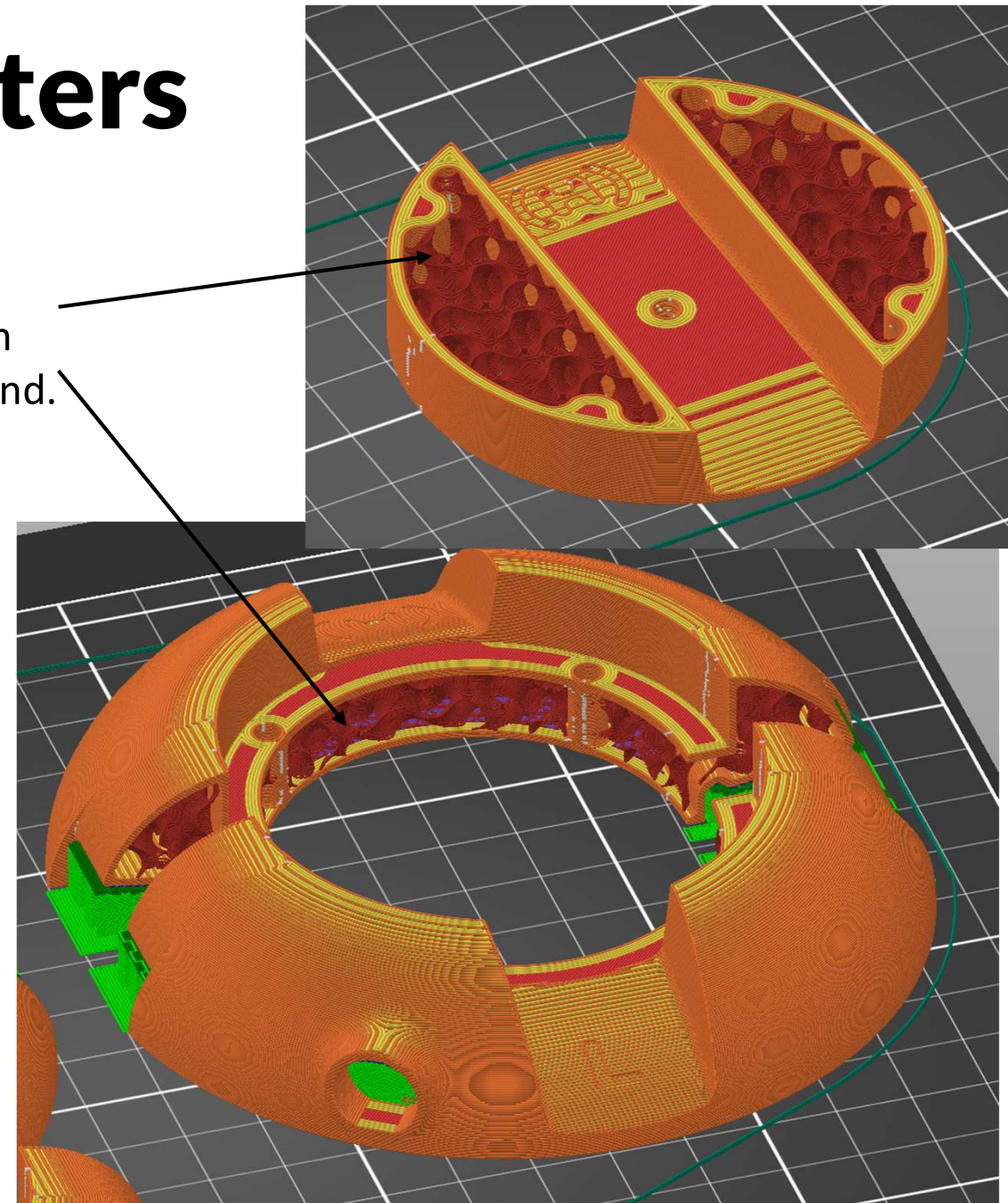
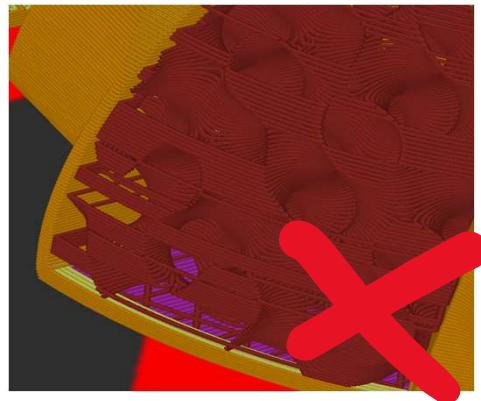
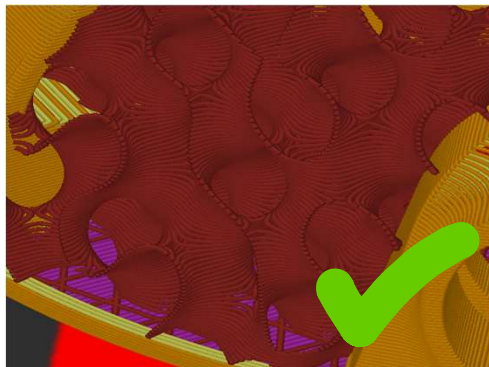


(H) Why Gyroid Matters

head(ame) uses an 11% gyroid infill in the Cones and Headband Mounts to improve sound quality and look cool as hell. They cut down on reflections on the back side of the driver that may distort your sound.

These areas are represented by “infill bodies” to trick your slicer into generating the infill properly.

Slicers try to “anchor” the infill to the other bodies, causing the infill to have extra lines we don’t want. Each software has a different way to anchor, and we will show you how to avoid them.



(H) General Slicing Notes

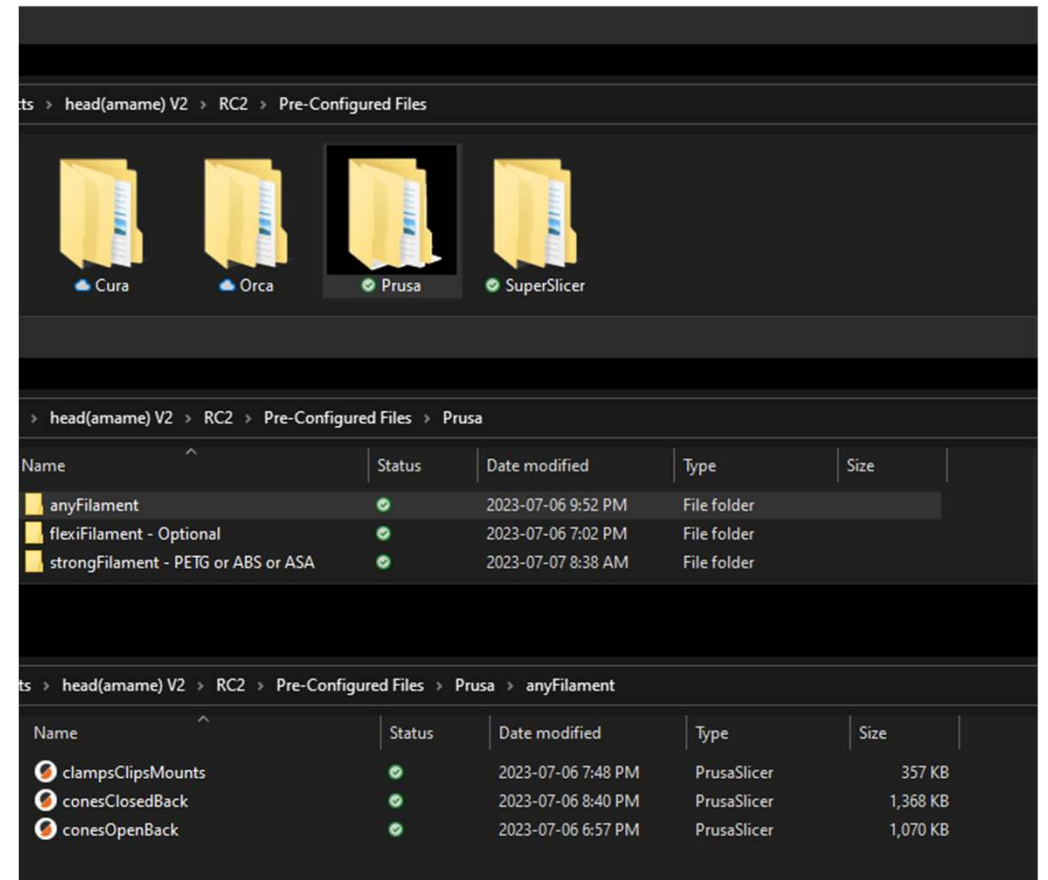
Area	Setting	Comments
Layer Height	0.2mm	General good layer height for all parts
Nozzle Size	0.4mm	No benefit from going smaller, larger
Build Plate Adhesion	User Choice	Brim and Raft are good if you are having adhesion issues
Wall thickness	2.0mm or more	On most walls, this will cause no infill.
Top/Bottom Thickness	2.0mm	This makes sure you have solid shells around infill bodies
Infill	11% on infill bodies, 100% all else	Gyroid cones/headband mounts, Rectilinear all else
Maximum length of infill anchor (PrusaSlicer and OrcaSlicer)	0 (not anchored)	Removes infill anchors – giving open-celled gyroid
Connect infill lines (Cura only)	Unchecked	Removes infill anchors – giving open-celled gyroid
Sparse Infill – (SuperSlicer)	Connected to hole perimeters	Removes infill anchors – giving open-celled gyroid

(H) Pre-Sliced models

Slicing might be the most annoying part of 3D Printing so I've pre-arranged and configured files for you!

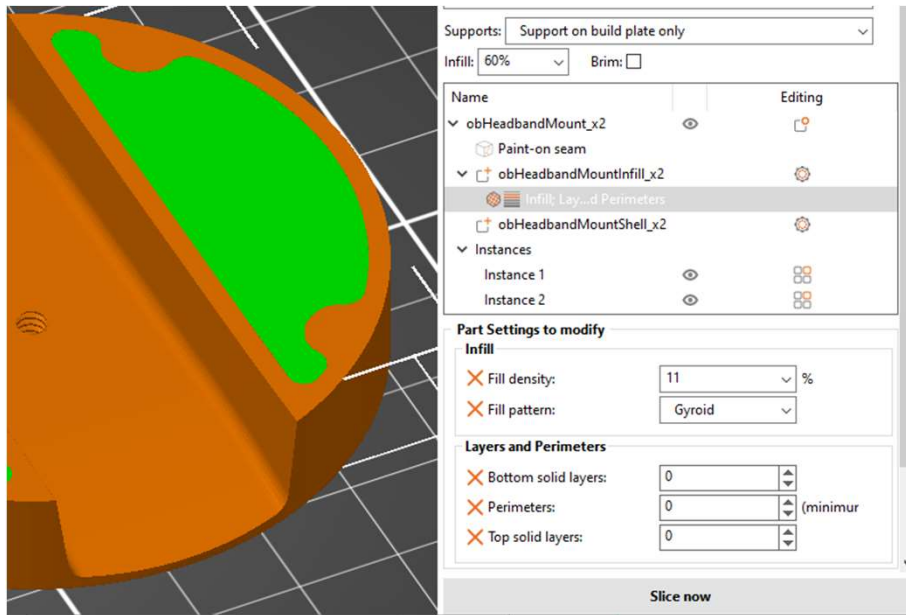
If you import the pre-arranged and sliced files into your slicer, you will still need to:

- Read this guide for slicer-specific checks
- Choose your machine and arrange for your build plate
- Tune your filament flow for top quality
- Make sure that infill looks correct before printing

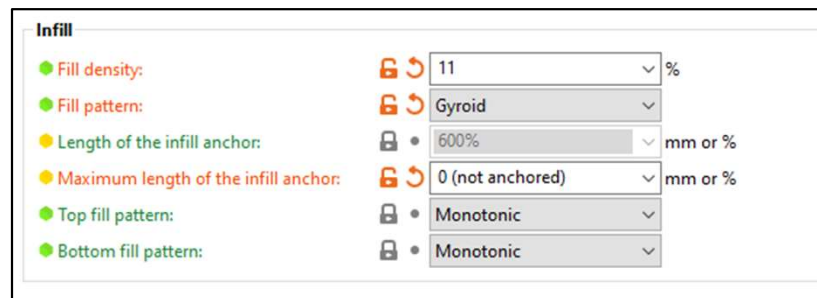


(H) PrusaSlicer Settings

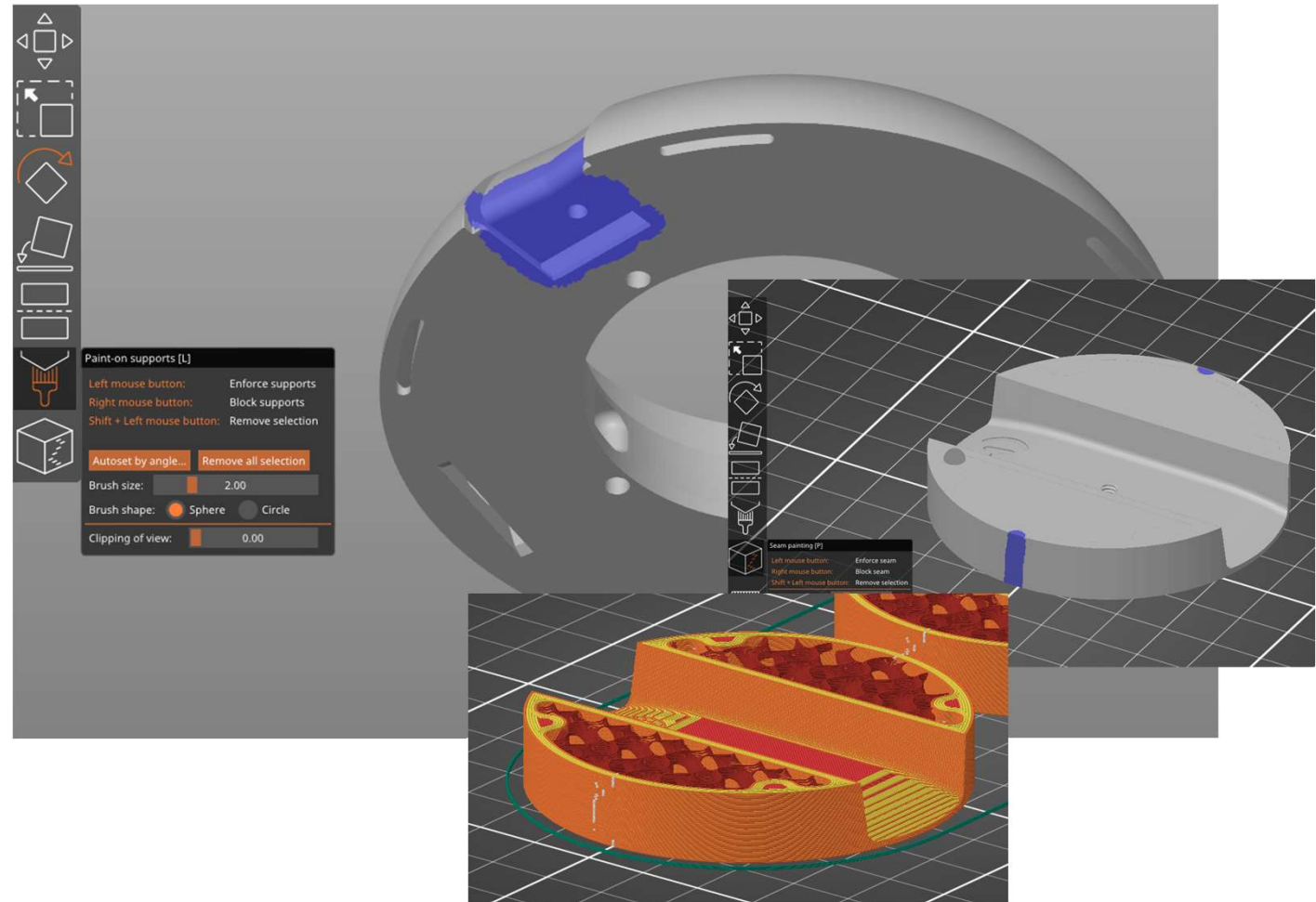
1. Infill body settings



2. Important infill settings for your profile

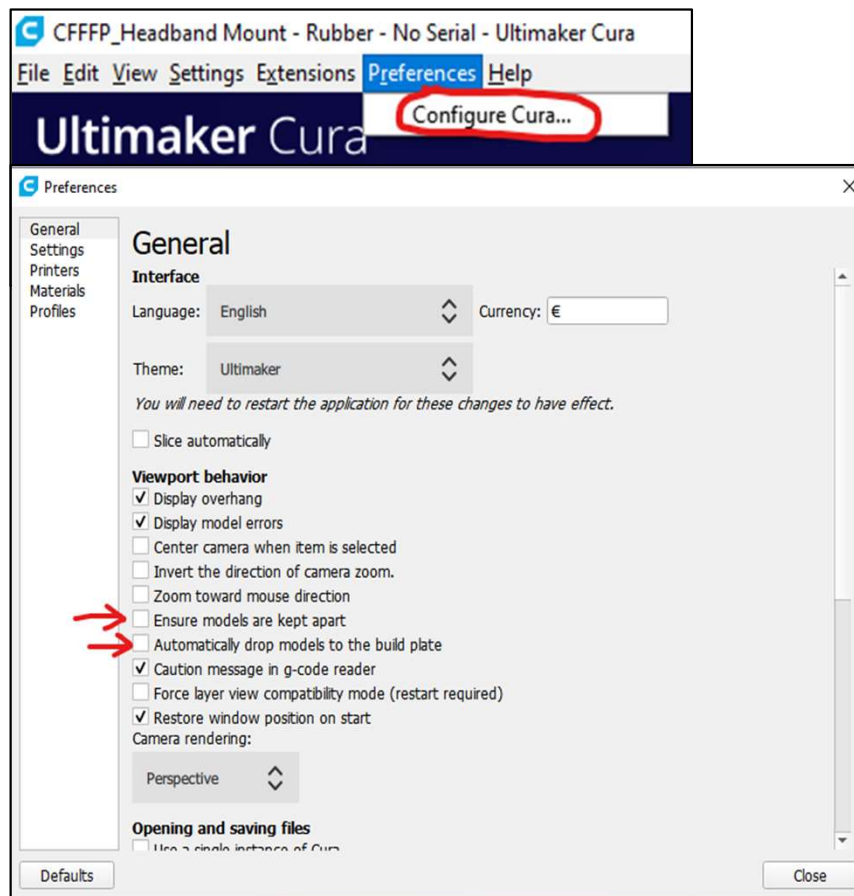


3. Prusa's Paint-on supports and seams keep your outer surfaces looking pretty!

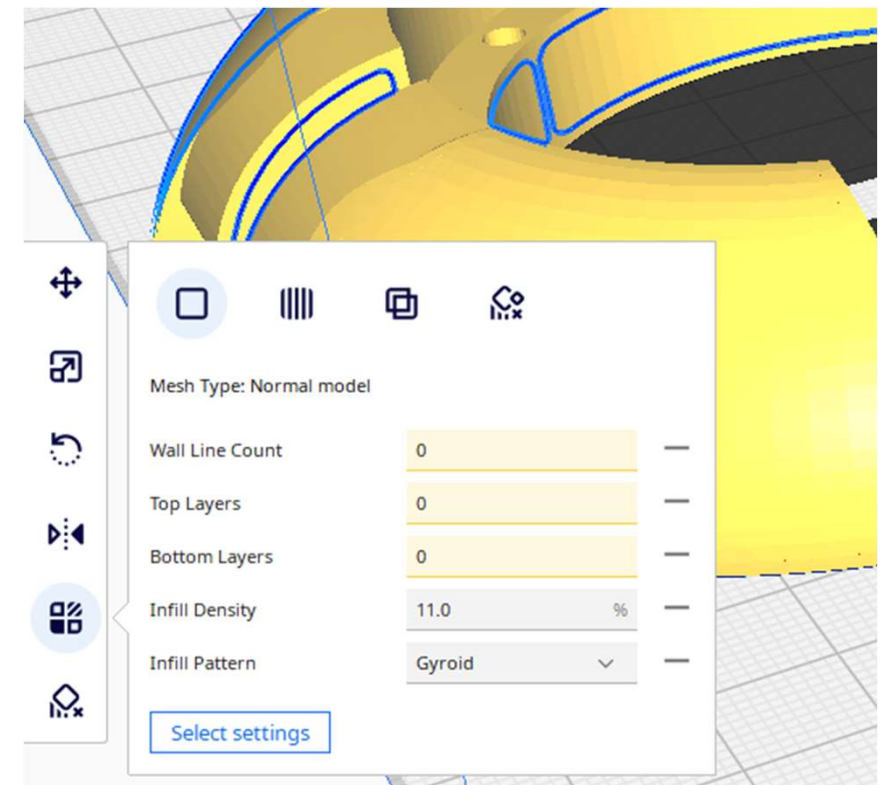


(H) Cura Settings

1. Before opening Cura Files, make sure these two boxes are un-checked, so the models don't move around on import

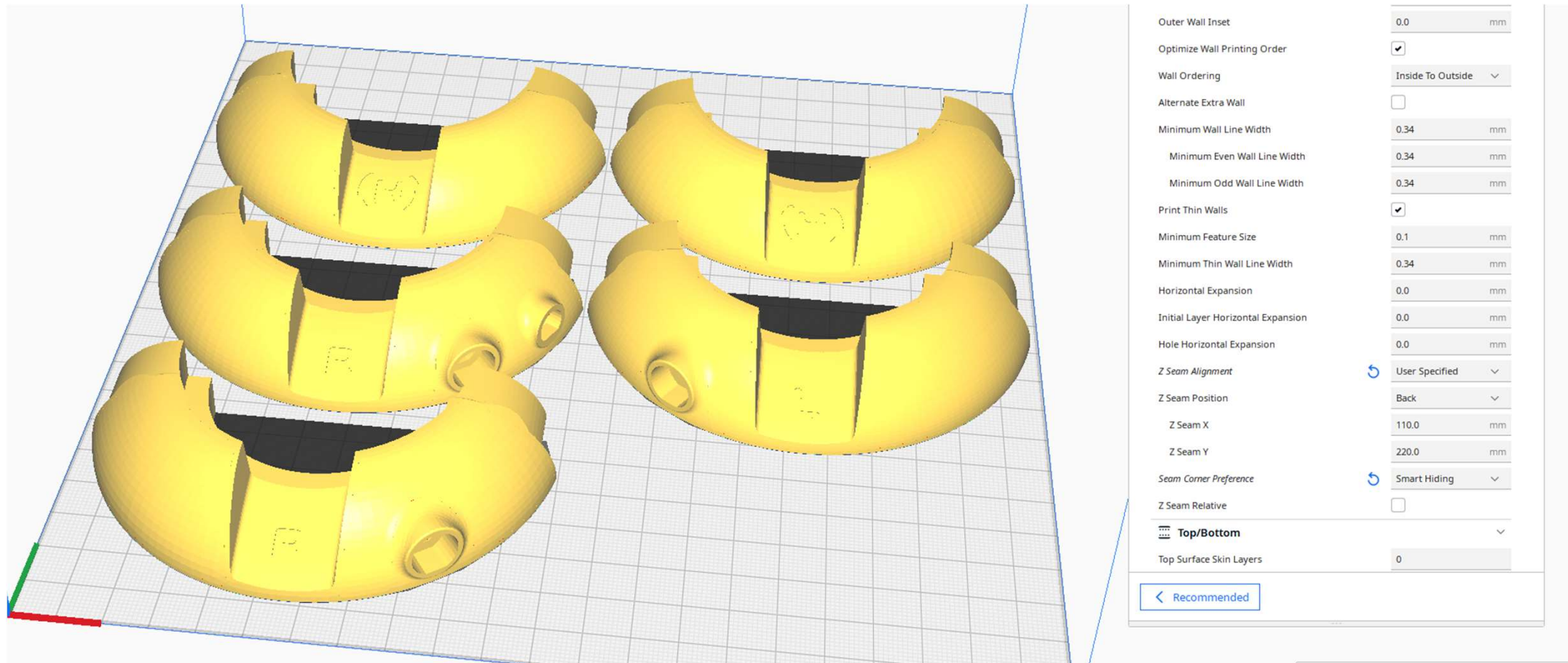


2. Required infill body settings



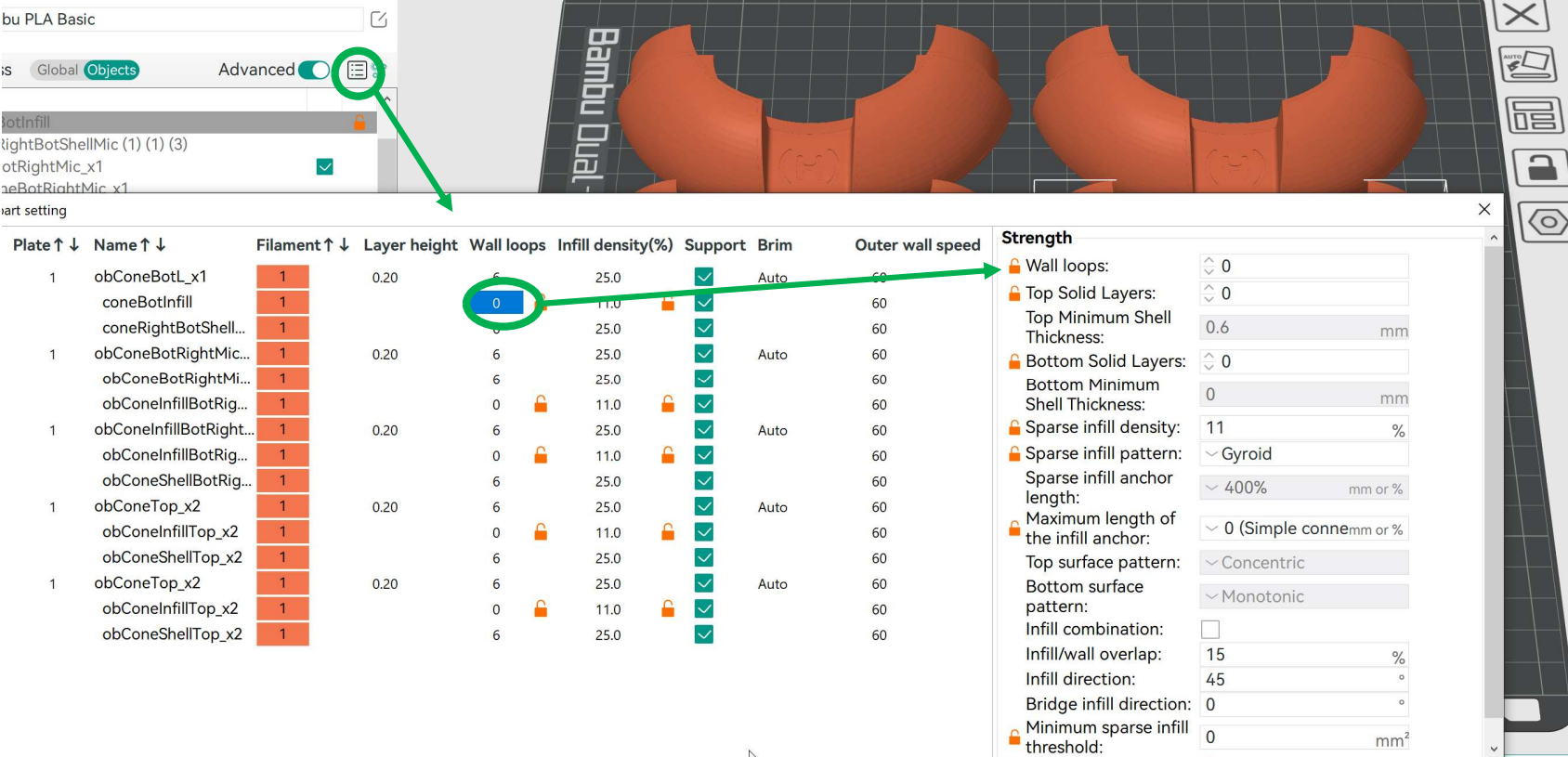
(H) Cura Settings continued

3. To keep your seams hidden, point the open end of your cones to the back, and set Z seam alignment to the back.



(H) Bambu - Orcaslicer Settings

BambuStudio doesn't yet support the required infill anchor settings for head(ame), so slice with OrcaSlicer!
<https://github.com/SoftFever/OrcaSlicer/releases>



The screenshot shows the Bambu Studio interface. The 'Objects' tab is selected, and a green circle highlights the 'Advanced' toggle. A green arrow points from this circle to the 'Wall loops' column in the settings table. The table lists various parts and their settings, including 'obConeBotL_x1' and 'obConeBotRightMic...'. The 'Wall loops' column shows values like 6, 0, and 6. The 'Infill density(%)' column shows values like 25.0, 11.0, and 25.0. The 'Support' column shows checkboxes for support settings. The 'Brim' column shows 'Auto' for most parts. The 'Outer wall speed' column shows values like 60 and 60. The 'Strength' panel on the right shows settings for 'Wall loops', 'Top Solid Layers', 'Bottom Solid Layers', 'Sparse infill density', 'Sparse infill pattern', 'Maximum length of the infill anchor', 'Top surface pattern', 'Bottom surface pattern', 'Infill combination', 'Infill/wall overlap', 'Infill direction', 'Bridge infill direction', and 'Minimum sparse infill threshold'.

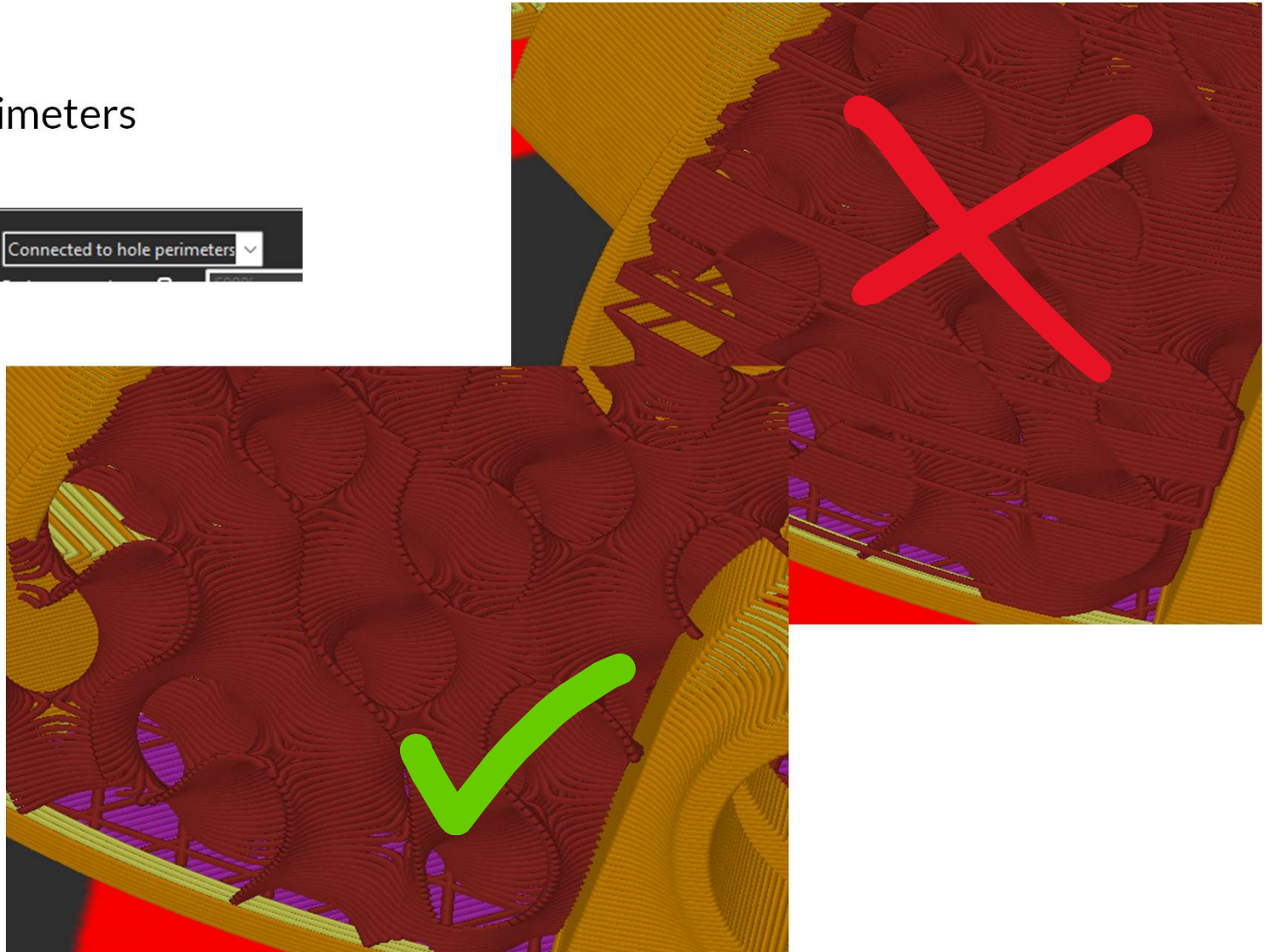
Plate ↑ ↓	Name ↑ ↓	Filament ↑ ↓	Layer height	Wall loops	Infill density(%)	Support	Brim	Outer wall speed
1	obConeBotL_x1	1	0.20	6	25.0	✓	Auto	60
	coneBotInfill	1		0	11.0	✓		60
	coneRightBotShell...	1		6	25.0	✓		60
1	obConeBotRightMic...	1	0.20	6	25.0	✓	Auto	60
	obConeBotRightMi...	1		6	25.0	✓		60
	obConeInfillBotRig...	1		0	11.0	✓		60
1	obConeInfillBotRight...	1	0.20	6	25.0	✓	Auto	60
	obConeInfillBotRig...	1		0	11.0	✓		60
	obConeShellBotRig...	1		6	25.0	✓		60
1	obConeTop_x2	1	0.20	6	25.0	✓	Auto	60
	obConeInfillTop_x2	1		0	11.0	✓		60
	obConeShellTop_x2	1		6	25.0	✓		60
1	obConeTop_x2	1	0.20	6	25.0	✓	Auto	60
	obConeInfillTop_x2	1		0	11.0	✓		60
	obConeShellTop_x2	1		6	25.0	✓		60

Strength

- Wall loops: 0
- Top Solid Layers: 0
- Top Minimum Shell Thickness: 0.6 mm
- Bottom Solid Layers: 0
- Bottom Minimum Shell Thickness: 0 mm
- Sparse infill density: 11 %
- Sparse infill pattern: Gyroid
- Sparse infill anchor length: 400% mm or %
- Maximum length of the infill anchor: 0 (Simple connection) mm or %
- Top surface pattern: Concentric
- Bottom surface pattern: Monotonic
- Infill combination: ☐
- Infill/wall overlap: 15 %
- Infill direction: 45 °
- Bridge infill direction: 0 °
- Minimum sparse infill threshold: 0 mm²

(H) Super Slicer

1. Sparse Infill – Connected to hole perimeters



(H) Thanks for Printing :)

If you are reading this...

I want you to know that I appreciate the effort you've put in to printing your own headphones. You will likely feel empowered now that you have produced a great product in your very own home or office. It's head(amame)'s goal to keep providing you things to print that can be infinitely repaired.

If you enjoyed this experience, please consider supporting head(amame) on [Patreon](#), submit for a [Serial Number](#), or just share your favorite songs with a friend on your new headphones.

If you feel like getting your hands dirty, the files are open for modification so that you can use them as your canvas to create and share.

I hope that you enjoy your head(amame) for years to come,

Morgan