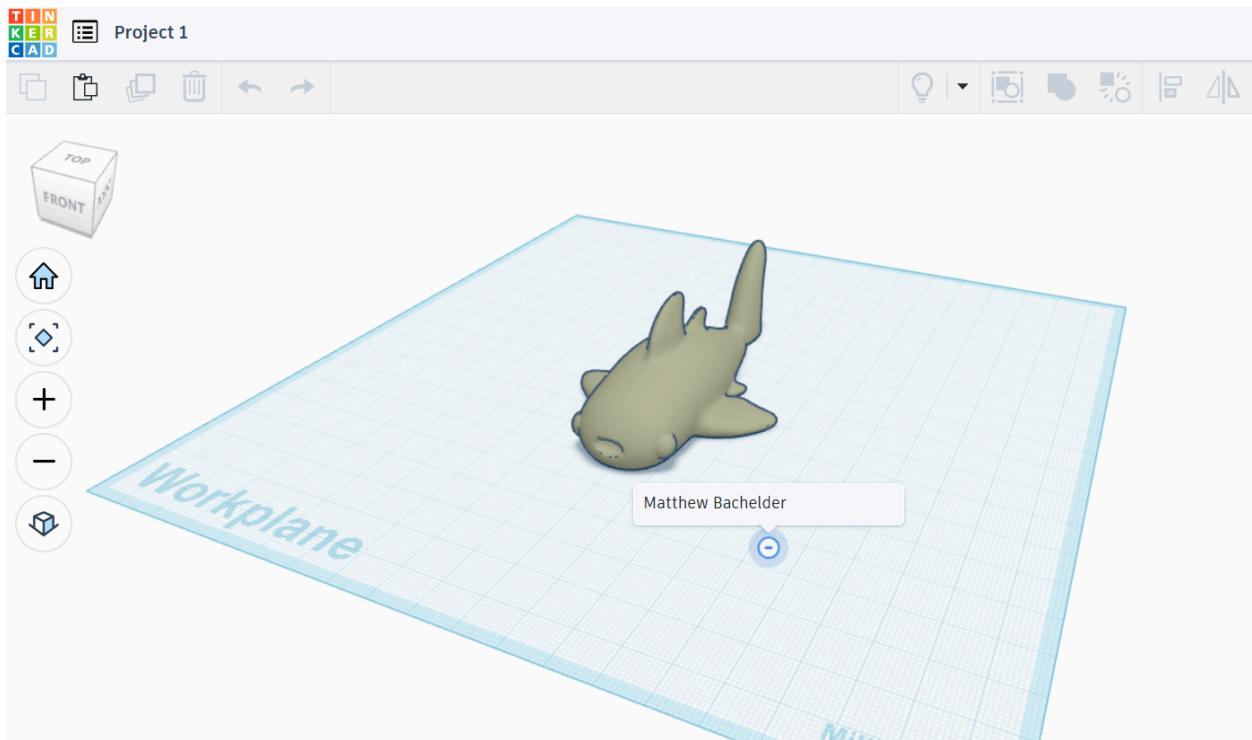


# Initial 3D Print

## Where the Design Came From

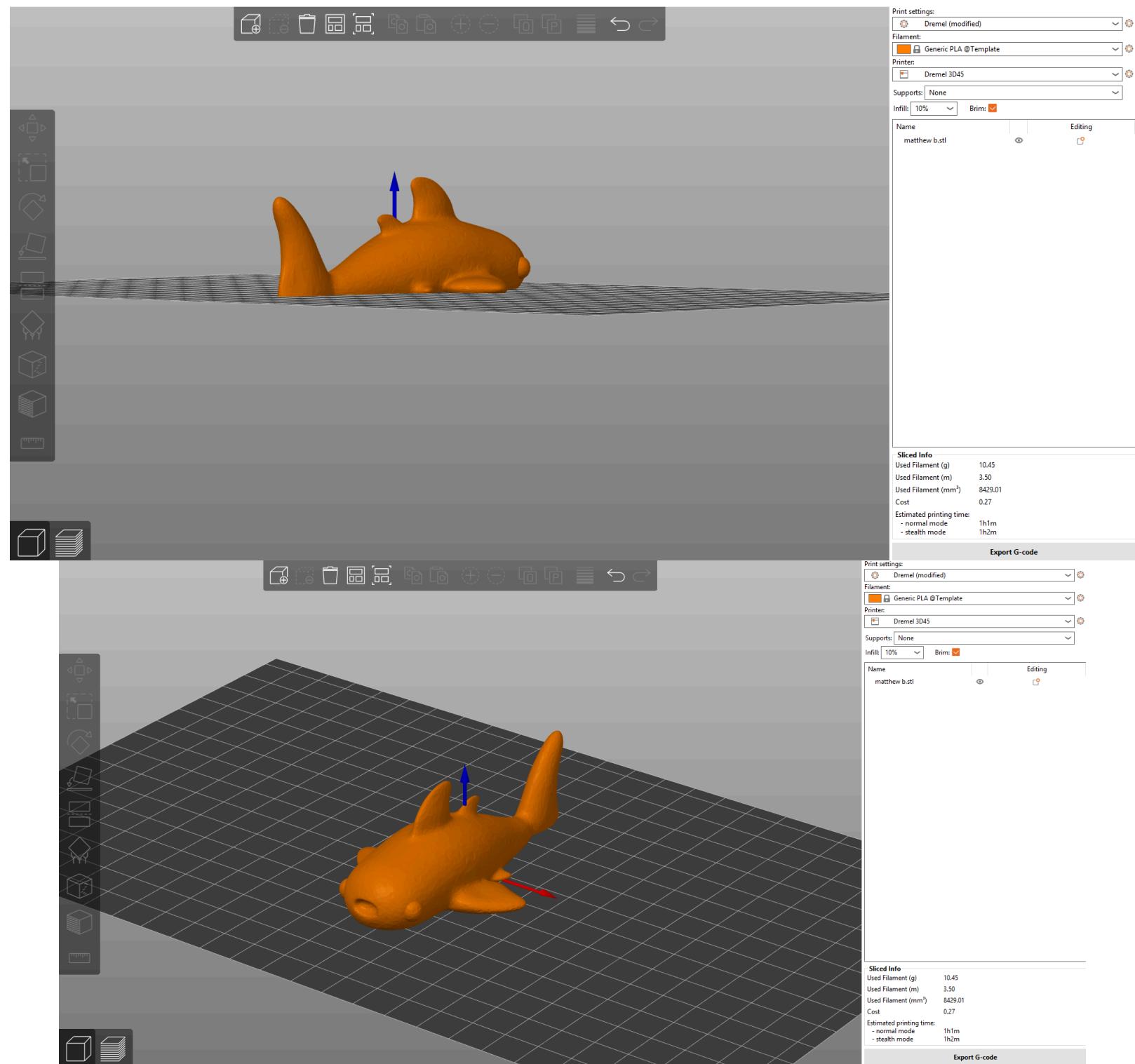
<https://www.thingiverse.com/thing:6854929>

## Tinkercad

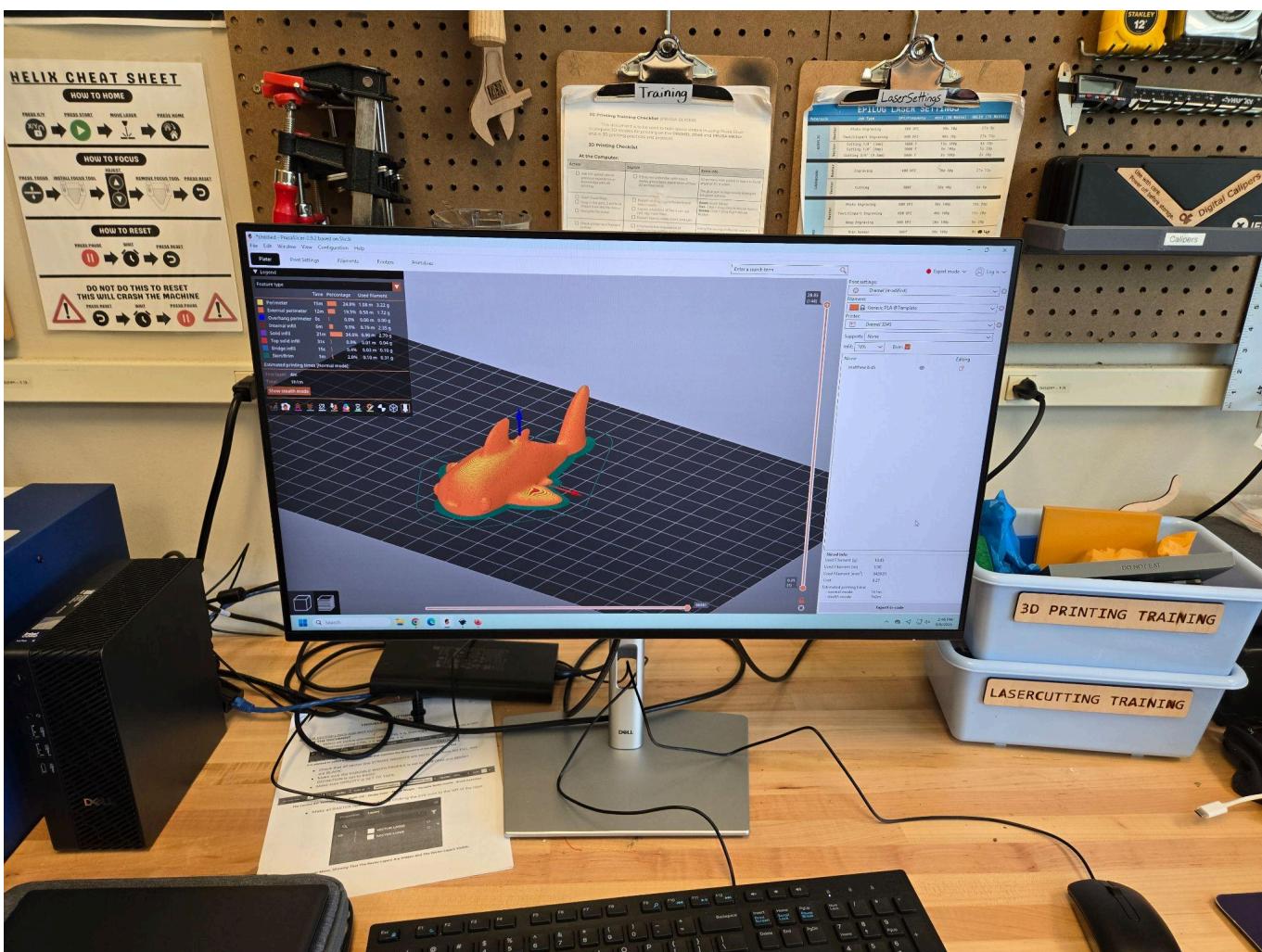
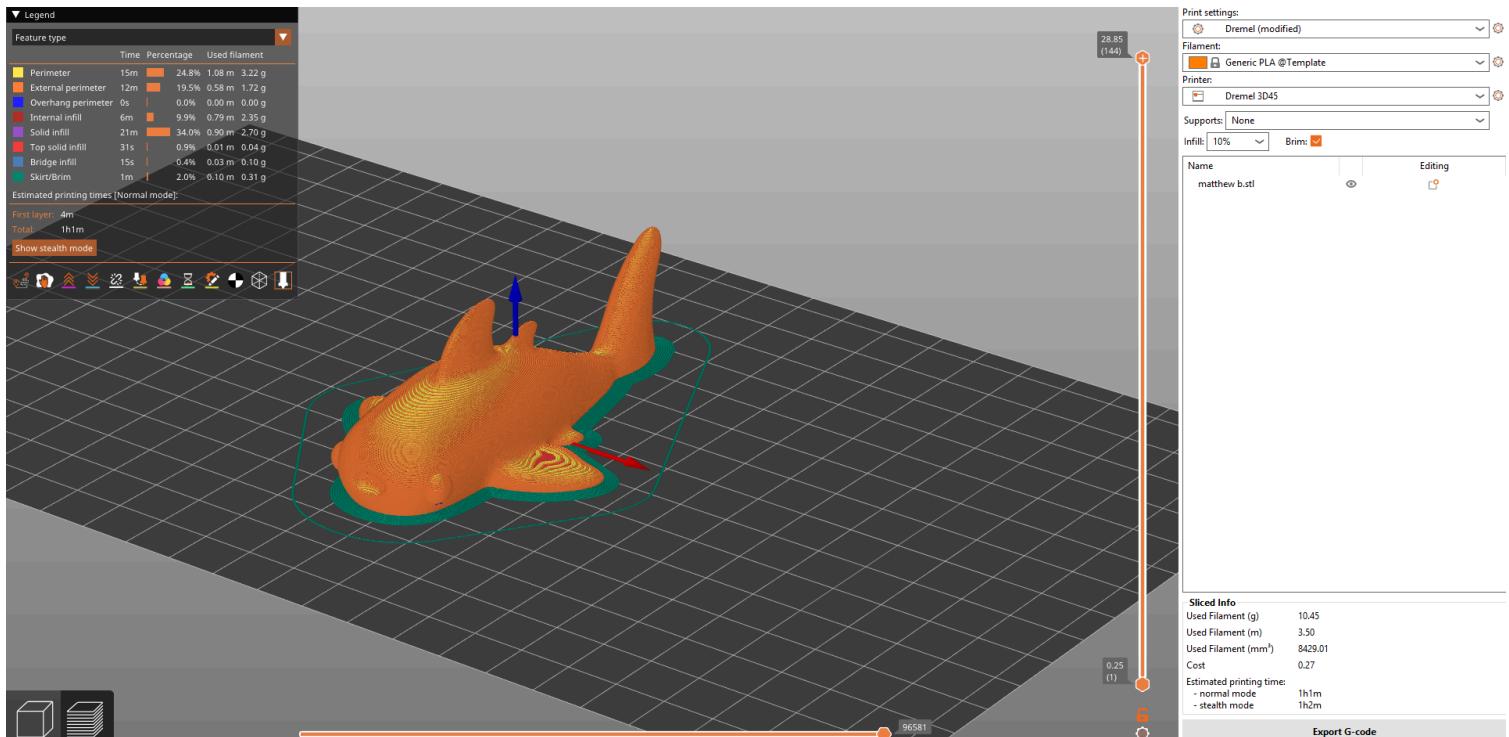


# Prusa Slicer

## Initial Model



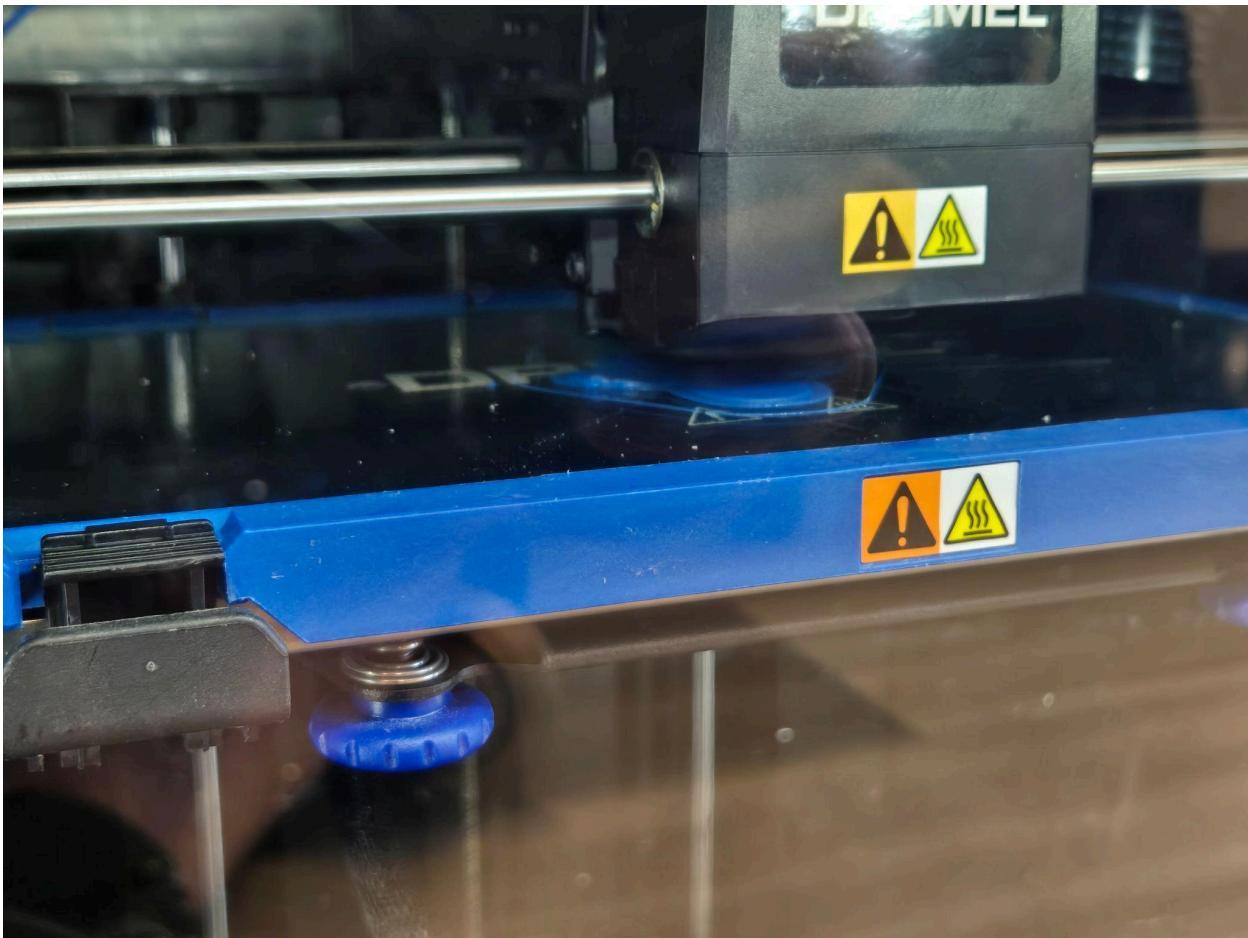
# Sliced Model

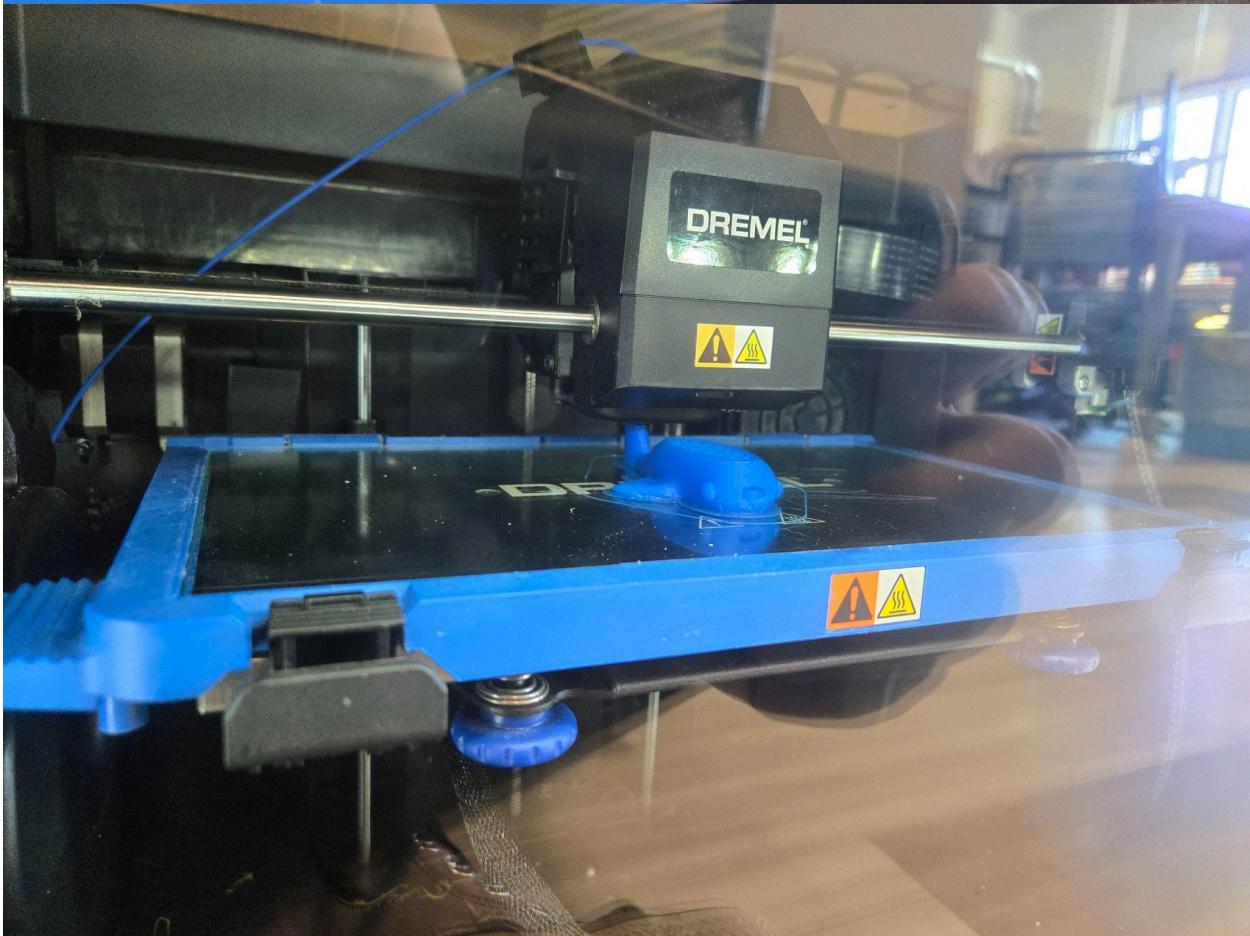
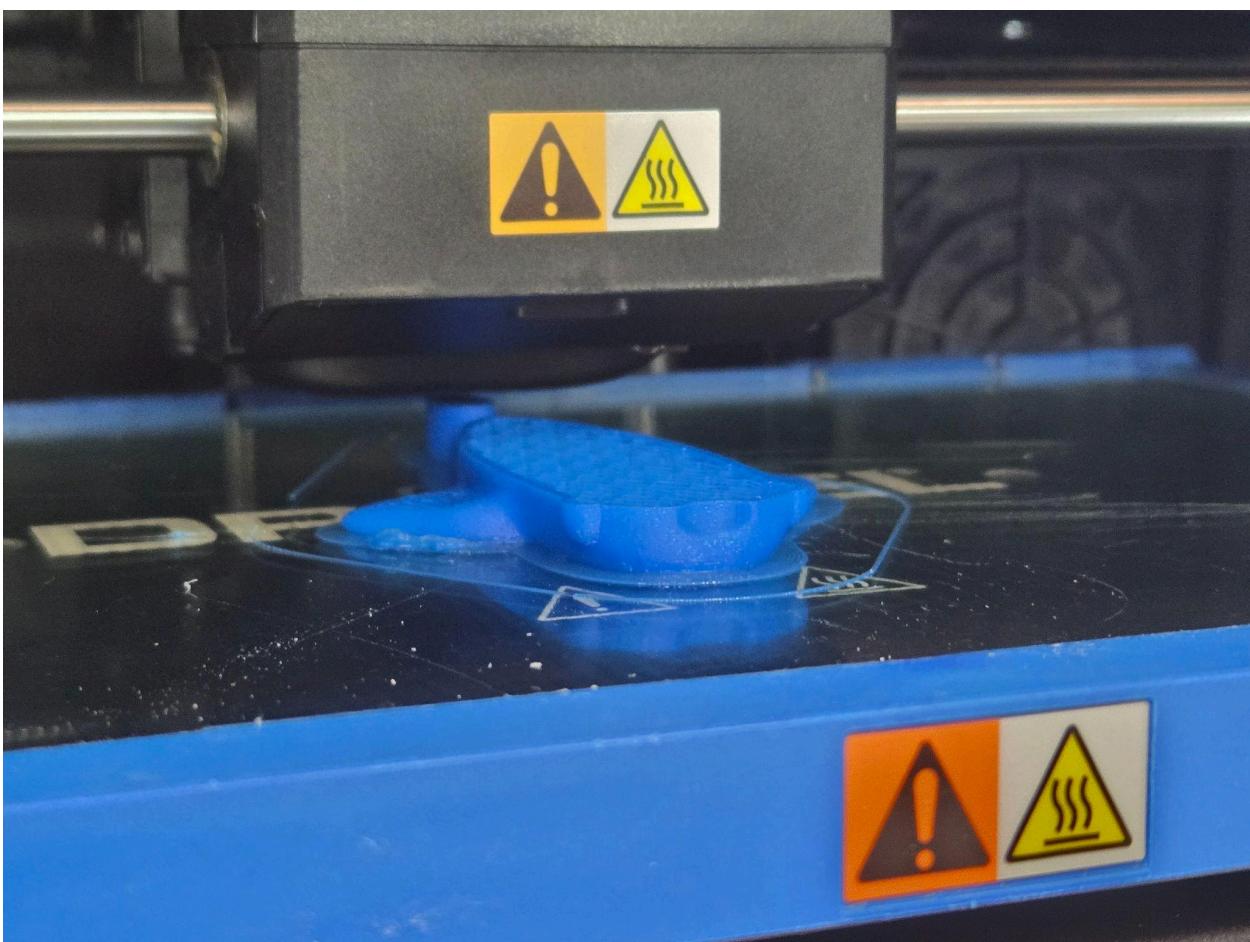


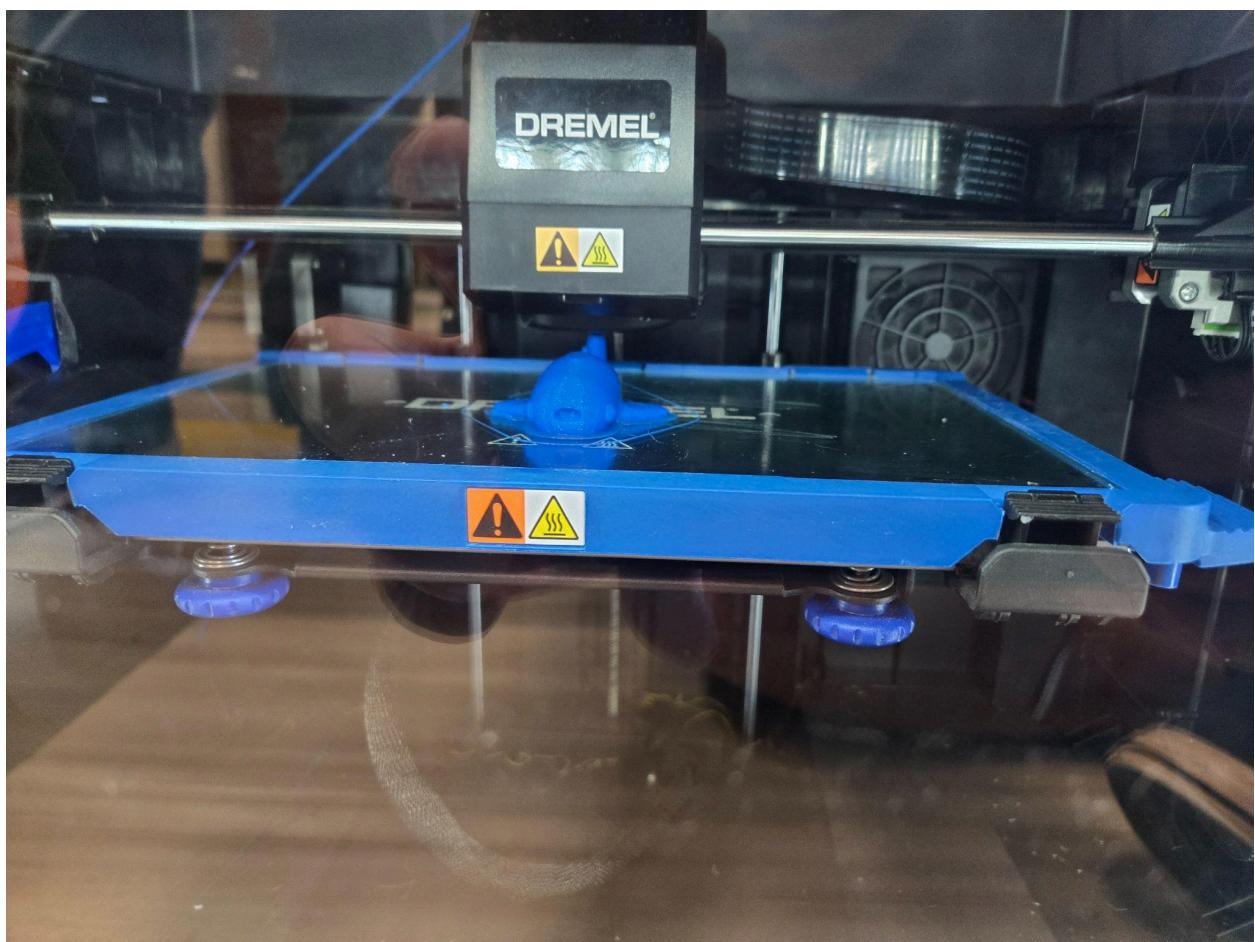
## Support Structures

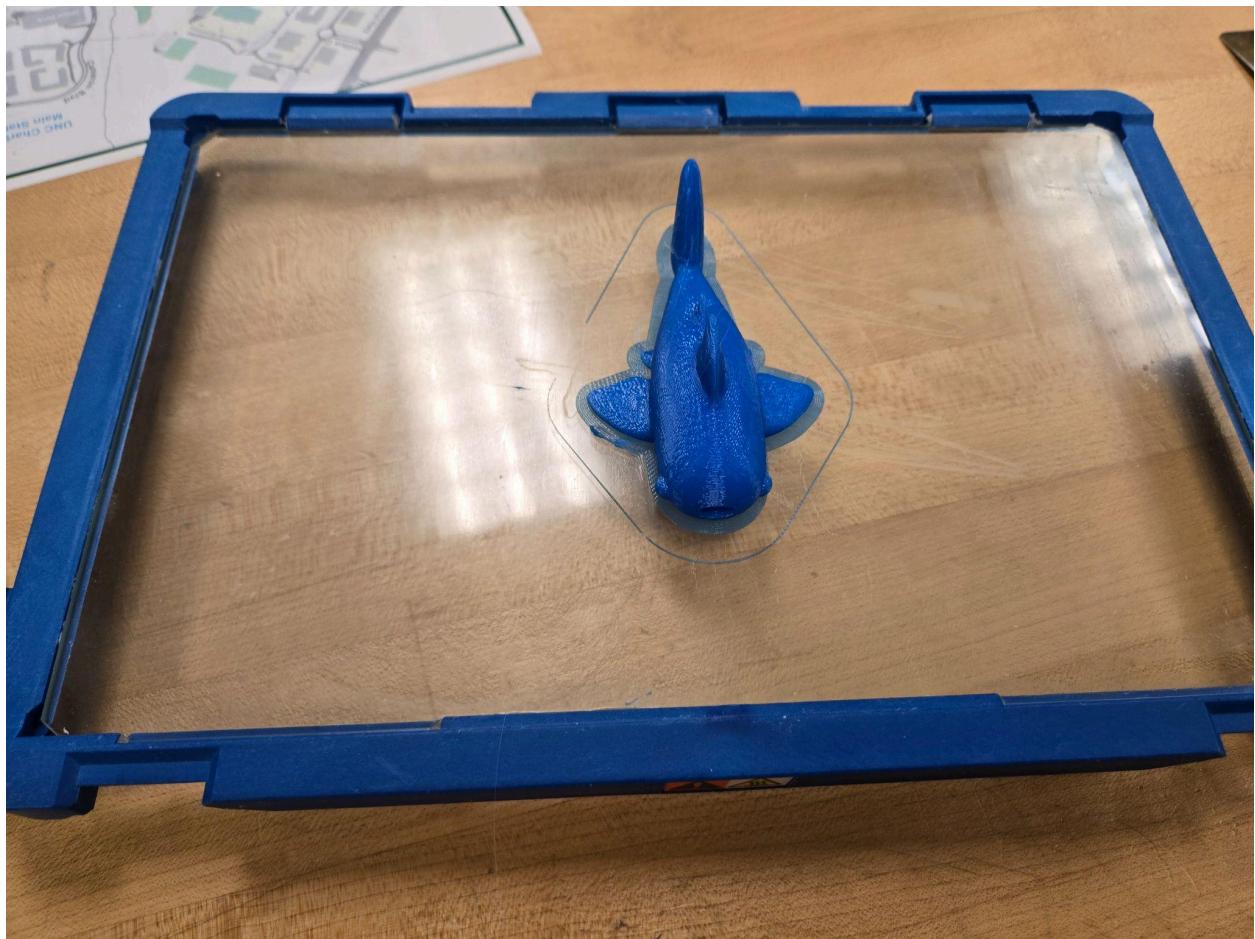
The only overhangs on this model were the small mouth, the eyes, and the tail, none of which warranted enough overhang for support structures. I spoke with the Makerspace staff to double-check this, and they agreed as well. The only support structure was the brim to help it stick to the pad, although it probably didn't need it. I included it anyway.

## 3D Print in Progress

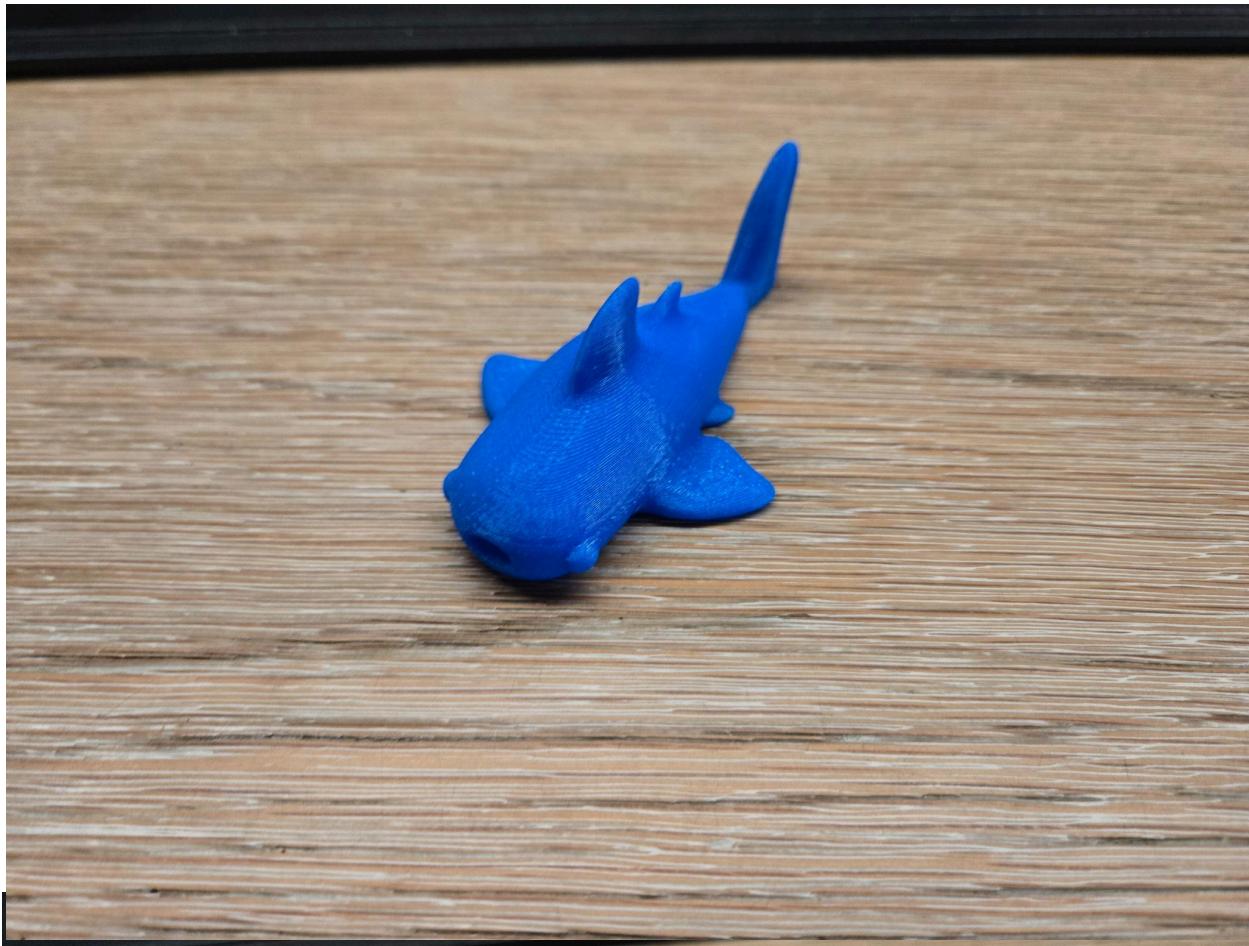








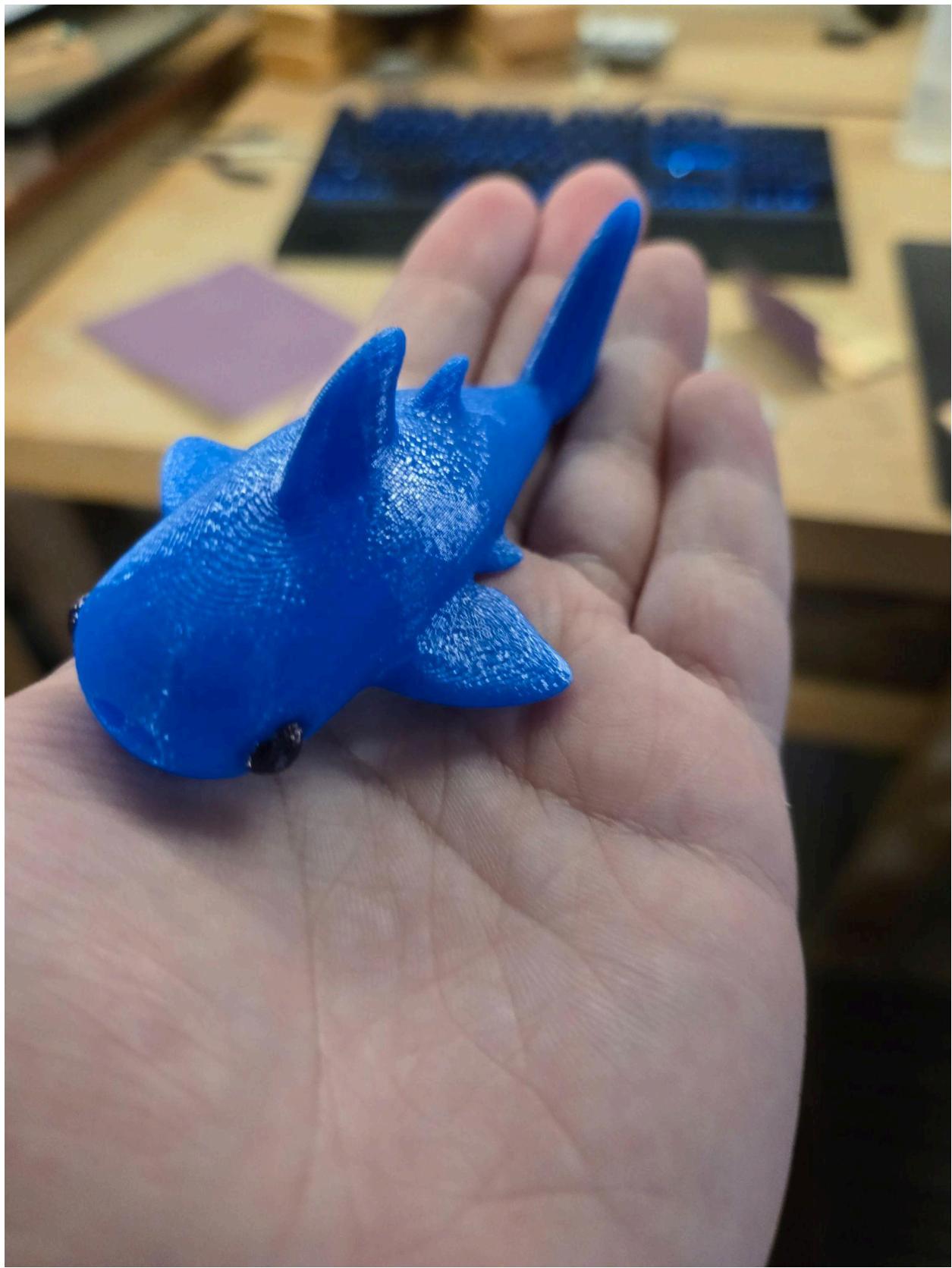
## Pre-Post Processing



# Post Processing







# Documentation

## During Print

Luckily, I did not have to print multiple times. My print worked on the first time around. I did have some light help from Alvin, but that was only figuring out the settings on Prusa Slicer. I was aiming for the blue PLA, but the printer was not working properly, so Alvin also fixed that for me (definitely not something I could do; there was material in there, but the machine said no material available). Other than that, the print went very smoothly. I scraped it off the plate, and it looked good. It wasn't super fuzzy and had very little plastic coming off of it that I was able to pick off.

## Post Processing

I was okay with the texture of the top, but the bottom still looked slightly sticky and uneven. I washed the print with dish soap before sanding down the bottom with 400-grit sandpaper, and that made the bottom much smoother. After that, I used a black Sharpie to black-out the eyes. After that, I was happy with my print.