Planktivore Housing Layout: Status Update

Summary:

I investigated four layouts in an attempt to understand the drag characteristics of each configuration. I'll later address the details of optimizing to reduce bow wake. For now, it made sense to see which shapes were the most streamlined (i.e. lowest total drag).

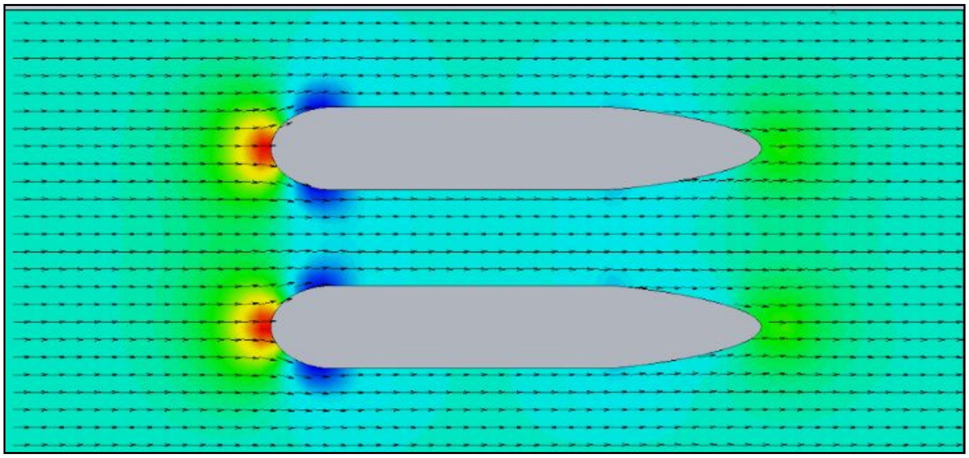
1. "Parallel tubes" — The layout Paul sent me
2. "Collinear tubes" — Two opposing cylinders perpendicular to flow
3. "Collinear foils" — Two streamlined opposing foils (like a hammerhead shark)
4. "Small foils" — Reduced diameter collinear foils (if the internal electronics did not constrain the minimum diameter)

I had issues getting SolidWorks flow simulation to agree with my expected results for foil drag. My simulation would yield drag coefficients that were ~5x greater than estimated for a known foil shape, regardless of my mesh and domain settings. I took data from here, which relies on MIT's XFOIL software for subsonic airfoils design.

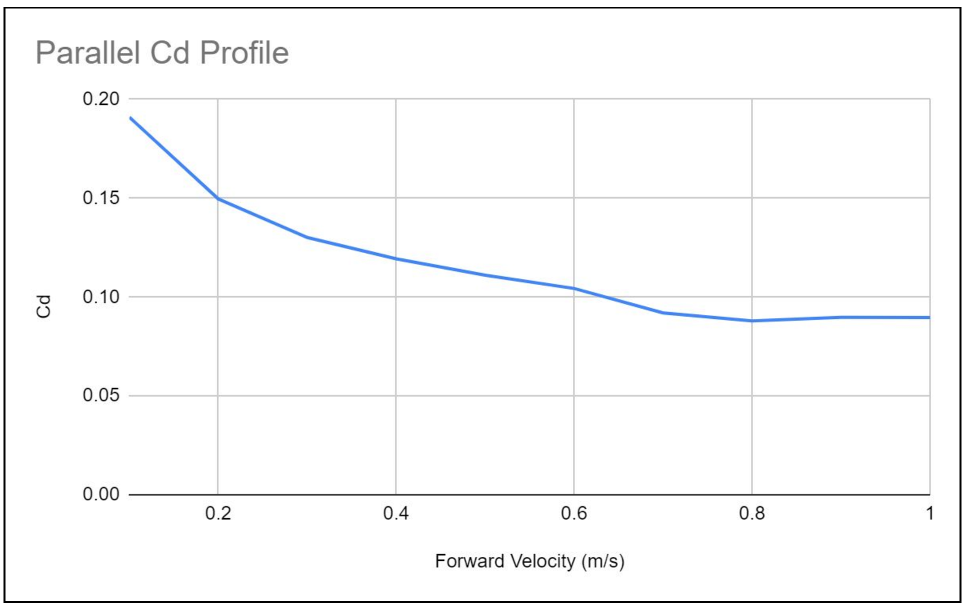
Theoretically, the opposed foil designs produce less drag than the parallel tubes, but the simulation results do not agree. I haven't been able to figure out why. For now, parallel tubes (like Zooglider) look most appealing, as that layout performed well under simulation and would be simpler to fabricate than a foil.

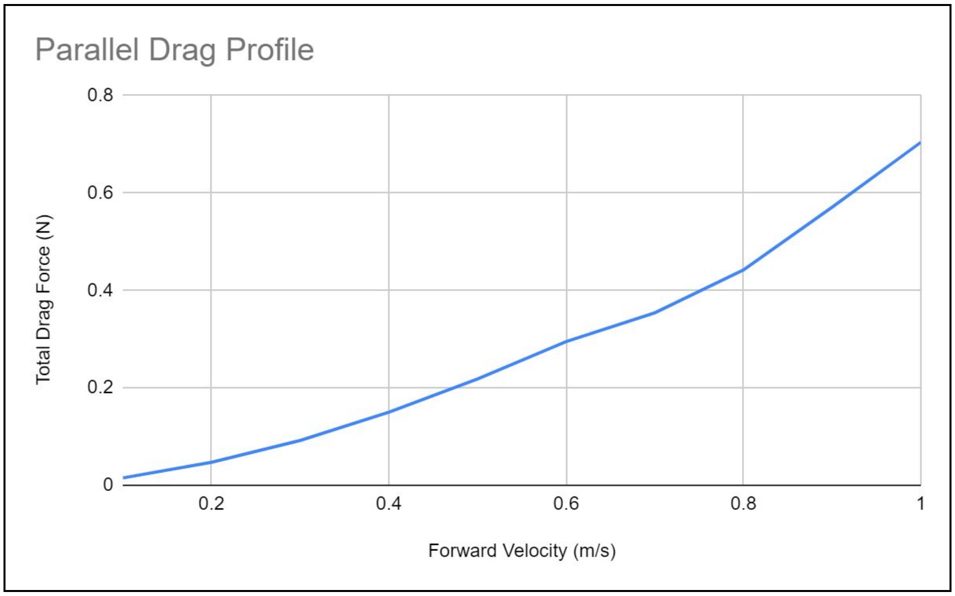
		Parallel Tubes	**Collinear Tubes**	**Collinear Foils (NACA 0024)**	**Small Foils (NACA 0024)**
O.D. (mm)		100	75	75	55
Reynold's # (1.0 m/s)		300,000	75,000	320,000	230,000
Frontal Area (cm^2)		157	546	546	400
C_d (1.0 m/s)	Simulated	0.08	0.53	0.072	0.075
	Empirical / XFoil	unknown	1.0	0.016	0.017
Total Drag Force (N)	Simulated	0.59	14.5	1.6	1.1
	Empirical / XFoil	unknown	27.3	0.44	0.34
Simulation Error vs Ideal: (o-e)/e		unknown	-47%	+350%	+224%

Parallel Tubes:



Pressure distribution and velocity vectors in parallel configuration.



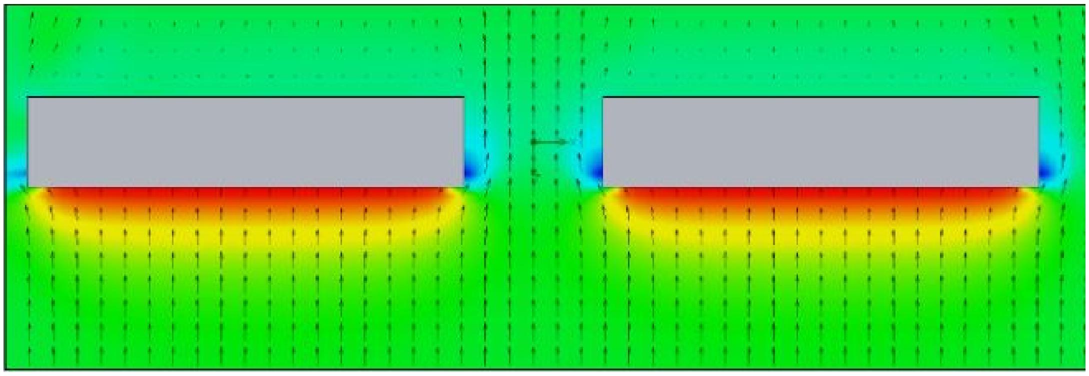


Simulation results showing a relatively coarse mesh approximation of drag coefficient (top), and total Drag (bottom).

The parallel tubes seemed to perform well, but I don't have an empirical baseline to compare the simulation results to. I couldn't find a nice chart with drag data about this elliptic bow, parabolic stern, parallel midbody shape. There were some papers about submarines that have the same shape, but I didn't see windtunnel results for the generic shape.

There isn't too much to say here, other than geometry optimization could improve performance some. However, I am worried about how realistic SolidWorks CFD is at high Reynolds numbers, so I don't know if doing a geometry optimization study will be meaningful.

Collinear Tubes:



Pressure distribution and velocity vectors in parallel configuration.

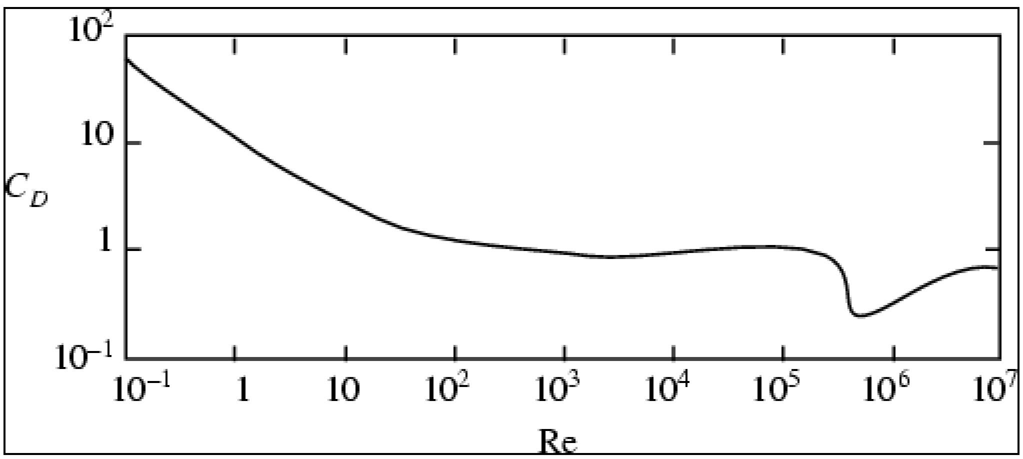


Fig. 1: Drag Coefficient of a Cylinder as a function of Reynold's Number. Data collected empirically.

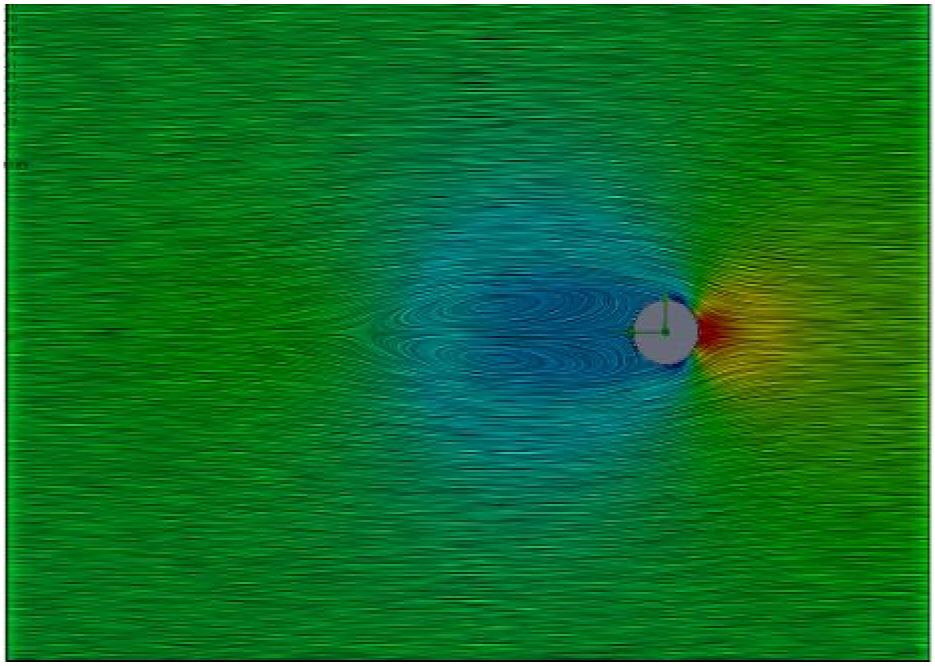
Using empirical data, we can estimate the drag experienced by two collinear cylinders perpendicular to the direction of flow at 1 m/s in water:

$$Re = \rho v D/\mu = 1000 * 1 * .075/.001 = 75,000$$

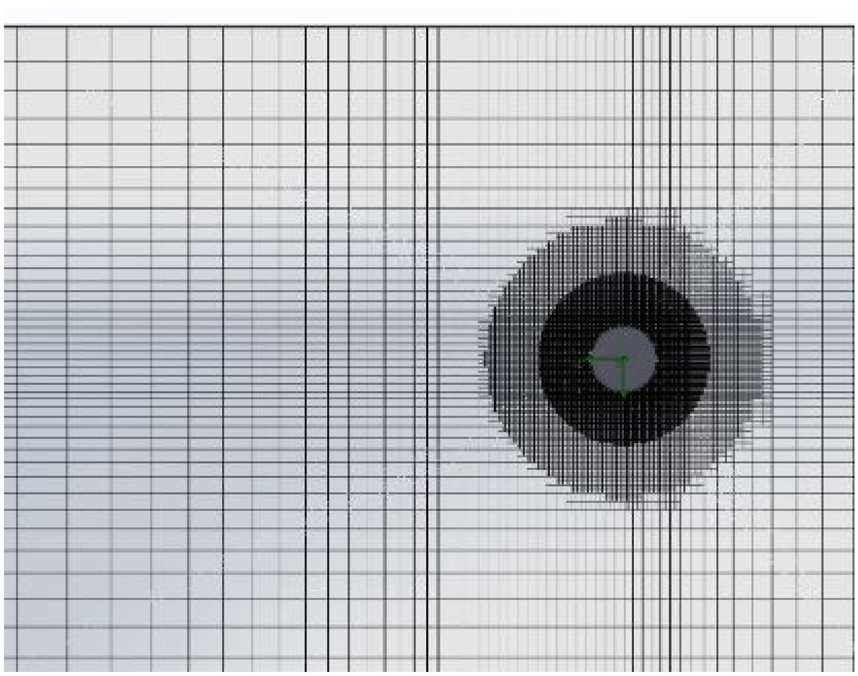
$$\text{From Fig. 1, } C_d \approx 1$$

$$F_d = \frac{1}{2} C_d \rho v^2 A = 0.5 * 1 * 1000 * 1^2 * .0546 = 27.3\ N$$

Whether you believe the empirical estimation of 27.3N or the simulated result of 14.5N, it's pretty clear that the cylinder has about the same hydrodynamic properties as a brick! This configuration is not practical, but useful for comparing simulation results to empirical results. The simulation results are accurate up to about Re = 10,000, where error in drag coefficient is approximately 10%. Even with a very fine adaptive mesh and a large computational domain, the results at Re >= 75,000 differ by about 50% from expected, likely because turbulent behavior becomes harder for SolidWorks to model. This is troubling for foil predictions, because the Reynold's numbers are well above 75,000, implying that results may not be accurate.

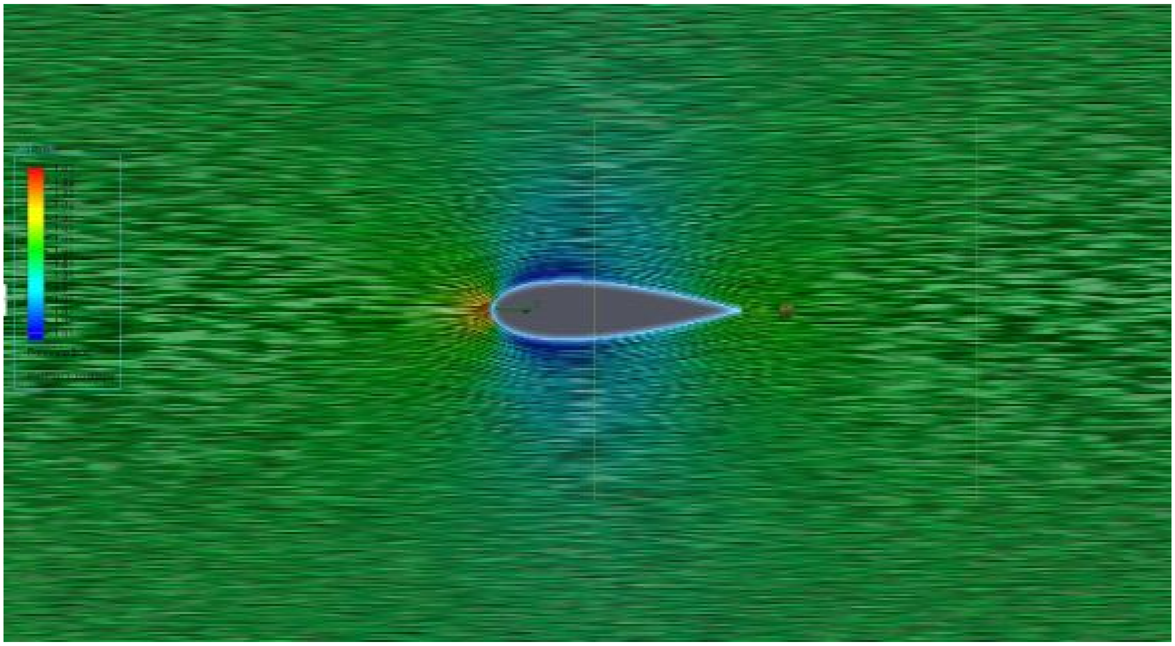


Turbulent behavior behind cylinder.

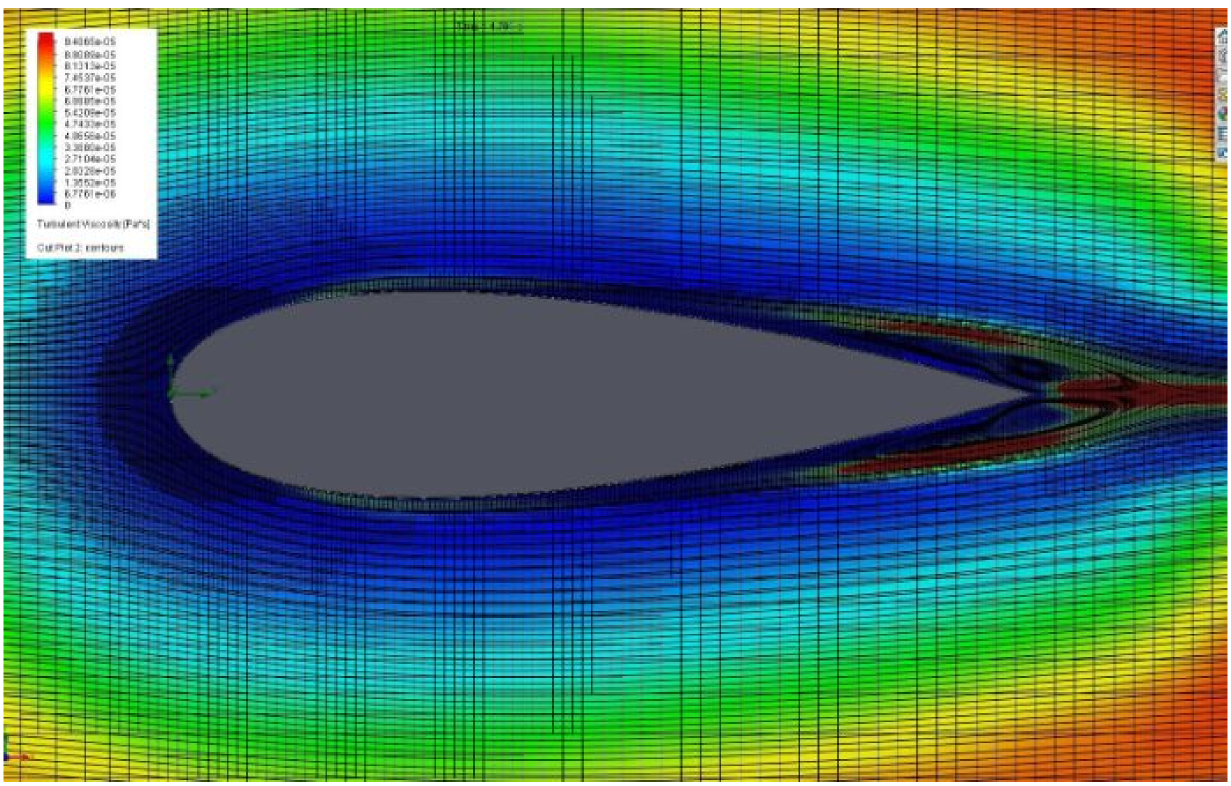


Example 2D equidistant mesh refinement. Refinement increased accuracy up to about Re = 10,000.

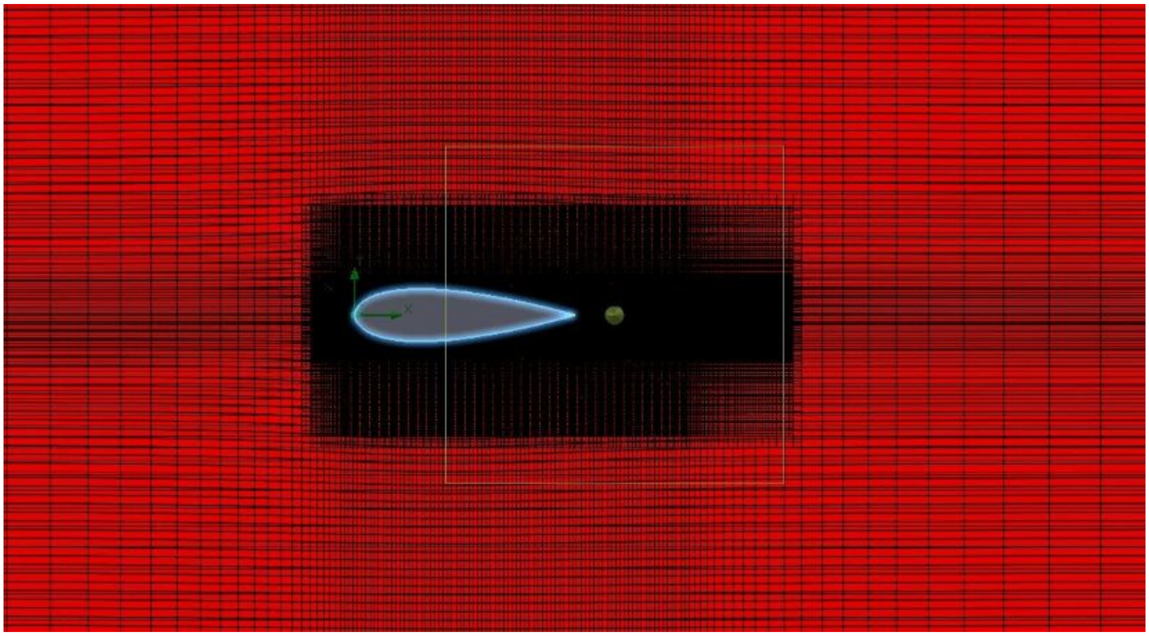
Collinear Foils:



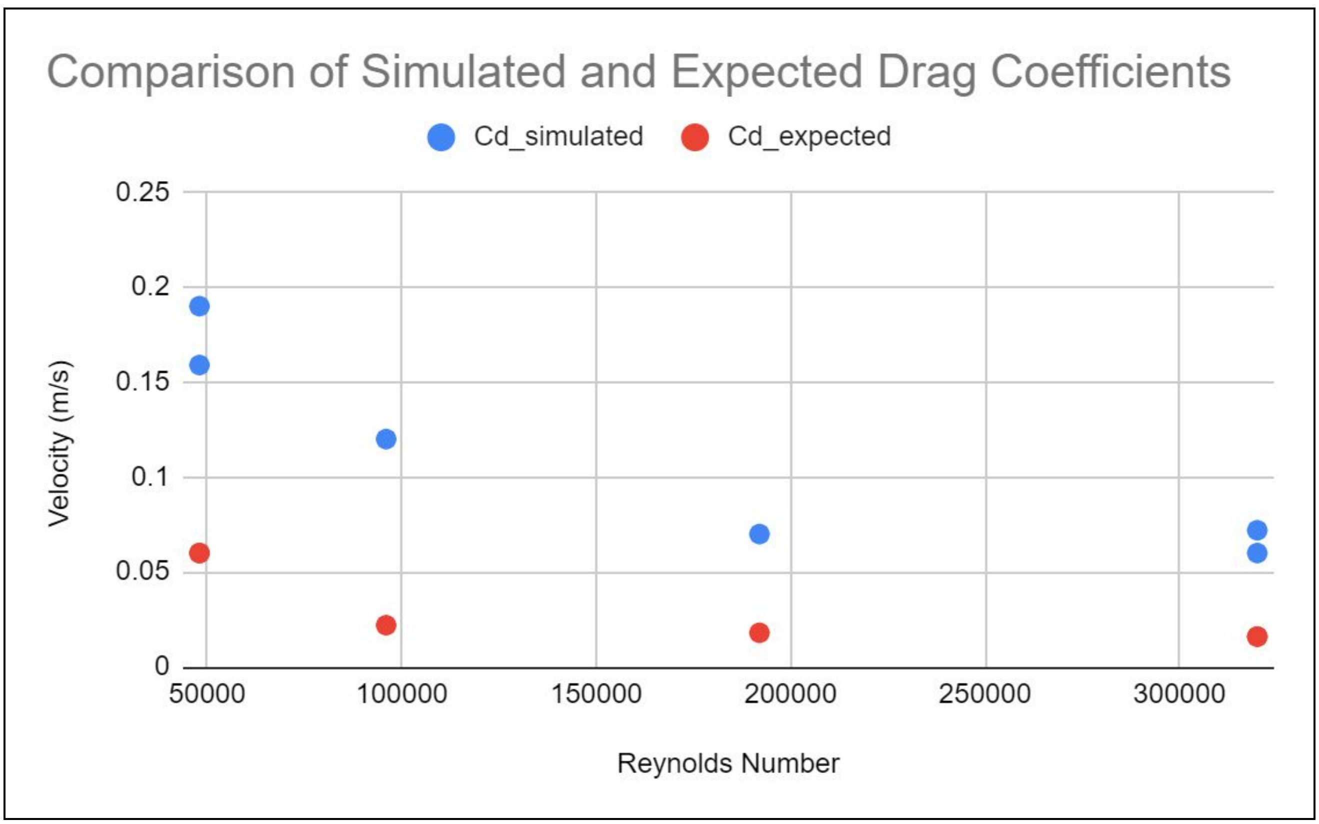
Streamlines and pressure distribution for a NACA 0015 foil using a fine uniform mesh. Re = 320,000.



Turbulent viscosity for a NACA 0015 foil using adaptive mesh refinement, Re = 320,000. Note the turbulence at the trailing edge of the foil.



Another attempt at changing up the mesh ... refined based on a rectangular zone close to the foil. None of the mesh changes brought the measured drag coefficient anywhere close to the expected.



Comparison of simulated drag coefficient and drag coefficient from XFOIL for a NACA 0024 foil. At Re = 50,000 and Re = 320,000, you can see that attempts at refining the mesh didn't bring the Cd down to expected levels.

Future Work:

- Figure out disparity between CFD and XFOIL results
- Select most promising layout
- Look into shear+bow wake as a function of distance from imaging volume
    - Understand plankton sensitivity to shear
- Optimize to minimize Cd and shear profile