Fluid Dynamics Lab Technical Memorandum

Jacob Fain
E80 Section 1
Team 11
Dated: March 8, 2024

Abstract

Fluid dynamics is a fundamental element of applications in aerospace and marine engineering. It comprises the dominant physics as structures move through or are moved past by gases, such as Earth's atmospheric air, and liquids, such as the ocean water into which the AUV will be deployed at this project's conclusion. Mastery of fluid dynamics can therefore inform the geometric design of structures intended to traverse fluids, for such purposes as maximizing the thrust efficiency of self-propelled vehicles like the AUV. The following experiment was performed in order to calculate the lift and drag experienced by various structures, both as measured through the use of a wind tunnel, as well as simulated in COMSOL Multiphysics software. Comparing these two approaches led to the conclusion that COMSOL simulations are a sufficient but non-ideal proxy in the absence of physical testing. This experiment applied fluid dynamics to the consideration of changes to the shape and propulsion of the AUV, which were ruled out as it was deemed that the lift and drag the AUV will experience are negligible.

Theory

Aerodynamic Forces

Lift and drag are the two components of the aerodynamic force, the net mechanical force that acts on an object's center of pressure as it experiences motion relative to a liquid or gas [1]. Drag, D is the component that is antiparallel to the object's motion, and so is analogous to the kinetic friction between an object sliding along a surface. It is a function of the fluid's density, ρ, the velocity of motion, V, and the object's size and shape, as expressed in a reference area, A. This reference area is usually chosen to be the largest cross-sectional area of the object that is normal to the velocity [2], and the overall effect of its geometry on the drag force is described by a untiless coefficient of drag (Equations 1A, 1B). Meanwhile, lift, L is the component of the aerodynamic force that is perpendicular to the object's motion. It is also a function of those same properties, except it incorporates a different reference area, namely the largest cross-sectional area parallel to the velocity [2], and thus a resulting coefficient of lift (Equations 2A, 2B).

$$D = \frac{1}{2}\rho V^2 A C_D \tag{1A}$$

$$C_D = \frac{2D}{\rho V^2 A} \tag{1B}$$

$$L = \frac{1}{2}\rho V^2 A C_L \tag{2A}$$

$$C_L = \frac{2L}{\rho V^2 A} \tag{2B}$$

Lift can be utilized to the benefit of vehicles designed to traverse air or water, such as in airplanes wherein no vertical thrust need be provided to maintain altitude at sufficient horizontal velocity. On the other hand, drag necessarily opposes the motion of a vehicle as it moves forward through a fluid, and so must be overcome by its thrust. It is therefore useful to predict the magnitudes of lift, and especially drag, when considering the design of an AUV.

Similitude for Modeling

Similitude describes the use of measurements that are captured for a model, in order to estimate the behavior of a real-world system. An accurate prediction requires geometric similitude, wherein the model and system have congruent dimensions in all directions, as well as kinematic and dynamic similitude. In aerodynamics, kinematic similitude is attained when there is similarity between the model and system in their compressibility parameter, or Mach number, which is the ratio between an object's relative speed in a fluid and the speed of sound [3]. Note that the object's speed can be calculated by rearranging Bernoulli's Equation for flow in response to a pressure differential, ΔP [4], in the absence of an altitude change, to obtain Equation 3:

$$V = \sqrt{\frac{2\Delta P}{\rho}}. \tag{3}$$

Dynamic similitude is achieved by obtaining equality in the viscosity similarity parameter, or Reynolds number, which is the ratio of the inertial and viscous forces of the fluid acting on an object [3]. The Reynolds number, Re, is a dimensionless function of the length of the object against which the fluid moves, l, the bulk or dynamic viscosity of the fluid, μ, and the density and relative speed of the fluid [5], according to Equation 4:

$$Re = \frac{\rho V l}{\mu}. \tag{4}$$

By arranging an experiment performed on a scale model in such a way that its Reynold number, Re_M is equal to that of the real-world system, Re_S, one can identify the flow speeds required for each to exhibit similitude (Equations 5A, 5B). This makes it possible to estimate the forces that the real-world system will experience at a given velocity.

$$Re_M = \frac{\rho_M V_M l_M}{\mu_M} = Re_S = \frac{\rho_S V_S l_S}{\mu_S} \tag{5A}$$

$$V_S = V_M \frac{\rho_M l_M \mu_S}{\rho_S l_S \mu_M} \tag{5B}$$

Experimental and Analytical Methods

It was decided that performing wind tunnel trials at different velocities on a 1:3 scale model of the unmodified AUV would, via similitude, provide useful insight into the forces on the AUV as it navigates water. These same trials would also be conducted for three nose cones. Results for all four objects would then be compared against calculations from computational fluid dynamics (CFD) performed in COMSOL [2]. The nose cones provided data points to examine the fidelity of COMSOL to physical testing, informing future approaches to aerodynamics trials.

Wind Tunnel Trials

A calibration curve between fan speed and wind speed was obtained by varying the fan RPM at 5% increments with no object on the force measurement stand, tracking the resulting pressure differential read by a digital manometer, and computing Equation 3 to find the resulting flow speed. This curve was utilized to determine the wind speed at each fan speed as it was incremented from 15% to 55%. This resulted in the lift and drag measured by the force measurement stand varying for the AUV scale model and nose cones – Nose Cone 1 being a flat cap, Nose Cone 2 being a rounded hemisphere, and Nose Cone 3 being a pointed cone. These forces were read off from the test chamber's LVDT box, and saved to Google Sheets for analysis.

CFD

Models of all four objects were then created in or imported into COMSOL. In each trial, one object was centered in a mockup of the wind tunnel test chamber, through which a simulated fluid could flow at various speeds. All four objects and the tunnel had the same dimensions as their real-world counterparts. For the nose cones, simulating air of the same density and at the same speeds as in the physical wind tunnel ensured uniformity between the physical and computational trials. For the AUV, rather than using its 1:3 scale model, its real dimensions were simulated. In this case, water was the simulated fluid, flowing at speeds scaled from those of the scale robot in the wind tunnel according to Equation 5B. Velocity profiles and pressure contour plots for the nose cones were outputted, as well as values for the lift and drag forces experienced by the AUV and nose cones, which were also tabulated on the spreadsheet for future comparison.

Analytical Comparison

The lift and drag force data collected for each object was used to calculate their Reynolds numbers, lift coefficients, and drag coefficients, via Equations 4, 1B, and 2B, respectively. Since drag was of particular interest, its force and coefficient was then plotted against Reynolds number at each speed for each object. Given the similitude between the AUV trials, and the uniformity of parameters between the nose cone trials, their results were overlaid (Figures 1, 2). Regression fits for the physical and simulated nose cones were used to calculate simulation error.

Experimental Results

As is evident in Figure 1, there was significant deviation between the COMSOL simulations and the wind tunnel trials. The aforementioned regression analysis revealed that for Nose Cone 3, the simulation error was on the order of 30%, and for Nose Cone 2, it remained around 50%, although it never surpassed 5% for Nose Cone 1. Upon creating similar plots for the measured and simulated drag experienced by the scale model and simulated AUV, respectively, it was found that the simulated values were an order of magnitude smaller than the measured values. It was hypothesized that there was an error in the simulation speeds calculated to achieve the same Reynolds numbers (see Equation 5B), where the 1:3 scale length was used in place of the actual length. Therefore, a correction was approximated by multiplying the COMSOL drag force and coefficient outputs by a factor of 9, as appears on Figure 1. There was once again a discrepancy between the physical and simulated results, primarily in the AUV's coefficient of drag.

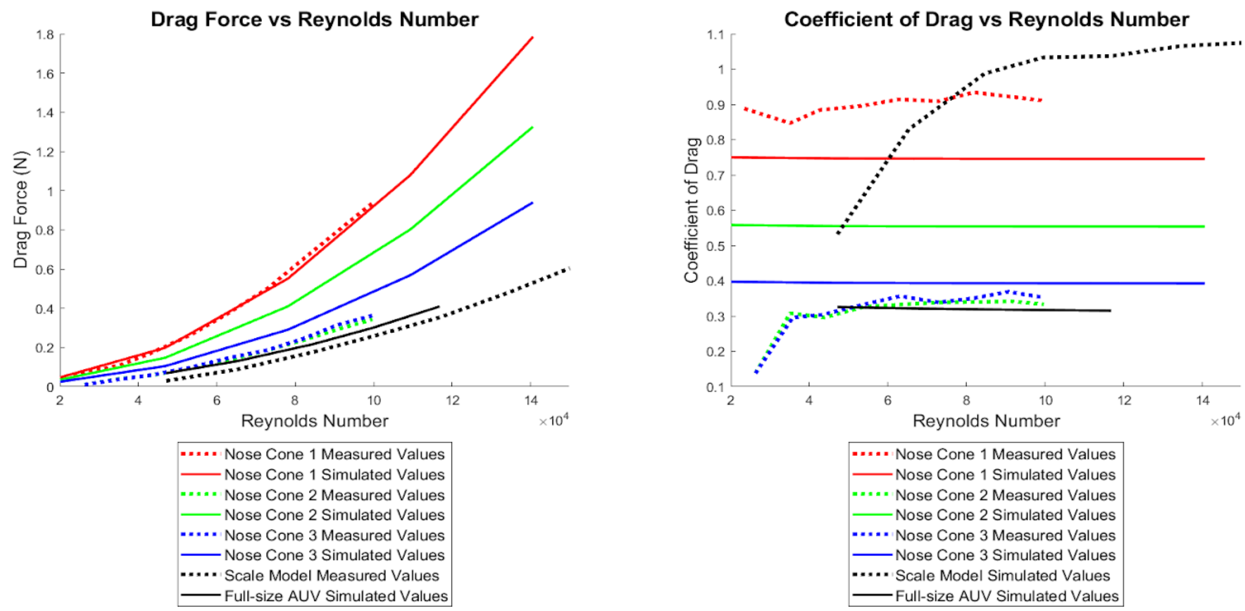

Figure 1. Drag force and coefficient of drag versus Reynolds number for each object.

Conclusion

This experiment highlighted the differences between fluid dynamics trials performed in a wind tunnel versus in COMSOL, due to various errors. For one, a small amount of air audibly escaped the test chamber. The behavior of fluids at the front and rear surfaces of the COMSOL tunnel differed from those in the physical wind tunnel, since the latter was self-contained rather than opening into empty space. Additionally, laminar flow was assumed when performing the simulations, and coarse meshes poorly approximated the objects' curvature. Future work should involve further measures to better seal both the physical and simulated tunnels. Testing the 1:3 scale model in a water tunnel might also yield results that cannot be anticipated from similitude in a wind tunnel. Though COMSOL does not necessarily faithfully calculate the actual forces that the AUV or any other object would experience in actuality, in the lack of a better alternative for CDF, it was agreed that COMSOL suffices in the absence of access to real-world testing. Furthermore, the experimental parameters in the wind tunnel were transferred via similitude to a full-scale COMSOL model of the AUV. The velocities it is expected to experience are similar to those used in the COMSOL trials (0.17 m/s to 0.42 m/s). Assuming the COMSOL simulations were sufficiently accurate, these findings reflect the drag that the AUV will experience as it navigates water at various speeds. Given an AUV mass of 3.225 kg, the acceleration due to the largest force (Figure 1) is merely 0.1268 m/s^2. Therefore, it was decided that the drag (and lift) experienced by the robot is small compared to its forward thrust, especially at lower velocities, and weight and buoyancy force. The measured drag forces and coefficients for Nose Cones 2 and 3 were extremely similar (Figure 1), suggesting that multiple approaches to designing the shape of the AUV could yield very similar results. Considering this, and that the drag force on the AUV is negligible compared to the other forces it will experience, it was concluded that no augmentations to its geometry or propulsion need be implemented, saving time and resources.

References

[1] N. Hall, “Rocket aerodynamics,” *Beginners Guide to Aeronautics*. Glenn Research Center, National Aeronautics and Space Administration, [online document], 2023. Available: Glenn Research Center, https://www1.grc.nasa.gov/beginners-guide-to-aeronautics/. [Accessed March 7, 2024].

[2] “E80 SP24 - Lab 6: Fluid Dynamics,” *hmc-e80.github.io*. https://hmc-e80.github.io/E80/labs/lab6/. [Accessed March 7, 2024].

[3] N. Hall, “Similarity parameters,” *Beginners Guide to Aeronautics*. Glenn Research Center, National Aeronautics and Space Administration, [online document], 2021. Available: Glenn Research Center, https://www1.grc.nasa.gov/beginners-guide-to-aeronautics/. [Accessed March 7, 2024].

[4] “14.6 Bernoulli’s equation,” *University Physics Volume 1*. University of Central Florida, [online document], 2016. Available: Pressbooks, https://pressbooks.online.ucf.edu/osuniversityphysics/. [Accessed March 7, 2024].

[5] N. Hall, “Reynolds number,” *Beginners Guide to Aeronautics*. Glenn Research Center, National Aeronautics and Space Administration, [online document]. Available: Glenn Research Center, https://www1.grc.nasa.gov/beginners-guide-to-aeronautics/. [Accessed March 7, 2024].