Lab 6: Fluids Technical Memo

E80
George Davis
Harvey Mudd College

Abstract:

The goal of the Fluids lab was to compare the results of simulated and tested drag forces and their corresponding coefficients on different objects. With these results concepts of similitude and dimensional analysis will be used to relate to a larger underwater robotics project like a submarine. The test subjects used were three model cylinders each with a differently shaped tip (cone, cylinder, and hemisphere). The results were gathered from a range of data over multiple wind speeds with COMSOL and in the HMC wind tunnel. The drag forces on the objects were measured using the LVDT-reported simulated values and then visualized by plotting against the calculated Reynolds number determined by the characteristics of the fluid. It was then determined that the simulated and tested results matched the shape of the curves but there appeared to be discrepancies in the exact lift and drag values. These differences can be attributed to unideal conditions in the wind tunnel compared to the ideal conditions in the software. For example, the geometry of each of the test subjects could not be exactly the reported shape which would affect the fluid flow around it.

Introduction:

The motivation for the fluids lab is to build skills in fluid mechanics of objects in simulation and in wind tunnel testing and use concepts of similitude to upscale to a larger project. To do this, several fundamental concepts are used to describe the fluid properties and the response of the object while the fluid is moving around it. The Reynolds Number is a very important dimensionless parameter that can be used to describe constant parameters of the fluid and the test object [2]. In this lab, the Reynolds Number (eq.1) will be the independent variable to the dependent Coefficients of Drag and Drag Forces of the test object (eq.2) [2]. The test subjects in question are a set of three cylinders with different nose attachments in the shape of a cylinder, cone, and hemisphere. Together these three parameters will effectively describe the relationship between fluid flow and the object.

$$Re = \frac{pUl}{\mu} \qquad \text{(Eq 1)}$$

where;
Re = Reynolds Number [dimensionless]
p = Density of the Fluid [kg/m^3]
U = Velocity [m/s]
l = Characteristic Length [m]
μ = Dynamic Viscosity [kg/m*s]

$$F_D = 0.5pV^2AC_D \qquad \text{(Eq 2)}$$

where;

F_D = Drag Force [N]
p = Density of the Fluid [kg/m^3]
V = Velocity [m/s]
A = Reference Area [m^2]
C_D = Coefficient of Drag

In order to relate the results we get from wind tunnel testing and simulation Buckingham's Pi theorem and the concepts of similitude will be used. This will allow the results of our three model cones to be scaled to a different-sized application such as a submarine. Using the dimensionless parameter such as Reynolds Number it will be possible to relate the results obtained to a different scale. However, these objects must have similarity. Similitude is achieved if the body dimensions share the same linear scale, the fluid streams are scaled and the ratio of lift and drag forces between each object is the same [2]. These three requirements are titled; Geometric, Kinematic, and Dynamic similitude respectively.

In this lab report, there will be three pertinent sections that will describe the setup, analysis, and conclusion of the lab. In the Experimental Method section, there will be a description of how the concepts previously mentioned will relate to our experiment and how they will be tested. The Results sections will contain the important figures and point out the relevant aspects of each plot. Finally, in the Conclusion, there will be an analysis of the meaning of the results discussed in the Results section.

Experimental Methods:

The Reynolds number and the coefficient of drag and drag force are the three values being measured in the lab. Several constant parameters are necessary to construct each of these pieces of data. The Density of the Fluid and Dynamic Viscosity were determined based on the ambient temperature of the wind tunnel and assumed to be 20℃ in simulation. Characteristics of the object such as the length and the reference area were predetermined by measuring the object itself. The reference area is the area of the plane orthogonal to drag force. The characteristic length is the length along the object in the direction of motion of the fluid [2].

To collect data in simulation COMSOL physics software will be used. The goal of this lab is to understand the fluid flow of air at 20℃ around the nose cones at a wide range of wind speeds (5 [m/s] to 45 [m/s]). The corresponding Reynolds number can be calculated using Eq.1 with the predetermined density (1.2 [kg/m^3]), invariable characteristic length (0.15 [m]), and dynamic viscosity ($1.8*10^{-5}$ [kg/m*s]) [1]. The results will be generated using a discrete mesh integration over the surface of each of the nose cones. This will output a drag force from which the coefficient of drag can be calculated with Eq.2. There reference area for each of those nose cones with simply be the cross-sectional area of the cylinder ($1.93*10^{-3}$ [m^2]) .

To collect data in practical testing a wind tunnel will be used. There will need to be a similar range of air velocities so the data will be comparable between test and simulation. Unfortunately, unlike COMSOL the wind tunnel is an unideal testing environment and there won't be the same control over previously exact parameters such as wind speed. In order to begin testing there needs to be a relation between wind speed and fan speed, a quantity we can measure. A calibration curve relating the RPM of the fan and the measured velocity via a Pitot-static tube. The Pitot-static tube converts a measured Differential Pressure to an Air Velocity using the Bournuli Equation Eq.3 [3].

$$V = \sqrt{\frac{2\Delta p}{p}} \qquad \text{(Eq. 3)}$$

where;
V = Air Velocity [m/s]
Δp = Differential Pressure [Pa]
p = Density of Fluid [kg/m^3]

When it comes to measuring the drag force on each of the three nose cones a Linear Voltage Displacement Transducer (LVDT) will report the values. Then, rearranging Eq.2 the coefficients of drag can be calculated.

Results:

The results will be presented in two figures that display two comparisons; differences between the three nose cones and between the simulation and wind tunnel testing. In the figures, each of the three nose cones has its own color with the simulated results in the solid line and wind tunnel tested results in the dashed line. Nose Cone #1, #2, and #3 refer to the cylinder, hemisphere, and cone respectively. The average differences between the simulated and tested results were calculated by finding the magnitude difference of the interpolated functions.

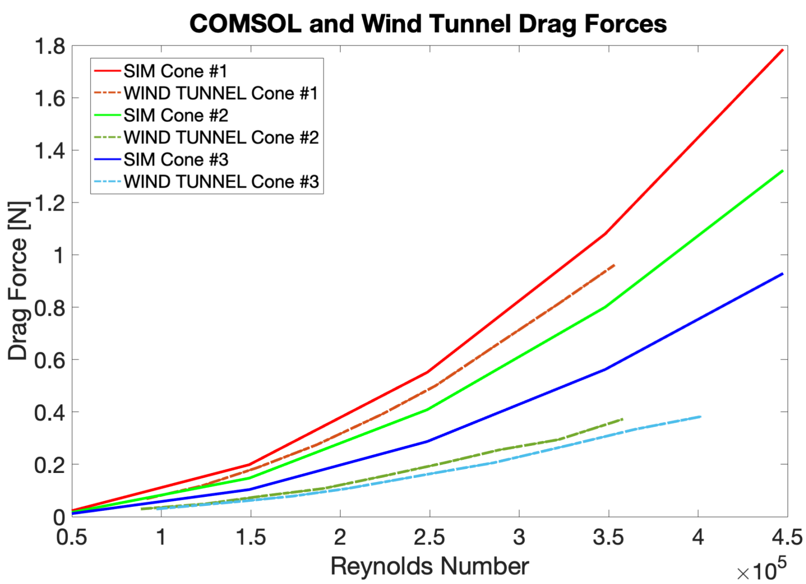

Figure 1: *Graph of Drag Force (eq.1) and Reynolds Number (eq.2) of the three nose cones in simulation and wind tunnel test chamber.*

Figure 1 visualizes the results of the simulated and tested Drag Forces (Eq.2) plotted against the Reynolds number (Eq.1). The nose cone that is most optimal for minimizing the drag force appears to be Cone #3 (cone nose) in both simulation and test. The nose cone that maximizes drag force between the subjects in Cone #1 (cylinder nose). The average difference in drag force for Cone #1 is 0.0727 [N], Cone #2 is 0.2115 [N] and Cone #3 is 0.1221 [N]. The difference between simulated and tested drag forces is greatest with Cone #2 (hemisphere nose). Additionally, the wind tunnel test results are substantially shifted downwards compared to the corresponding simulated results. This can be attributed to the several ideal assumptions made by COMSOL and the unideal circumstances of the wind tunnel. For example, with the simulation that was being run on the nose cones all geometry and texture of the objects were built to be ideal. Meanwhile, in the wind tunnel the nose cones were machined to precision but still had non-uniformities that could lead to different results. Furthermore, the wind tunnel introduces a substantial amount of error in the measurement devices available to users.

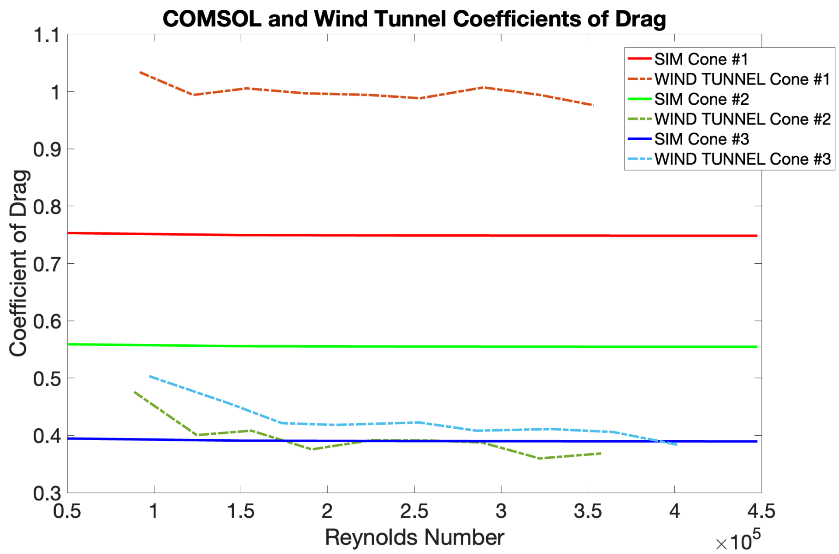

Figure 2: *Coefficient of Drag (eq.1) and Reynolds Number (eq.2) of the three nose cones in simulation and wind tunnel test chamber.*

There appear to be major discrepancies in Figure 2 between the simulated and wind tunnel-tested coefficients of drag. The order of greatest to least coefficient of drag in simulation is Cone #1, Cone #2, and Cone #3 but in the test, it is Cone #1, Cone #3, and Cone #2. The average difference in coefficient of drag for Cone #1 is 0.2483 [N], Cone #2 is 0.1657 [N] and Cone #3 is 0.0.0386 [N]. The greatest difference between the simulated and tested coefficient of drag is with Cone #1. Again, these differences can be attributed to several unknown sources of error from the wind tunnel along with the ideal conditions of the simulation. These sources of error can include, variations in ambient temperature, air density, dynamic viscosity and rough placement onto the subject mount on the wind tunnel.. The cone that minimizes coefficient of drag and travels with the least resistance through the air appears to be Cone #3 (hemisphere nose). The cone that maximizes the coefficient of drag and travels with the most resistance is Cone #1 (cylinder nose).

Conclusion:

Judging from the results gathered we can conclude that Cone #3 or the cone nose is the most optimal for minimizing drag force and coefficient of drag. In both figures 1 and 2 the Cone nose has the lowest of each of the studied variables. This means that this nose cone travels through the air with the least amount of drag and will need to be used in order to minimize the fuel consumption of our underwater robotics application. However, it seems that Cone #2 or the hemisphere nose is a close second and may need to be considered when finding the optimal shape of our applied design. Some compromise between the two rather polar geometries could achieve a more optimal shape and more testing will need to be done to confirm this.

Since the objects and tested fluids share similitude with an underwater robotics application like a submarine the results found here can be scaled using Buckingham's Pi theorem and dimensional analysis. This is a window into the power of small-scale testing in fluid mechanics problems. It was determined in a lab that some hybrid of a cone and hemisphere nose object would minimize the drag force of the fluid flow. This conclusion can be applied to different fluid flow applications and yield the same results.

For future experiments, it would be interesting to test the validity of the hypothesis that came out of the conclusion. Are cone/ hemisphere hybrid noses the least resistant objects to fluid flow? Additionally, designing an applicable submarine or torpedo device that Buckingham's Pi theorem could be used to relate the lab results to would be interesting to see how accurate it is in real life. What was learned in writing this lab report was how to holistically communicate a complex message to a reader with figures and equations.

References

[1]Engineers Edge, "Viscosity of Air, Dynamic and Kinematic | Engineers Edge | www.engineersedge.com," *Engineersedge.com*, Jun. 11, 2018. https://www.engineersedge.com/physics/viscosity_of_air_dynamic_and_kinematic_14483.htm

[2] M. Blakeslee, "Similitude and Non–Dimensional Numbers," *help.altair.com*. https://help.altair.com/hwcfdsolvers/acusolve/topics/acusolve/training_manual/similtude_and_non_dim_numbers_r.htm#:~:text=Non%E2%80%93Dimensional%20Numbers - (accessed Mar. 07, 2024).

[3]"Pitot Tubes," *www.engineeringtoolbox.com*. https://www.engineeringtoolbox.com/pitot-tubes-d_612.html#:~:text=Pitot%20tubes%20can%20be%20used