2023 Annual Conference & Exposition
Baltimore Convention Center, MD | June 25 - 28, 2023
The Harbor of Engineering Education for 130 Years
Paper ID #39128

Measuring the drag forces on Corvette car model

Dr. Manohar Chidurala, Western Kentucky University

Manohar Chidurala teaches Thermal-fluid sciences course sequence, Instrumentation and Experimentation, and Mechanical Systems at WKU.

Riley Bishop
Mr. Brandon Charles Sekula, Western Kentucky University

Mr. Sekula is a rising senior at Western Kentucky University (WKU), studying Mechanical Engineering with a focus on Thermo-Fluids. He conducts industry related research and assists in organizing and maintaining WKU's Material Science laboratory. He is conducting his own research in conjunction with Dr. Benjamin Dinan on the classroom clay produced by WKU's ceramics department, testing the compression strength, and identifying the mechanical properties through the manufactured clay's microstructure.

Mr. Nathaniel Burris, Western Kentucky University

Nathaniel Burris is a rising senior at Western Kentucky University where he studies Mechanical Engineering. There, he helps with undergraduate research in the Thermal-Fluids Laboratory. He is currently an engineering intern at Globe Electric. He is the vice president of the ASME student chapter at WKU, a member of Tau Beta Pi, and an Eagle Scout.

Measuring the Drag Forces on Corvette Car Model

Abstract

Western Kentucky University (WKU) currently owns a Hampden model H-6910 wind tunnel, a model designed for educational purposes. This model operates at a maximum air velocity of 50 miles per hour. The H-6910 wind tunnel also contains a test section of only 8 inches by 8 inches. Over the past two years, the undergraduate students developed a data acquisition system through LabVIEW to analyze the aerodynamic forces on the external bodies with the help of senior design projects. For this study, an alternative system using force balance was developed to compare the accuracy of the data acquisition system.

The goal of this study was to demonstrate the capabilities of the WKU's Wind Tunnel by measuring the aerodynamic forces on external bodies such as the Cylinder and Corvette Car Model (small-scale, 3-D printed) using a newly developed Force Balance system. This was done by conducting experiments using the wind tunnel in the fluids lab and comparing these results to computational models. For this experiment, a 3-D printed car and a cylinder were used for testing in the wind tunnel test section and a pressure transducer was used to measure the free-stream velocity. A force balance was then utilized to securely mount objects in the wind tunnel while directly measuring the total lift and drag forces. In a parametric study, the validation of the force balance system was carried out for a cylinder in which the force balance results were verified with the computational fluid dynamics (CFD) data. Then the computational model was developed to simulate the fluid flow over Corvette Car in the ANSYS workbench and then it was utilized to compare the Force balance (experimental) results. In the end, the results from all parts of this research were compared to one another to validate the performance of the force balance system.

This research will provide useful knowledge to the Mechanical Engineering program at WKU and give students valuable research experience in the field of aerodynamics. Students will be capable of understanding the applications of what they learn in the classroom to solidify the educational background that they are receiving at WKU.

Keywords

Aerodynamics, Experiments, CFD, Wind Tunnel, and Undergraduate Projects.

Nomenclature

ρ	Density of fluid
V	Velocity of fluid
μ	Viscosity of fluid
L_C	Characteristic Length of an object
Re	Reynolds Number

A Projected Area of an object
F_D Drag Force
F_L Lift Force
C_D Coefficient of Drag
C_L Coefficient of Lift

Introduction

In numerous engineering applications, external airflow induces significant aerodynamic effects that practicing engineers must consider. A few examples of these applications are fuel efficiency for automobiles and aircraft, and wind turbine blade analysis. Although the development of Computational Fluid Dynamics (CFD) allows researchers to perform simulated studies of external flow over objects, physical experiments remain invaluable to the data collection and analysis process. Physical experiments often reveal results that vary from computational models. The typical method of performing these experiments is the wind tunnel, a device that uses a fan to produce uniform airflow through a duct that contains a test section, an area in the middle of the tunnel where the test object is placed.

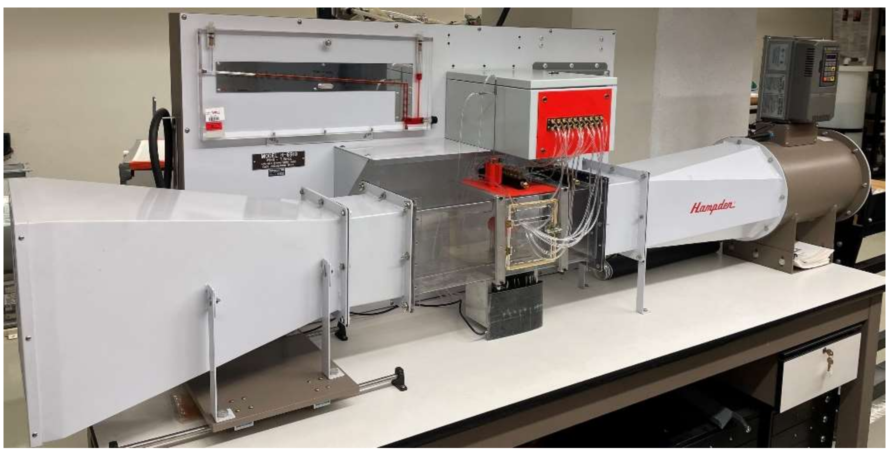

Figure 1. Hampton H-6910 Wind Tunnel

Western Kentucky University's Thermo-Fluids Laboratory currently possesses a Hampden Model H-6910 wind tunnel, shown in Fig. 1, designed for educational purposes. A fan on the right drives ambient air through the wind tunnel from the left to the right. Air enters the wind tunnel from the nozzle on the left. The air passes through the test section, the clear section in middle, in uniform flow. Uniform flow implies that the flow does not vary across the cross-sectional plane of the test section. It is crucial to achieving uniform flow because this condition occurs most often in realistic aerodynamic applications. After the test section, the air exits into the diffuser and through the fan to the right.

As a Senior Capstone project for the mechanical engineering program in 2021, a team of five undergraduate students worked with a faculty advisor to utilize the wind tunnel. They developed a data acquisition system to experimentally determine the aerodynamic properties of the lift and drag force of objects in external flow [1], [2].

Figure 2 shows below a two-dimensional airfoil with air flowing from left to right. This illustrates the concept of lift and drag forces.

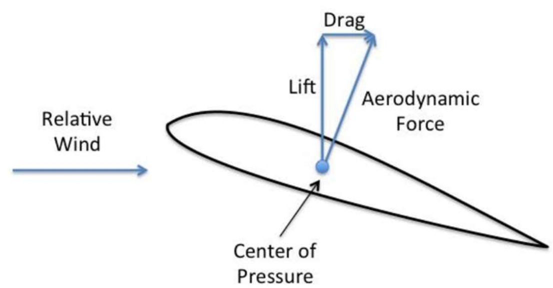

Figure 2. The Visualization of Lift and Drag Forces on an Airfoil

The airflow applies an aerodynamic force on the object that can be divided into two separate vector components: lift force and drag force. Lift force is an upward force perpendicular to the flow direction, and drag force is a parallel force to the flow direction acting in the same direction as the flow.

The total drag force acting on an object is the sum of two components: pressure drag and skin friction drag. The pressure on any point on the object applies a force perpendicular to the surface at that point; thus, pressure drag occurs due to a higher-pressure distribution at the front of the object relative to the back of the object (in relation to the flow direction). Skin friction drag occurs because of the frictional force imposed tangentially on the surface of the object by the flow.

On blunt bodies, such as the objects tested in this experiment, pressure drag is typically more significant than friction drag by a large margin, so the pressure distribution experimental method can be considered an appropriate approximation with a negligible skin friction force. However, an experimental method of directly measuring the total lift and drag forces would validate the accuracy of the results by comparing the total drag force to the pressure drag to determine the significance of skin friction drag in the experiment. As an honors supplement to the Capstone project, a force balance was developed as a direct force measurement system [3] based on the study of [4].

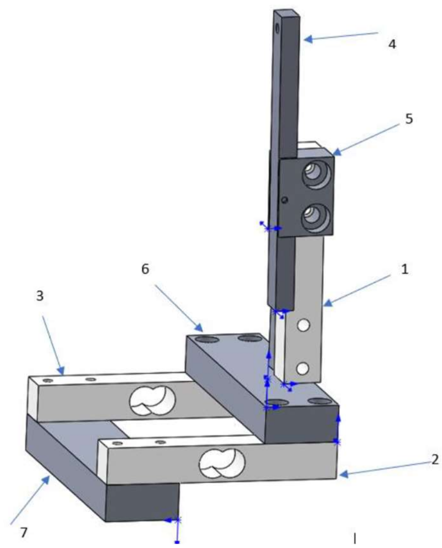

Figure 3. The Design of the Force Balance System

Working Principle of Force Balance System

The force balance utilizes three 1kg Load Cells. Two measure the lift forces (items 2 & 3 in Figure 3), and the third measures the drag forces (item 1 in Figure 3). These load cells are connected to an Arduino microcontroller that can convert the voltage differentials into meaningful values. The Arduino then transmits this data to Microsoft Excel using the data streamer add-in.

The drag load cell directly measures the drag on the test object without the need to adjust the readout value. To contrast, the readout from the lift load cells must be adjusted because the lift load cells counteract the moment created by the drag force due to how static mechanics work.

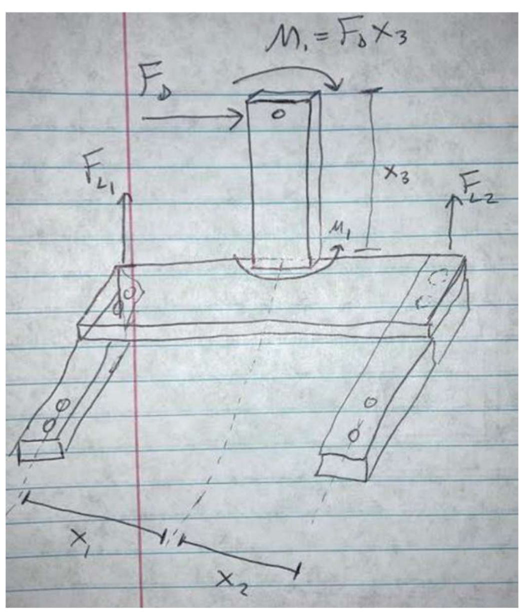

Figure 4. Statics Diagram for Drag Moment Adjustment

Using the statics illustrated in Figure 4, the adjustments for the lift force are [3]:

$$F_{L,1,adj} = F_{L,1,meas} - F_D * \frac{x_3}{x_1+x_2} \quad \textbf{(Eq. 1)}$$

$$F_{L,2,adj} = F_{L,2,meas} + F_D * \frac{x_3}{x_1+x_2} \quad \textbf{(Eq. 2)}$$

Physical Models

The test bed used is a Hampton H-6910 Wind tunnel with a test section of 23 in x 8 in x 8 in. A cylinder of 1.6 in diameter was tested, as shown in Figure 5, to find the lift and drag forces acting on a 3D-printed body. This was done with a flow velocity of 10.06 m/s, considering laminar flow conditions.

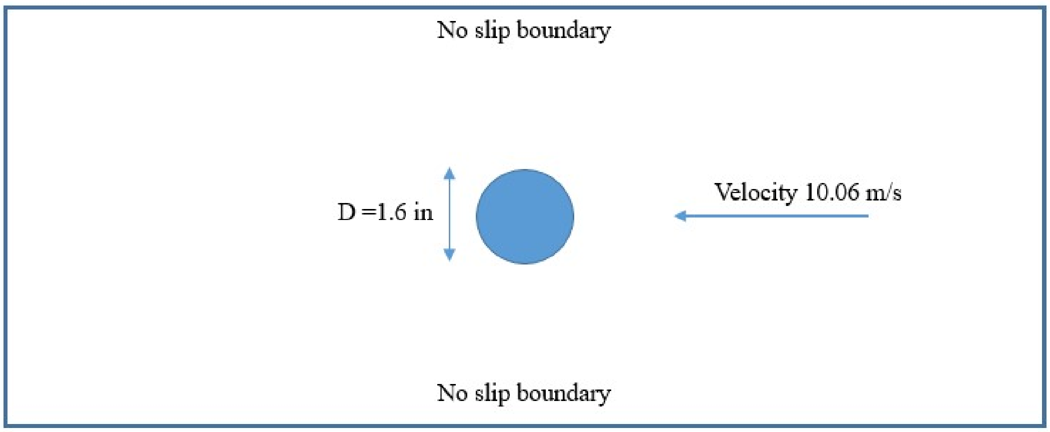

Figure 5. The Physical Model for the Cylinder

The modeling shows the no-slip boundary conditions on the walls of the test bed, as well as the direction of the flow velocity.

The same enclosure dimensions as well as velocity and no-slip conditions were used in the Corvette C8 simulations as shown in Figure 6.

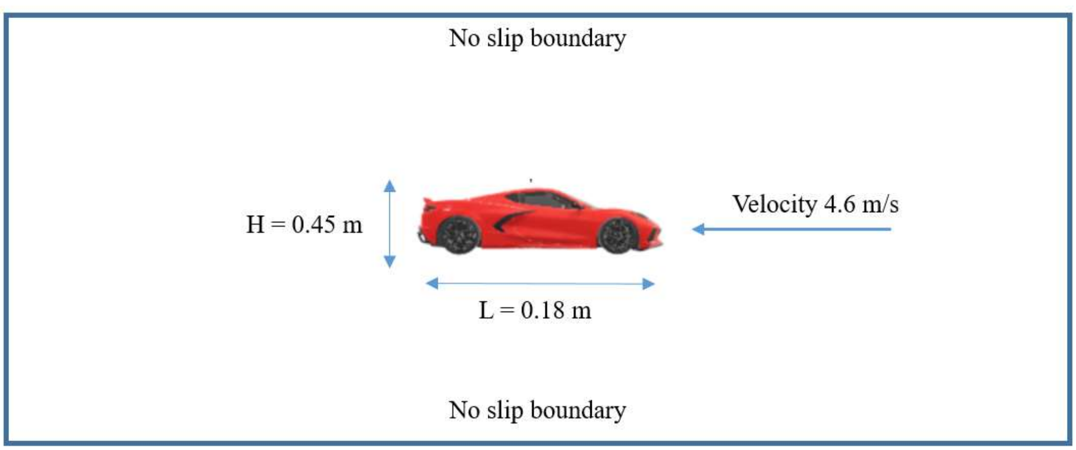

Figure 6. The Physical Model of the Corvette Car Model [6]

CFD Results

Utilizing ANSYS Fluent 2D simulation software, a simulation was created using a two-dimensional cylinder. This is applicable due to its symmetrical nature in three dimensions. The meshing around the cylinder was a biased meshing that allows for greater precision around the exterior of the cylinder, as shown in Figure 7.

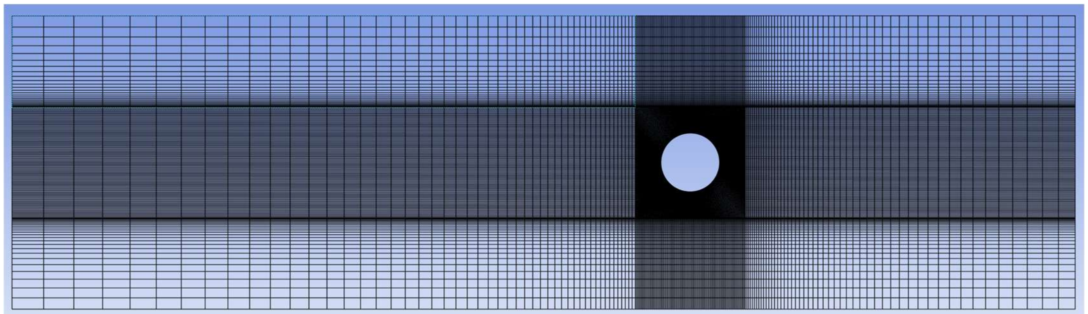

Figure 7. Bias Meshing around Cylinder to Increase Simulation Accuracy

A steady state with a wind velocity of 10.06 m/s was used to determine the aerodynamic forces on the cylinder. This allowed the flow to be laminar to negate vortices effects. Air at constant density, ρ = 1.225 kg/m^3, and viscosity, μ = 1.784 * 10^{-5} kg/m*s, was used to help establish the Reynolds number using Eq. 3, Re = 2.8 x 10^4.

$$R_e = \frac{\rho V L_c}{\mu} \qquad \textbf{(Eq.3)}$$

Validating the Reynolds number used in the simulation allowed for the comparison of the results gathered during the simulation to those of published values [4].

$$C_D = \frac{F_D}{\frac{1}{2}\rho V^2 A} \qquad \textbf{(Eq. 4)}$$

$$C_L = \frac{F_L}{\frac{1}{2}\rho V^2 A} \qquad \textbf{(Eq. 5)}$$

The values used for verification were the Coefficient of Drag, C_D = 0.828; Drag Force, F_D = 2.56 N; and Lift Force F_L = 0.112 N. These values are then used to compare and verify experimental results as well as published values [5]. The coefficient of lift is considered to be negligible as the symmetrical shape showed the same values above the cylinder as were under. Figure 8 illustrates that the coefficient of lift for the simulated cylinder fluctuated about zero during the simulation. This permitted the establishment of the numerical simulation results as reliable. With this parametric study, the research could move forward to determine the drag coefficient of the model car using both experimental and CFD studies.

Due to the limitations imposed by the student version of ANSYS 2023 R1, a simplified CAD model had to be constructed using the profile of the Corvette C8. Furthermore, a symmetry cut of the car model had to be employed for the meshing because ANSYS 2023 RI imposed another limitation of a face and edge limit of only 512k. The full model used surpasses this limit when using the whole model but not half of the model.

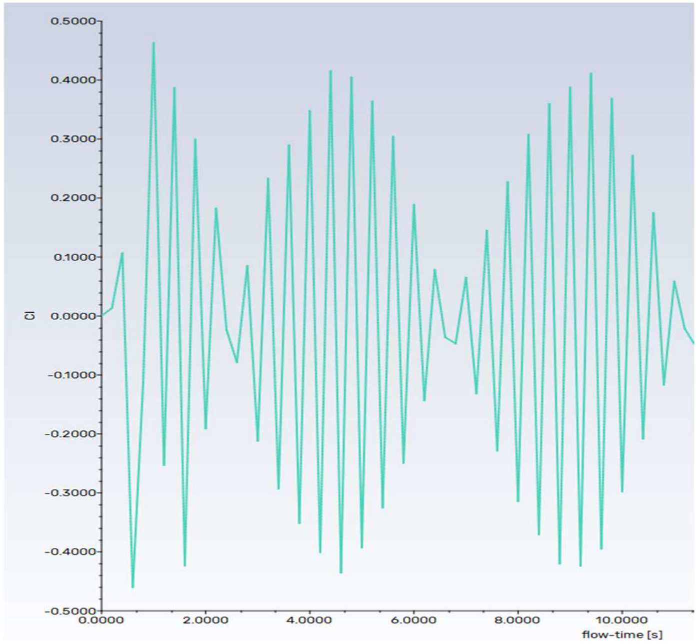

Figure 8. Coefficient of Lift for the Cylinder in ANSYS over time

To find the correct speeds to run the simulation, dimensional analysis was performed using constant Reynolds number calculations to find the correct velocity needed due to its change in characteristic length. This velocity was found to be 4.6 m/s to match the experimental velocity of 8.03 m/s. This was computed from a scaling factor of 1.799. This was not only done to achieve similarity but also to verify that the Reynolds number and drag and coefficient values respectively would be correct. In conjunction with the similarity analysis, it was assumed the air had constant density and viscosity in order to maintain consistency with experimental conditions.

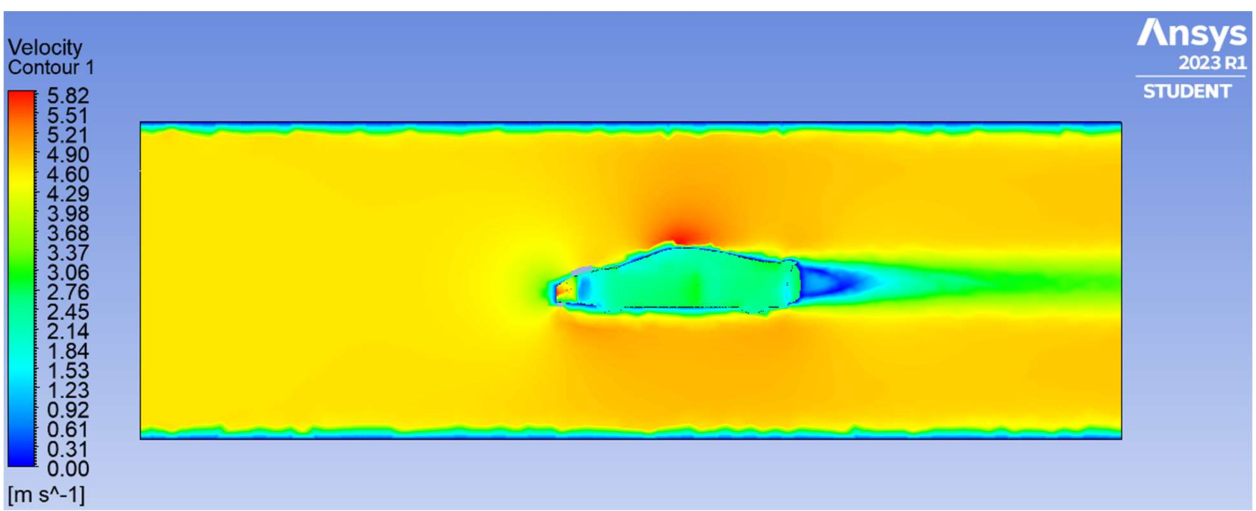

Figure 9. Velocity Profile on Corvette C8

A velocity profile was simulated to get an appropriate visualization of the flow around the car to match with expected results. The highest velocity recorded was at the top of the Corvette C8, this is due to the lower pressure, as shown in Figure 9. The velocity profile gives a good representation of how the car is going to experience drag and lift forces. A low-pressure zone can be seen behind the vehicle where velocity is at zero in Figure 10. Low-pressure zones want to equalize the surrounding pressure creating a centralized location of drag. The CFD simulation returned a drag force of $7.68 * 10^{-3}$ N on the vehicle half, 0.045m. A coefficient of drag was calculated utilizing Equation 4, this value is 0.351. This value has a percent error of 17.1% when compared to the published value of 0.3 [5] for a sports car.

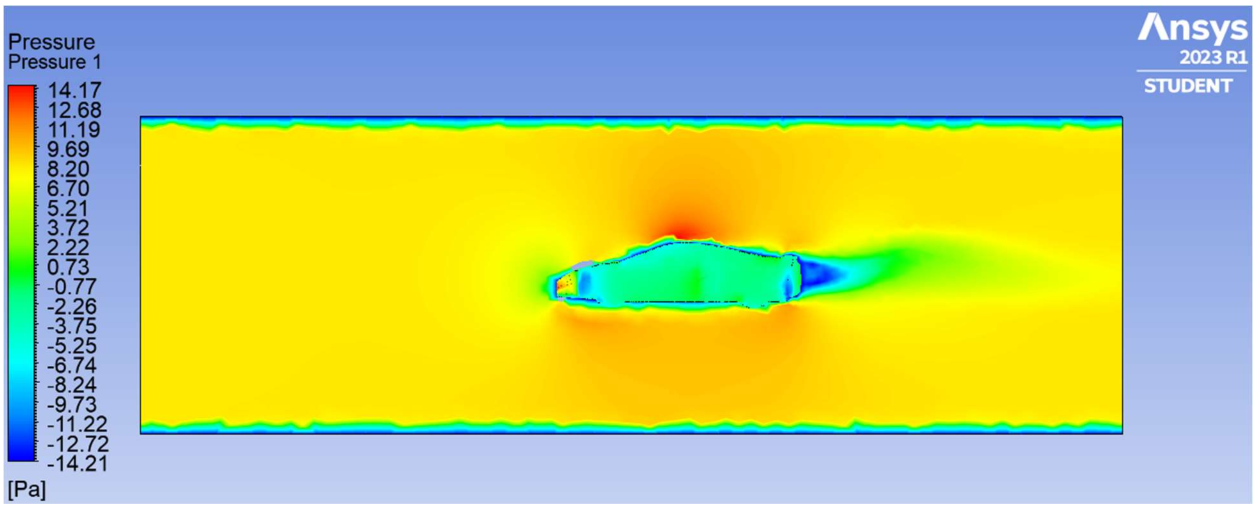

Figure 10. Pressure Profile on Corvette C8

Experimental Results

To confirm that the force balance is working properly, a 1.6 in diameter by 2 in long cylinder was chosen as a test object to be subject to a known wind speed. The cylinder was attached to the force balance and subjected to a wind speed of 10.06 m/s. Once the flow had fully developed, data from the force balance was collected for about one minute. The average drag force on the cylinder was calculated, F_D = 2.94 N. The average total lift, F_L = -0.226 N, was also calculated. The lift forces were properly adjusted (per Eq. 1 & 2) and then summed to find the total lift. The force balance has some drift and is quite noisy.

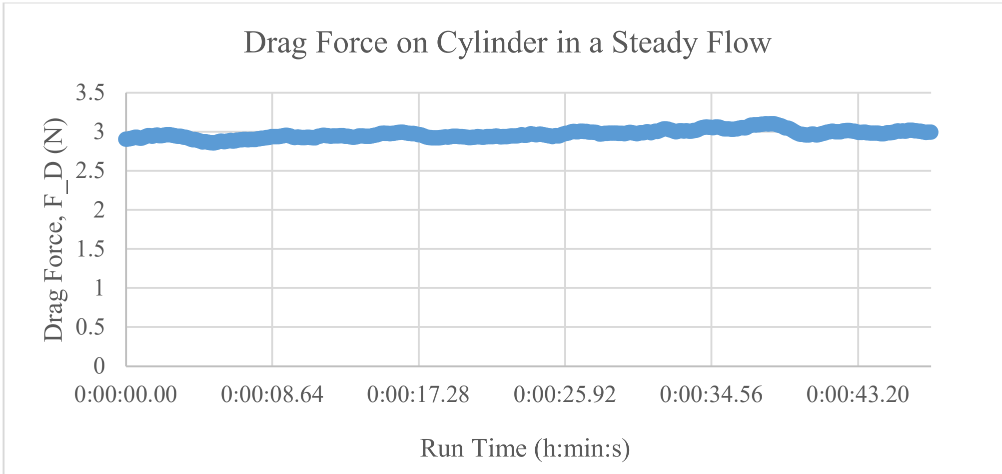

Figure 11. Drag Force on the Cylinder over Time

Figure 11 clearly illustrates the noise and drift in the force balance. It was noticed that the box holding the force balance was vibrating. These vibrations affected the values that the force balance read out. The peaks and troughs in Figure 11 are likely from these vibrations. The data also has a slight drift to it. It goes from 2.91 N to 2.99 N over a period of 48s. Taking the average over the period took care of the noise and this small drift for the purposes of comparing forces to simulated values.

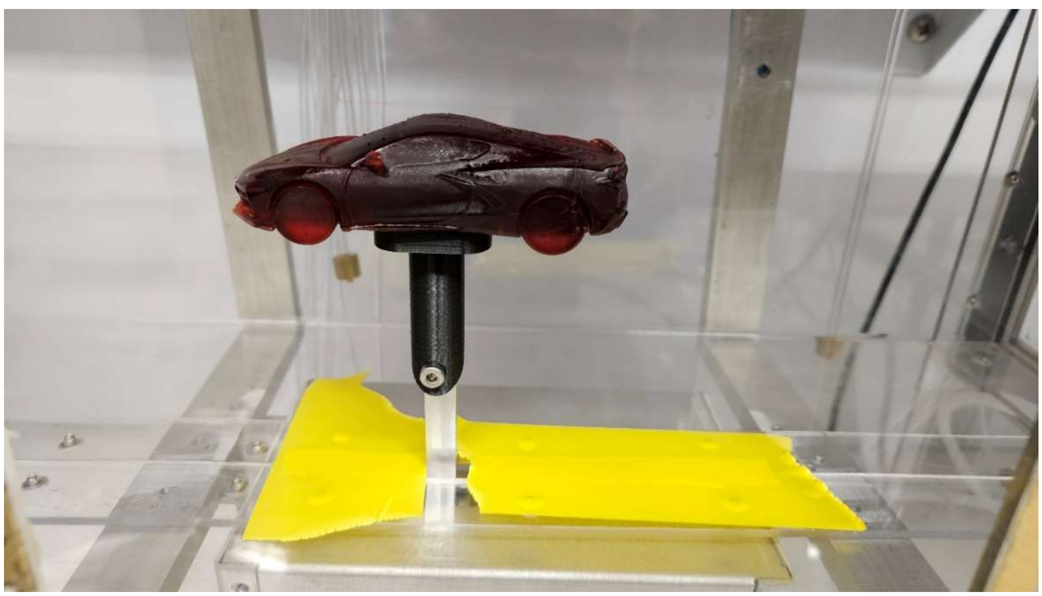

Figure 12. The Scale Model Corvette Mounted in the Wind Tunnel.

The scale model corvette was placed into the wind tunnel attached to the force balance as shown in Figure 12. The wind tunnel was run at 8.03 m/s. This slower speed was chosen due to the limitations of the wind tunnel and force balance. At this speed, the whole system, scale model corvette and stand, experienced 0.0806 N of drag.

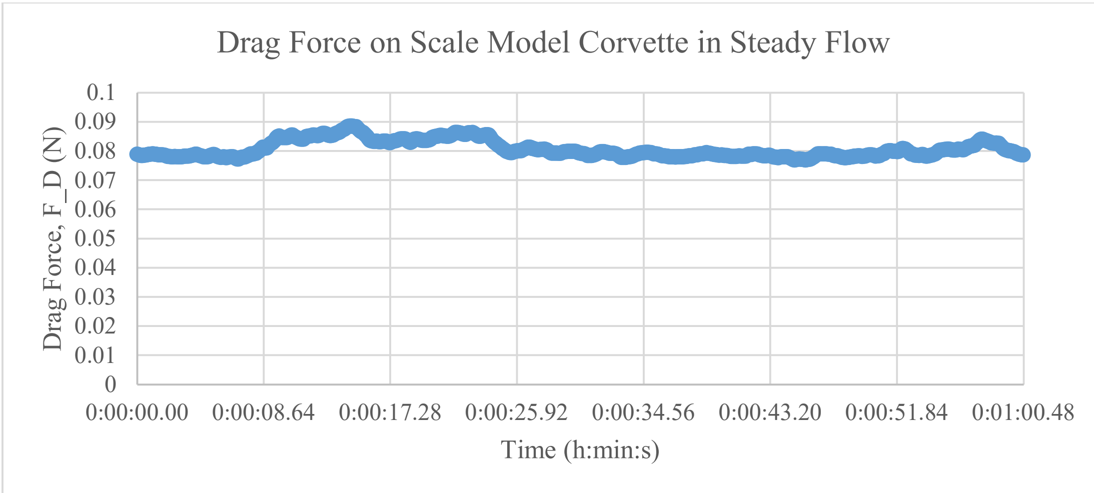

Figure 13. The Drag Force on a Scale Model Corvette over Time

Figure 13 demonstrates that force on the scale model corvette was fairly consistent over the one-minute time period. The increased value from the 8 s to 24 s balanced out by the rest of the data over the time period because the average value for this whole time period was taken as the drag force.

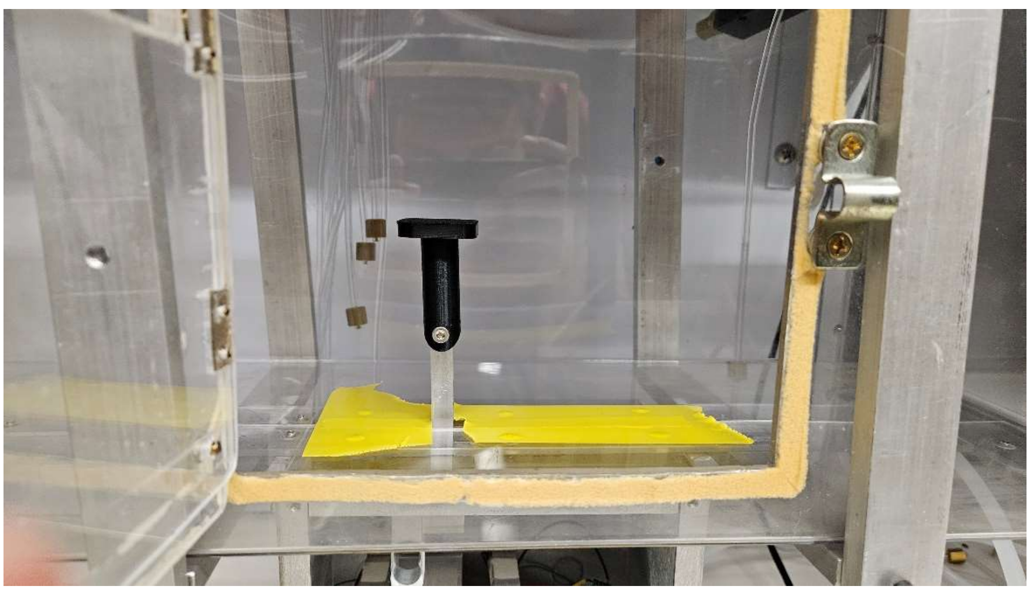

Figure 14. Only the Stand in the Wind Tunnel

A second set of force measurements was taken with just the stand, as shown in Figure 14, subjected to the 8.03 m/s airstream in the wind tunnel. This was done in order to calculate an offset for the drag coefficient.

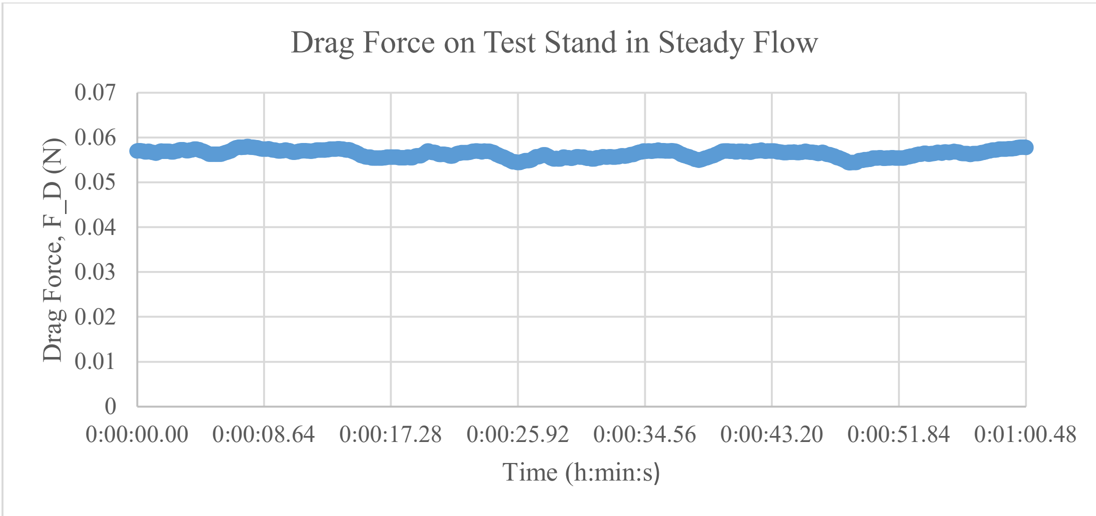

Figure 15. Drag Force on Test Stand over Time

The average drag force on the test stand was 0.0564 N. Figure 15 demonstrates that the drag force on the test stand was steady with time.

Using Equation 4, these two force values were then used to calculate the drag coefficient for both the whole system and only the test stand, as presented on Table 1.

Table 1. Coefficients of Drag Data

Test Object	A (cm^2)	F_D (N)	C_D
Scale model corvette and test stand	23.8	0.0806	0.860
Test stand only	12.6	0.0564	1.13

The difference between these two coefficients of drag was taken as the coefficient of drag for the scale model Corvette. The coefficient of drag for the scale model Corvette is 0.273. Using the published value for coefficient of drag of sports car [5], the scale model has a percent error for the coefficient of drag is 8.88%.

Conclusions

In summary, the objectives of this undergraduate project, to provide valuable laboratory and research experience to the students in the aerodynamics area, were achieved with the development of an alternative measurement system for measuring the drag and lift forces of blunt bodies.

The simulation predicted a drag force of 2.83 N on the cylinder. The experimental results showed a drag force of 2.94 N on the cylinder. The percent error with respect to the CFD results between these two values is 3.88%. The experimental drag did not consider the drag of the test stand. Despite this the two values agreed with one another between the CFD and experimental results.

In scale model corvette experiments, the drag force of the down stem was measured separately and considered. This allowed for a closer simulated and experimental value. It was the objective of the initial testing to get values within a reasonable error, allowing for experimentation with the Corvette model car. From the simulation of the Corvette C8, a drag coefficient of 0.351 was calculated from the simulated drag force. Multiple iterations of the simulation were used to verify the repeatability of data. The experimental results yielded a drag coefficient of 0.273. The CFD and experimental results have percent errors with respect to the published value of 17.1% and 8.88% respectively. One possible reason for these larger errors is comparing to the drag coefficient of a generic sedan or sports car [5] instead of the drag coefficient of the specific model tested (a Corvette C8). Even though the specific value was not utilized for comparison, the drag coefficients are pretty close but not within the ideal 5% range.

In the future, the force balance system will be utilized to study the deflections and bending moment of a stem and compared to both the analytical and fluid-structure interaction analysis. Also, the results of the force balance system will be compared to that of the data acquisition system at different speeds.

References

[1] R. Bishop, W. Fisher, A. Doom, E. Hollingsworth, B. Mazzoni. and M. Chidurala, "Wind Tunnel Data Acquisition System," in *IMECE 2021, Virtual Conference, November 1-5, 2021*.

[2] A. Doom, E. Hollingsworth, R. Bishop, W. Fisher, B. Mazzoni and M. Chidurala, "Verification and Validation of a Small Wind Tunnel Data Acquisition System," in *IMECE 2021, Virtual Confrence, November 1-5, 2021*.

[3] R. Bishop, "Wind Tunnel Data Acquisition System Development using LabVIEW," Honors thesis, Mahurin Honors College, Western Kentucky Univ., Bowling Green, KY, 2021.

[4] M. Tomin, M. Scipioni, and B. Gatti, "Design, construction and testing of a 3-component force balance for educational wind tunnels in undergraduate aerodynamics," *Journal of Aviation/Aerospace Education & Research*, vol. 29, no. 1, 2020.

[5] Y. Cengel and J. Cimbala, *Fluid Mechanics: Fundamentals and Applications*. 4th ed. New York, NY: McGraw-Hill Education, 2018.

[6] D. Holiday, "2020 Chevy Corvette C8 - Download Free 3D Model by David_holiday." *Sketchfab*, 17-May-2021. [Online]. Available: https://sketchfab.com/3d-models/2020-chevy-corvette-c8-5f143d22b5ca43aeb90e85f19940d5e8. [Accessed: Mar-2023].