Development of a Drag Coefficient Laboratory via Capstone Design

Mir Atiqullah
Associate Professor, Mechanical Engineering Technology.
Southern Polytechnic State University
Marietta, GA 30060

Norman Russell
Associate Professor, Mechanical Engineering Technology.
Southern Polytechnic State University
Marietta, GA 30060

MIR ATIQULLAH

PhD in Mechanical engineering from Purdue University in 1996.
Research areas include computational optimization in design, genetic algorithm and simulated annealing, parallel supercomputing, engineering education.
Teaching a broad range of engineering courses including machine design, materials science, engineering mechanics, manufacturing, and fluid mechanics.
Professional experiences include 10 years of industrial and 17 years of teaching engineering and technology areas.

NORMAN RUSSELL

PhD in Chemical Chemical Engineering from Institute of Paper Chemistry in 1965.
PE from Alabama.
Research areas include paper chemistry, engineering education.
Teaching a broad range of engineering courses including thermodynamics, fluid mechanics, and laboratory courses.
Has 29 years of industrial and 17 years of teaching experience.

Development of a Drag Coefficient Laboratory via Capstone Design

Abstract:

The concept of aerodynamic drag is fundamental to fluid dynamics. A drag force experienced by an object is related to drag coefficient, a dimensionless parameter. The effect of important factors, geometric shape of the object, air properties, and wind speed on aerodynamic drag is traditionally demonstrated in a Wind Tunnel.

In a standard fluid dynamics laboratory, the relationship between object shape and drag coefficient is vividly illustrated using a few distinctly different solid shapes: flat disc, hemisphere, sphere, and teardrop. The effect of stream lining an object shape is clearly demonstrated. In order to make the laboratory experience more related to real-world and more interesting to students, it was proposed to use model cars in wind tunnel in addition to the basic shapes. Three automobile models were selected: Ford Model T, Chevrolet El Camino, and 1970's Porsche. The models represent three distinct levels of stream lining. A group of senior students in the capstone Machine Design class was assigned to develop CAD models as well as scale model prototypes of these cars. Using the CAD model and a three-dimensional printer, physical scale models of the three autos were produced. A mounting technique was designed to secure the models in the wind tunnel. Flow simulation in Computational Fluid Dynamics (CFD) environment was employed to predict the Drag coefficient of each model. The predicted values fell within expected range of published values.

In the design phase the models were smooth; so the surfaces were finished to mimic the design condition. The models were tested in the wind tunnel at various wind speeds. Air temperature was monitored. Model frontal areas were available from the CAD design. Recorded data were used to calculate drag coefficients for comparison with predicted CFD

values. Percent error between predicted and experimental values ranged from 4.65% to 9.52%.

The whole process of developing CAD models, fluid dynamics simulation, rapid prototyping, wind tunnel testing and drag coefficient comparison generated in depth understanding and interest about real world fluid dynamics and especially drag.

The design process and elements will be presented to students in future fluids labs, and students will experiment with drag coefficients for the models to foster more interest and understanding in fluid dynamics and drag.

Introduction

Drag coefficient[1,2] is a characteristic value of a solid object moving in a fluid such as an airplane flying or a car cruising. It could be also be effect of fluid flowing over or around a stationary object like a building or a sail or a wind turbine. It is a measure of the resistive force that is developed due to geometry, orientation of the object and of course velocity and characteristics of the fluid such as density and implicitly viscosity. The drag coefficient consideration is very important in reducing drag in automobiles. For an average vehicle about 50-60% of power is used to overcome the aerodynamic effects to cruise on highway. Of course at low speeds it is the rolling friction of tires that consumes most power. A streamlined vehicle shape that generates low drag force is very important to get better fuel economy of vehicles at cruising speeds. The drag force varies directly as the square of the relative speed, given everything constant, such as geometry, orientation,, flow direction, object size, fluid density and viscosity. The equation for drag force F_d is given in equation 1 below,

$$F_d = -\frac{\rho v^2 C_d A}{2} \qquad (1)$$

Where,

ρ is the density of the fluid,

v is the relative velocity,

and A is the reference area.

It is important to note that the reference area for vehicles would be the frontal projected area. For other objects it could be the planform area such as that for an airfoil and it is the wetted area for a submerged object in flowing liquid.

The characteristic drag coefficient C_d can thus be calculated by equation 2, as

$$C_d = \frac{2F_d}{\rho v^2 A} \tag{2}$$

It is a dimensionless quantity and thus characteristics of the object in question.

If the fluid is compressible, such as air, the drag force is dependent on the Mach number $M=V/c$, where V is the relative speed and c is the speed of sound or wave propagation through that medium. For automobiles this Mach number factor is negligible because of low speeds relative to that of sound and can be ignored.

The Design Process

The capstone design project is taught in combination with the Machine Design[3] class. While the credit hour and time allocated for the project is not at par with the standard capstone class, the faculty and the design groups take the extra time and puts in extra effort in the project. This is the final opportunity for the students to apply their science, technology, engineering as well as mathematics (STEM) background in a project for which they will get credit while they enjoy the satisfaction of solving an engineering problem and demonstrate it.

The project is approached in stages as follows:

1. Problem statement from the customer or sponsor.
2. Design statement developed by the design group.
3. Customer requirements provided by the customer or sponsor. It is list of requirements and functionality that is expected from the solution or product. These requirements are

mostly non numerical but some could be measurable, if the customer so specified. Customer may also specify a separate ‘wish list’ that would be nice to achieve but not a condition of successful design.

4. Design Specification developed by the design group. This is in response to the customer requirements, fulfilling each and every item of the customer requirement list. These are mostly measurable outcomes or performance indices of the solution or product. It could be comparative as well in some appropriate situation.
5. Conceptual solutions developed by individual team members or as a group or sub-group. Often brain storming sessions are arranged to generate ideas and solutions.
6. Concept finalization and selection based on conceptual solutions developed, often combining ideas.
7. Embodiment design. This stage is the reign of engineering calculation, analyses and simulation for shape, size, feature, fit, assembly, materials, recycling etc.
8. Prototyping of the final design. This stage is to demonstrate, by using a prototype of the final design, that the final design meets the design specification, and thus meets the customer requirements. Most of the times our design groups are expected to develop a working prototype, although for complex or large projects a convincing simulation of the functioning is considered a successful design.
9. Documentation and communication:
    a. Maintains a design activity log book.
    b. Develops, updates, and follows a Gantt Chart.
    c. Midterm oral project review, evaluated by students and faculty
    d. Final presentation, evaluated by student peers and facylty.
    e. Final report
    f. Design poster.

Procedure

One of the authors who usually teaches Fluid Mechanics[4] labs and always asked for student input at the conclusion of the course. Students’ feedback indicated the interest of a lab that dealt with Drag Coefficient that is applicable in the real world. For some time students conducted the a drag coefficient lab using simple geometric shapes, such as a sphere, a hemisphere, a flat circular aluminum plate, and a tear drop shaped object that was generated by the rapid prototyping machine. These simple shapes gave students an understanding of how wind tunnel testing is conducted but are not as interesting and relevant as testing real world object such as model cars and possibly airplane models. The instructor, one of the coauthors, assigned the task of developing the drag lab to one of the capstone design groups. The task was divided into the following steps:

- Select three car models with maximum varying streamlines shapes.
- Acquire the selected model toy cars, if possible.
- Measure or estimate the dimensions and develop CAD solid models using SolidWorks[5].
- Simulate flow over these models using the Computational Fluid Dynamics software.
- Determine average drag coefficient for each model.
- Generate scale models of these CAD models using the departmental rapid prototyper[6].
- Test these scale models in the wind tunnel at various speeds as used in CFD simulation.
- Compare the results and the relative Drag coefficients.

Three cars of differing eras and models were chosen. These were 1920's Ford Model T, 1970's Porsche 917, and 1979 Chevrolet El Camino pickup truck. These cars shown in figure 1.

1970's Porsche 917 1979 Chevrolet El Camino 1920'S Ford Model T

Figure 1: Model cars chosen for drag coefficient lab development.

The solid models were developed for each of these models which are shown in Figure 2. These CAD models were the bases of both computational fluid dynamics simulation as well as generating solid prototypes on the prototyping machine, thanks to the integrated software package for easy data transfer between models.

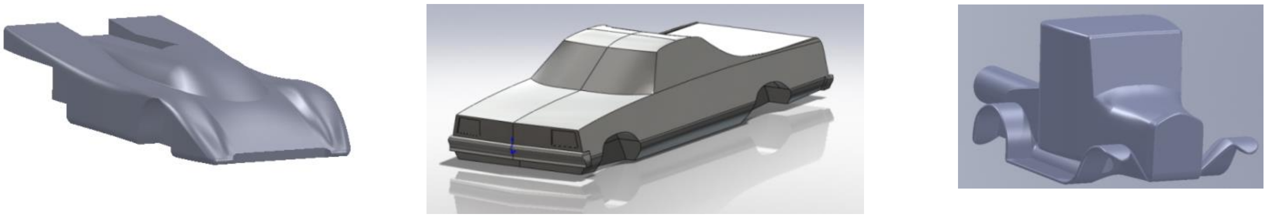
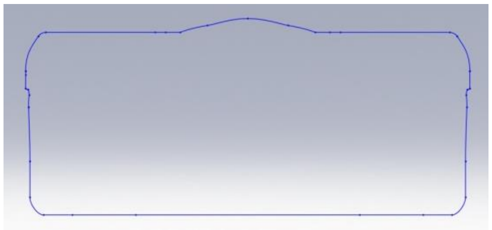


Figure 2: CAD models of (from left) Porsche, El Camino, and Model T.

Computational Fluid Dynamics (CFD), tool is integrated within the SolidWorks software package and allows the simulation of flow over solids just as in a wind tunnel. All the input parameters such as velocity, temperature, area, etc. to solve an equation for drag coefficient are inputs to the simulation. The characteristic area used, in our case the frontal area, came from the frontal area of each model, easily derived from the cad model. The frontal area is a silhouette of the car looking at it from the front as shown in figure3, in this case for the Porsche model.



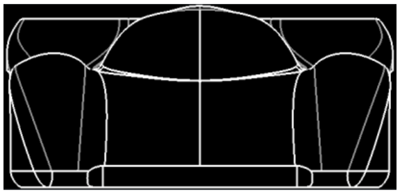


Figure 3: Frontal area determination from CAD models

Then the CFD simulation was performed for each CAD model and respective drag coefficient was calculated. Next step was the wind tunnel testing. For this stage scale models of the cars were needed, for which the 3D printer (rapid prototype) was used. The 3D printer works like a computer controlled (CNC) hot glue gun that builds an object in successive layers from bottom up. It uses two different materials; one for building the model and the other for supporting overhanging sections. The support materials are easily removable leaving only the model. Depending on the geometry of the object, build orientation, and resolution of the 3D printer, the model surfaces are not smooth like a finished product. However a little hand polishing with common abrasive cloth was enough to achieve sufficient surface smoothness that would not cause high surface friction. The hand finished plastic scale models are shown in figure 4.

Figure 4: 3D printed scale models. The wheels are excluded for simplicity.

Testing and results

The plastic solid models from the 3d printer were then mounted using aluminum sleeves inserted from the back and horizontal holders inside the wind tunnel. Tests were then conducted on each car using an AEROLAB wind tunnel. Since drag coefficient depends on the speed, all simulation and wind tunnels tests were done for a speed of 30 mph. Averages of several tests are shown in table 1. Drag coefficient data from CFD simulation is also tabulated and the deviations are calculated as percentage of CFD simulation values. While the C_d value for Porsche in wind tunnel testing was slightly higher than that from CFD simulation, it was somewhat lower for the other two models. In all cases the deviation was less than 10%.

These results were not only close to each other in both methods, but variations were a combination of positive and negative. This may indicate that the results obtained are close to ideal ones.

Of course more testing is needed while making sure each test is performed in identical conditions. Especially wind tunnel test parameters and mounting must be closely monitored and controlled for consistency. Furthermore scale model building and surface preparation experience were invaluable experience as part of the experimental process.

Table 1: Comparison of drag coefficients from CFD simulation and wind tunnel testing.

Car	**CFD simulation**	**Wind Tunnel**	**Percent deviation (%)**
Porsche	0.43	0.45	4.65
El Camino	0.21	0.19	-9.52
Model T	0.69	0.63	-8.70

Conclusion

The goal of developing a laboratory exercise for teaching drag coefficient was successful. A capstone design group was employed to develop the CAD, CFD and scale models and perform the tests under the supervision of the authors. It fulfilled the wish of the design group to study the car models for drag behavior. It is expected that when a laboratory exercise will be fully developed and conducted this fall semester, it will be one of the attractive labs students will do. The 'Drag Lab' would include theoretical background study as well as computational (CFD) simulation and physical model testing in a wind tunnel. The enthusiasm of the design group members was remarkable knowing their efforts and study will benefit future students like themselves. This lab exercise will also remind the future students of the level of study and efforts that went into its development by peer students like themselves. The addition of this lab in fluid mechanics lab syllabus will serve the objective of bringing real world problems into engineering and technology classrooms.

Bibliography

1. Chadwick, S. G. and Hsske, S. J. The drag coefficient of tennis balls. In: SUBIC, A. and HAAKE, S., (eds.) *Engineering of sport - research development and innovation.* Blackwell, pp-169-176, 2000.
2. C. Norberg, “Flow around a Circular Cylinder:Aspects of Fluctuating Lift,” Jurnal of Fluids and Structures, V. 15, 3-4, pp 459-469, April 2001.
3. Bernard J. Hamrock, Steven R. Schmid, Jacobson, “Fundamentals of machine Elements”, 2nd Ed. McGraw-Hill Higher Education, 2004. 980 pp.
4. M. Moran & H. Shapiro, “Fundamentals of Engineering Thermodynamics”; 7th Edition;.Dec. 2010, Wiley.
5. Manual and help files for the CFD simulation within SolidWorks software package.
6. Manual of the Dimension 3D printers by Stratasys.