

Establishing Temperature Dependency of the Aerodynamic Drag using CFD and Experimental Analysis

Ajinkya Krishnakumar, Parth Sharma*, Yash Tambi and B. Ashok

School of Mechanical Engineering (SMEC), VIT University, Vellore - 632014, Tamil Nadu, India;
ajinkya.krishnakumar@outlook.com, parthsharma231@gmail.com, yash.tambi@gmail.com, ashok.b@vit.ac.in

Abstract

Objective: The study focuses on finding out, whether the flow temperature plays a role in the aerodynamic drag experienced by the moving body or not. **Method/Analysis:** For doing so, a small scale wind tunnel was constructed and its initial recorded data was matched with the simulated results. This gave an extent to which the fabricated tunnel is accurate or correct in determining drag. Next the temperature of flow in tunnel would be changed physically, to get drag for an extended range of temperature difference. The results will be observed with the help of plotted graphs, and conclusions would be drawn. **Findings:** The study revealed the temperature dependency of drag. It tends to increase with temperature. With an increase in 3°C in the temperature the drag coefficient was found to increase by 0.025. **Novelty:** The tunnel design can be further improved to provide accurate dependence relation.

Keywords: Acquisition System, Aerodynamic Drag, Digital Data, Drag Coefficient, Drag Dependence on temperature, Flow Temperature, Wind tunnel

1. Introduction

Automobiles have been a crucial part of human lifestyle since a prolonged period of time. Their range of affect can vary from the daily earning of a small paper boy, to a country's much bigger economy. Since a long time thus, an automobile's design and geometry has been a foremost concern among engineers for achieving an efficient and effective work flow. Aerodynamic study is a vital way for making such designs worthy. Forces such as drag and lift are an integral part of this study and hence their analysis is essential. A prominent way for measuring such forces is by using a mechanical device, wind tunnel. The effectiveness of wind tunnel was first demonstrated by the wright brothers, who developed their early wing design, in a small low-cost wind tunnel. Since then this device has evolved, and now find its application in non- traditional applications also, such as simulating flow used to cool engine as well as electronic systems.

Wind tunnel basically is a tubular structure that has manmade wind, blown through it. It is designed to generate air flows at different velocities for analysis procedure, through a test section. Wind tunnels work on the idea, that a stationary model with air moving around it depicts a similar system, as of a real full-scale airplane moving through stationary air does. Wind tunnels are typically used in aerodynamic research to analyze the behavior of flows under varying conditions, both within channels and over solid surfaces. Huge wind tunnels serve as a vital platform of research for automobile industries and their product bodies. However due to several constraints this tunnel analysis is often carried out on a reduced scale, and then the results are interpolated to get real time data. To achieve the same Reynolds number as for the real application, the kinematic viscosity or flow velocity has to be changed. In most wind tunnels air is used at atmospheric pressure and hence only option left is to increase the wind velocity.

*Author for correspondence

According to¹, the wind tunnels were designed to provide a well-defined, controllable, uniform flow of air for experimental and design validation purposes. He also provided examples to illustrate the application of low speed wind tunnel for the investigation of vehicle aerodynamics. In² presented an overview of results from the wind tunnel test of a 1/4-scale V-22. The small-scale prop rotor was tested on the isolated rotor configuration of the Tilt Rotor Aero acoustic Model (TRAM). Representative examples of air loads, acoustics, structural loads, and performance data were provided and discussed.

For the study of wind tunnels, the work of³ was taken into account. It focused on study of various wind tunnel parameters, types of tunnels, wind tunnel size and shape, air delivery, inlet-outlet dimensions and length of wind tunnel. In⁴ covered the process of design and fabrication of the small wind tunnel. In completing this project, a Computer Aided Drawing (CAD) called Solid Works was used for designing. The drag force on a sphere in an air stream was measured at various free stream velocities below 100 m/s. The raw data were processed according to classical equations of fluid mechanics which define the drag coefficient. An impression of fluid field flow around a sphere was also captured using white smoke. Method of analysis the flow in test section was shown by using strings. In yet another study Michael⁵ described the wind tunnel testing methodology that has been applied to testing over 200 airfoils at low Reynolds numbers (40,000 to 500,000). Along with this, the effects of atmospheric stability on flow in urban street canyons were studied using a stratified wind tunnel along with the designing and evaluation of a low-speed wind tunnel with expanding corners⁷. Furthermore, for the CFD approach to a wind tunnel setup, a study was referred in⁸. To study the pressures and their distribution around the test object, the similarity analysis along wake region of elliptic cones, by was taken as an analogy⁹.

2. Objective and Methodology

In the following study, the flow temperature effect on the aerodynamic drag is found out. A small experimental setup is used to provide a practical approach to the study. For the legitimacy of this setup, its initial readings are compared with that of software and validated. Later the flow temperature is varied physically, and the pressure readings along the test piece axis are recorded. Furthermore, aerodynamic drag coefficient for all the

readings is calculated and plotted against the varying temperature to observe the dependence.

The software analysis was also done of the test model simultaneously, to get a different set of flow variables. The software chosen for this was Ansys14TM. From the obtained values further calculations were conducted in the software itself. The details of the whole procedure can be found out in Section 3. The experimental analysis of air resistance on aero foil is done, by constructing a reduced scale wind tunnel from a simple 200 mm diameter PVC pipe forming the test section. The electrical harness used includes pressure and temperature sensors, along with a common LCD display. In the analysis, these sensors are used to get the pressure difference across the prototype, as well as temperature associated with the flow. The difference so measured, is then used to calculate the force experienced by the cross-section of the test model, which is later utilized in the calculation of drag coefficient. The nozzle and collector were not incorporated as the aim of this study was to only to establish temperature dependency of the aerodynamic drag. A detailed description has been discussed in the Section 4.

The two sets of results, obtained through software and experimentally, are compared to find out the accuracy of the wind tunnel. This validation of the tunnel has been carried out in the Section 5. Once an accuracy factor is obtained, the further experiment is conducted. The flow temperature is varied using a hair drier and the consequent pressure changes were recorded. Following this, the calculation of drag coefficients for all the pressure differences recorded was done and further multiplied with the accuracy factor, to get real time results of the aerofoil. The step by step depiction of the whole process adopted, has been provided in the form of a flow chart in the Figure 1.

3. CFD (Software) Analysis

The simulation of the mean flow characteristics was done using the K epsilon turbulent model. It is a two transport equation (PDEs) model. Equations of k (Equation (1)) and ϵ (Equation (2)), together with the eddy-viscosity stress strain relationship constitute the k - ϵ turbulence model, where ' ϵ ' represents the dissipation rate of for turbulent kinetic energy ' k '. This model leads to stable calculations that converge relatively easily, by making reasonable predictions for many flows.

For turbulent kinetic energy k :

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + P_k + P_b - \rho \epsilon - Y_M + S_k \quad (1)$$

For dissipation rate ϵ :

$$\frac{\partial(\rho \epsilon)}{\partial t} + \frac{\partial(\rho \epsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] + C_{1\epsilon} \frac{\epsilon}{k} (P_k + C_{3\epsilon} P_b) - C_{2\epsilon} \rho \frac{\epsilon^2}{k} + S_\epsilon \quad (2)$$

The software analysis of the test model was done using ANSYS™ 14.0 and Solid Works™ 2014. The inputs provided to the system constitute of a simulating flow of speed 43.26 m/s (closest to the experimental blower speed) with both laminar and turbulent nature. Also the fluid density was taken as 1.225 kg/m³. The test model was constructed using Solid Works™ 2014 as shown in Figure 2 and was

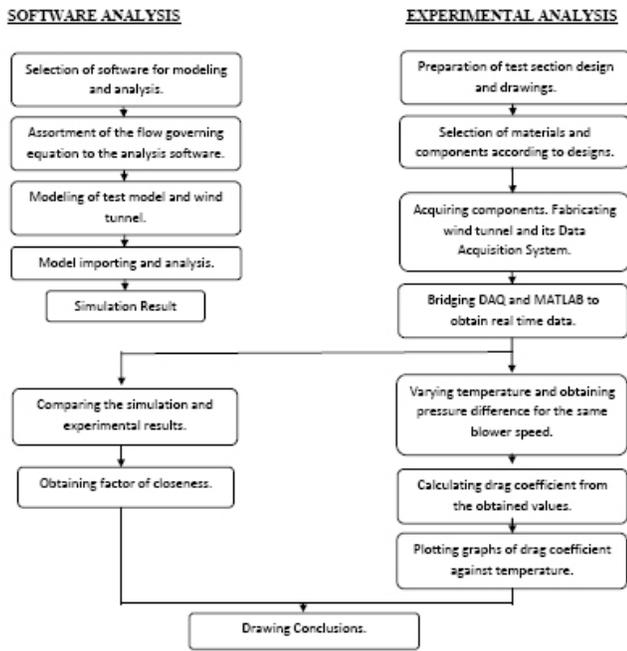


Figure 1. Methodology of the entire process.

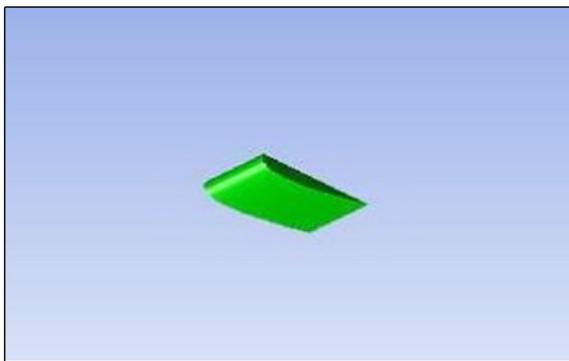


Figure 2. Test Model used as a surface for drag.

estimated to a real design model. It was later covered with a cylindrical wind tunnel section and meshed, as shown in Figure 3 and Figure 4.

After the successful completion of simulation, the obtained result indicated aerodynamic drag coefficients over 200 iterations which are depicted in Figure 5. The

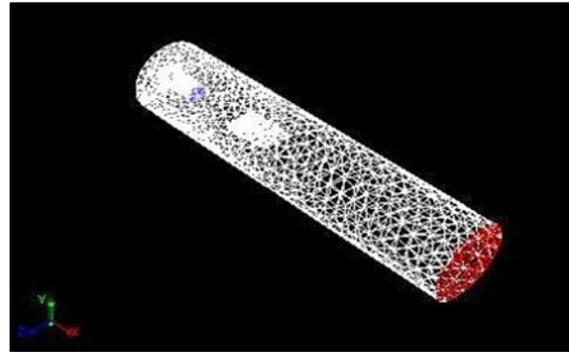


Figure 3. Simulation test model inside flow meshing.

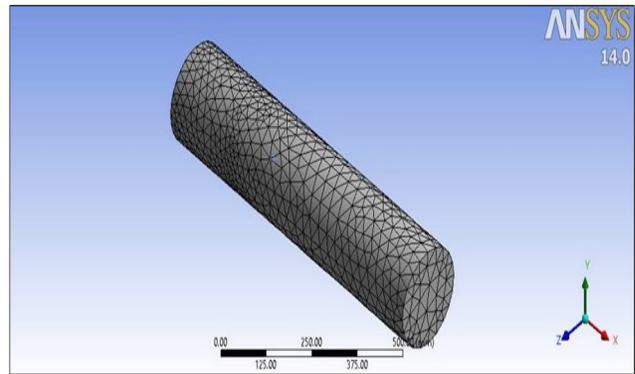


Figure 4. Meshed wind tunnel (depicting air flow).

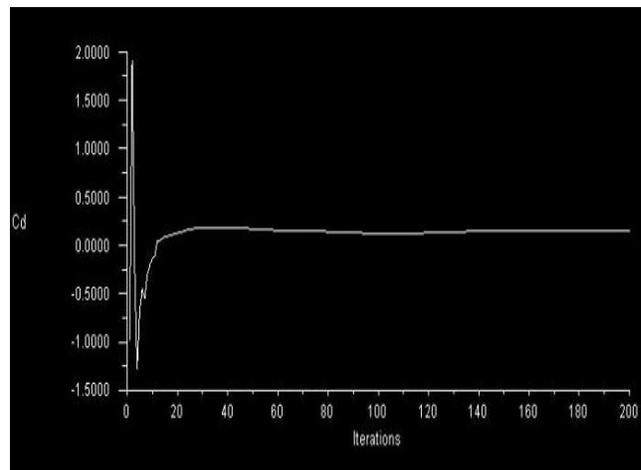


Figure 5. Simulated drag coefficient values.

average value obtained from these iterations was found to be $2.6703e-01$. This value was later compared with experimental value to get a contrast between the two evaluation methods.

4. Experimental Analysis

The experimental analysis is done by collecting data through a reduced wind tunnel section. The test section is made out of a 1 m long PVC pipe. The collector fan is fitted to the one end of the tunnel and seal packed using hard board and m seal, as it can be viewed in Figure 6. While on the other side centrifugal blower is placed carefully to generate an axial flow. The blower used was a BOSCH (18000 RPM, $2.8 \text{ m}^3/\text{min}$). M8 bolts are used for attaching the fan holder to the pipe, rigidly.

The electric mesh used for collecting data includes a pressure sensor namely Bosch BMP 180, which is a piezoresistive barometric pressure sensor with a least count of 1 pa. The maximum pressure that can be measured is 2 bar (2 atm). It is also capable of measuring temperature with a least count of 0.1°C and hence provides temperature compensated values of pressure. The output is in digital form (I2C). The LCD used is a common 20×4 LCD available off the shelf. The data acquisition system is mounted onto the test section using m-seal and two-way tape strips as shown in Figure 7. The test model is mounted using two sheet metal strips, bending them into 2 C-sections and attaching them in inverted fashion to the pipe as well as the model. It is placed exactly at the center of the tunnel cross section and is kept under consideration that a minimum gap is maintained from the walls. Also the model



Figure 6. Inside view of test section.

placement is such that the linear distance of it from the two fans is more than ten times the length of the model itself. This helps to reduce turbulence and maintain a boundary layer over the surface. The model is firmly situated using bolts and nuts. The sketch of the setup is shown in the Figure 8.

To perform the experimental analysis, the collector fan is switched on followed by the blower fan. One pressure sensor is placed in front of prototype in free flow, while other is attached to the tail section of aerofoil to get different values of pressure. The temperature sensor is placed in the free flow and used simultaneously to gather data. The values of temperature, pressure are recorded and displayed on the LCD, while the same are also given out through UART. A USB-UART bridge is used to connect the entire electronics setup to the PC via USB. Data is collected in



Figure 7. Test section with electric. harness

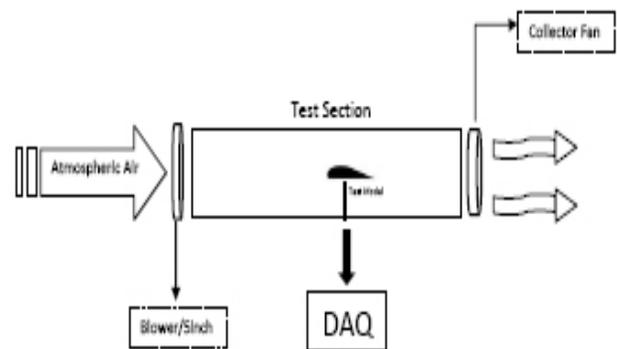


Figure 8. Sketch of the experimental setup

the PC on MATLAB, where a GUI has been developed to show the recorded data. The software MATLAB is used for data collection and packing. First the data is collected then parsed and stored into the respective variables of temperature and pressure. This is followed by storing of data in an array and graph plotting.

The data recorded can be easily viewed using Microsoft Excel. It was then used to calculate the drag coefficient for each and every pressure reading that was recorded. Later these readings were taken average of to get the average drag coefficient from the experimental analysis and compare its closeness to the software data obtained.

4.1 Calculation Process

Once the data is procured it is recorded in tabular form. The drag coefficient is calculated for each and every pressure value that is recorded from the test section. It is done by using the Drag Equation (Equation (3)), where C is the coefficient of drag, ρ is the density, u is the flow stream velocity and A is the cross sectional area of the profile. The flow velocity was calculated using the blower flow rate and cross-sectional area.

$$Fd = \frac{1}{2} \rho u^2 CA$$

This drag force, F_D is also calculated using the conventional relation of pressure difference along with area ($F = \Delta P * A$), and compared to give the value of drag coefficient. This can be viewed under Table 1.

The pressure at the front of the aerofoil was found to achieve a constant value around 985.7 hPa (P_2). This pressure can be utilized to find out the pressure differential for every reading ($\Delta P = P_2 - P_1$). Furthermore, the calculations were done considering two constants i.e., ρ (density) and V (volume flow rate). The most common value of density, 1.225 kg/m^3 , and the volume flow rate of $2.8 \text{ m}^3/\text{min}$ (rated blower rate) were used. For the particular reading considered, the value of coefficient was found to be 0.6159. All the rest readings were used to calculate a series of drag coefficients. At the end this range was averaged to get an approximate drag coefficient at the ambient temperature. This value was found to be $C_{D(\text{atm})} = 0.578$.

After obtaining the drag coefficient at room conditions, the temperature was varied using a hair blower. The temperature kept on increasing with a difference of 0.1°C between two consequent readings, giving different values of the pressure differential. All the readings were extracted by the MATLAB and exported according to their corre-

sponding prevailing temperatures. For each temperature value, the corresponding coefficient value was calculated and tabulated in Table 2. The results were then used to plot a graph between the flow temperature and the drag coefficient experienced by the test model (Figure 9). A linear trend-line was also utilized to show the relationship among the two variables.

5. Validation

The validation process aims at authenticating the experimental method, carried out for the analysis of test model in the small scale wind tunnel, by bench marking

Table 1. Pressure data obtained from MATLAB and its consequent drag coefficient values (at ambient temperature)

Read	Humidity	Temperature (deg C)	Pressure	C_d
1	44	31.9	9.8566	0.619412
2	44	31.9	9.857	0.615922
3	44	31.9	9.8565	0.620284
4	44	31.9	9.8569	0.616794
5	44	31.9	9.8569	0.616794
6	44	31.9	9.8566	0.619412
7	44	31.9	9.8567	0.618539
8	44	31.9	9.8568	0.617667
9	44	31.9	9.8566	0.619412
10	44	31.9	9.8573	0.613305
11	44	31.9	9.8568	0.617667
12	44	31.9	9.8571	0.615049
13	44	31.9	9.8564	0.621156
14	44	31.9	9.858	0.607198
15	44	31.9	9.8645	0.550491

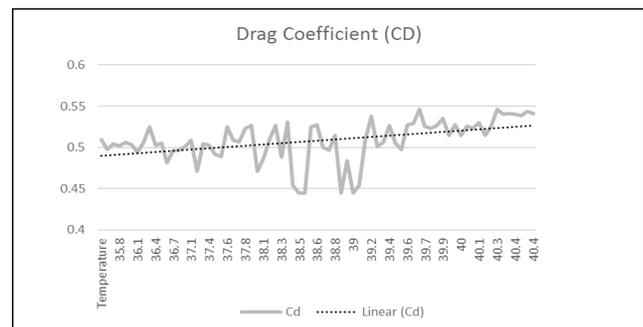


Figure 9. Variation of drag coefficient with temperature.

Table 2. Pressure data obtained from MATLAB and its consequent drag coefficient values (constantly increasing temperature)

No.	Temperature (deg C)	Pressure	C_D
1	35.5	9.8616	0.509488
2	35.7	9.863	0.497274
3	35.8	9.8622	0.504253
4	35.9	9.8625	0.501636
5	36	9.862	0.505998
6	36.1	9.8623	0.503381
7	36.2	9.8634	0.493785
8	36.3	9.862	0.505998
9	36.4	9.8598	0.525191
10	36.5	9.8624	0.502509
11	36.6	9.8621	0.505126
12	36.7	9.8648	0.481571
13	36.8	9.8632	0.495529
14	36.9	9.8631	0.496402
15	37.1	9.8626	0.500764

the results of software simulation. According to the two methods the drag coefficient at ambient conditions was found to be: $C_{D(\text{exp})} = 0.578$ and $C_{D(\text{soft})} = 0.267$. From these two values it can be inferred that the prepared wind tunnel has a closeness factor of 0.4602. This much closeness has been considered sufficient enough to reflect that the results obtained by this research are worthy.

6. Conclusion

From the graph plotted (Figure 9), a positive slope of the plot can be easily inferred. Thus there is an increase in the surrounding (atmospheric) temperature, the flow of air around an automobile also changes and hence affects the drag experienced by the vehicle. The drag coefficient was found to increase with the rise of temperature, according

to the experimental analysis. The reliability of the experiment was also proved, by comparing its result with the software simulation. And it was found enough, to consider its result worthy. This showed that the temperature dependency of drag coefficient found in the later experiment was legit. Hence it should be considered as an important governing variable in the aerodynamic studies. However due to geometry constraints and the assumptions involved in the experiment, a rigid and valid relationship was not established between the flow temperature and drag force experienced by the vehicle under consideration.

7. References

1. John L. Design and application of low speed wind tunnels; 2002.
2. Young O, Larry A. Overview of the testing of a small-scale propotor. National aeronautics and space administration Moffett field CA AMES research center. 1999.
3. Kakate VL. Study of measurement and control aspects of wind tunnel. Int J Innov Res Electr Electron Instrum Control Eng. 2014; 2(3):1–4.
4. Yang KNC. Design of wind tunnel (fluid flow analysis). Diss Universiti Malaysia Pahang; 2012. p. 1–24.
5. Michael SS, Deters RW, Williamson GA. Wind tunnel testing airfoils at low Reynolds numbers. AIAA Paper. 2011; 875:1–32.
6. Kiyoshi U. Wind tunnel experiments on how thermal stratification affects flow in and above urban street canyons. Atmospheric Environment. 2000; 34(10):1553–62.
7. Lindgren B, Johansson AV. Design and evaluation of a low-speed wind-tunnel with expanding corners. 2002.
8. Fischer O. CFD approach to evaluate wind-tunnel and model setup effects on aerodynamic drag and lift for detailed vehicles. No: 2010-01-0760. SAE Technical Paper; 2010.
9. Sundararaj M, Elangovan S, Sridhar BTN. Similarity analysis of pressure distribution along the wake axis of elliptic cones. Indian Journal of Science and Technology. 2014; 5(7):1–5.