A Technical Review – Comparative Assessment of Aerodynamic Forces Generated Due to Wind Deflector of Passenger Car by CFD and Experimental Method”

Mayank K Parmar
PG Student
Automobile Department
L D College of Engineering, Ahmedabad

Prof. Dhruv U.Panchal
Assistant Professor
Automobile Department
L D College of Engineering, Ahmedabad

Prof. Twinkle D Panchal
Assistant Professor
Automobile Department
Indus University, Ahmedabad

***Abstract* - Reduction of drag is becoming a very important challenge for all the automobile manufacturers as they are competing intensely to produce passenger cars with better gas mileage. Lower drag provides better performance such as higher top speed and better stability. The body shapes of passenger cars are primarily designed to meet the top speed, economical, and aesthetic requirements. The aerodynamic characteristics directly affect the stability', driving characteristics, safety, operation, and oil consumption of automobiles. This experiment’s objective is to analyse aerodynamic characteristics of passenger car using wind tunnel method and Computational Fluid Dynamics(CFD) approach.**

Keywords — Computational Fluid Dynamics(CFD), Wind tunnel

I. INTRODUCTION

Automotive aerodynamics is the study of the aerodynamics of road vehicles. Its main goals are reducing drag and wind noise, minimizing noise emission, and preventing undesired lift forces and other causes of aerodynamic instability at high speeds. For some classes of racing vehicles, it may also be important to produce down force to improve traction and thus cornering abilities.

The importance of the good vehicle aerodynamics parameters in design of vehicle is being increasingly recognised. Constant striving for improved economy dictates the importance of the study of the vehicle drag and importance of the vehicle handling emphasize the need for a through appreciation of aerodynamics.

It should be stressed that above the speed of 70km/Hr.(43.5mph) aerodynamic drag exceeds 50 percent of the total resistance to motion and above 100km/hr.(62mph) it is most important factor.

An automobile is a small object submerged amid vast surrounding of air. The motion of the vehicle takes place through a large mass of either stationary air, or air in the motion. The air exerts force on the automobile. It is the superstructure of the body of the vehicle which is mainly exposed to the air. An arbitory shaped body will experience a larger air resistance which impose that there is more loss of engine power. Consequently less power will available to propel the automobile thereby caushing less load carrying capacity and slow speed for the same fuel consumption.

The force exerted by air on a moving auto vehicle had two components.

- In the direction of motion
- In the perpendicular to motion

The force in the direction of the motion is called drag and that in the perpendicular direction is known as lift.

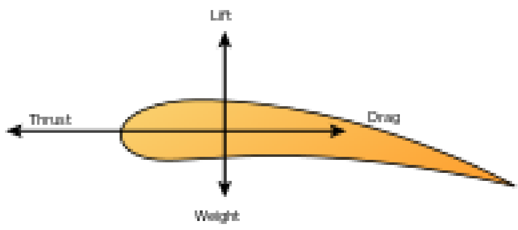

Fig 1.1 aerodynamic forces

Computational fluid dynamics (CFD), using computer simulation, analyses systems of fluid flows, heat transfer, and associated phenomena such as chemical reactions. Examples of areas CFD can be applied to are; design of internal combustion engines, aerodynamics of aircrafts and vehicles, meteorology (weather prediction), and external environment of buildings (wind loads and ventilation). CFD has many advantages over experiment-based approaches, such as reduction of lead times and costs of new designs, study systems under hazardous conditions, systems that are impossible to study with controlled experiments and, the unlimited level of detail of results. There are also problems with CFD. The physics are complex and the result from CFD is only as good as the operator and the physics embedded. With today's computer power, there is a limitation of grid fineness and the choice of solving approach (DNS, LES and turbulence model). This can result in errors, such as numerical diffusion, false diffusion and wrongly predicted flow separations. The operator must then decide if the result is significant. While presently, CFD is no substitute for experimentation, it is a very helpful and powerful tool for problem solving.[1]

When working with CFD a number of different steps are followed. These steps are illustrated in figure.

Figure 1.2 The CFD process

To be able to confirm the results from the CFD simulations the aerodynamic forces have to be measured in the real world. The easiest way today is to use a wind tunnel. A wind tunnel, simplified, is a big fan that blows air onto a test subject, which is located in a test section. The test subject is connected to a balance that measures forces. There are a lot of things that can be tested in a wind tunnel; aerodynamic forces and moments, yaw conditions, engine cooling performance, local flow field measurements, climate effect and aeroacoustic. There are many ways to design a wind tunnel, but there are two basic types of wind tunnels and "two" basic test sections. From these basic configurations there are an enormous number of different configurations.

Types of wind tunnel

- Open circuit wind tunnel
- Closed circuit wind tunnel

II. REVIEW OF PAPERS

1) Ravindra A. Viveki, Dr.N.K.Chougule, This paper describes how the drag force affect to the performance of the car. . The objective of this study is to quantify effect of roof curvature on the drag and understand the mechanisms of drag production. Studying flow over passenger car is costly in wind tunnel due to cost for the setup as well as the number of runs required for successful drag reduction and optimization. With the use of CFD these costs are avoided and multiple runs can be set up at the same time for comparison and optimization. It is motivated for this study by using a CFD approach to analyse the flow over a MIRA reference car. In order to reduce the aerodynamic drag of cars, modification in roof is done on the baseline model and three parameters like velocity, thickness to chord ratio and position of thickness to chord ratio are used for the optimization purpose to reduce drag using Taguchi method and air flow will be simulated.

After simulation it was observed that as roof curvature increases from the mid position of roof, wake zone reduces hence vehicle drag coefficient reduces by 2.58%. Roof curvature will lead to more low pressure zone above roof and this will lead to more lift and was found to be less than 1%. More lift will lead to less vehicle stability at higher speed. So we have optimized between drag reduction and lift increment. More the smooth curvature, more the drag reduction.[2]

2) Emmanuel Guilmineau, This paper describes the flow over the Ahmed body by experimental method and by CFD. . Wake flow is two-dimensional for low incidences of the rear slant, then becomes three-dimensional when the angle of the hatchback approaches 30^0 and reverts to two-dimensional behaviour for angles higher than 30^0 where above this angle, a sudden drop in drag occurred. In this paper, he investigate numerically the flow around the Ahmed body for the base slant angles 25^0 and 35^0 . Results are compared with experimental data. The two-dimensional behaviour of the flow, for the slant angle 35^0 , is well predicted, whereas the transition of the wake to a fully three-dimensional, for the slant angle 25^0 , is not reproduced.

For the larger slant angle, computations with or without the stilts on which the body was supported in the wind tunnel give similar results. However, the drag is increased due to the presence of the stilts. All turbulence models predict the topology of the flow correctly. On the other hand, the EASM model gives a better estimate of the drag. By taking of account the stilts, this model gives a value to less than 3% of the experimental value. At the 251 slant angle, all simulations predict massive separation, whereas the experiment shows reattachment about half-way down the centre of the face.[3]

3) Y. Wang, Y. Xin, Zh. Gu, Sh. Wang, Y. Deng and X. Yang This study is dedicated to the investigation of the aerodynamic characteristics of three typical rear shapes: fastback, notchback and squareback. . The object was to investigate the sensibility of aerodynamic characteristic to the rear shape, and provide more comprehensive experimental data as a reference to validate the numerical simulation. In the wind tunnel experiments, the aerodynamic six components of the three models with the yaw angles range from -15^0 and 15^0 were measured. The realizable k-ε model was employed to compute the aerodynamic drag, lift and surface pressure distribution at a zero yaw angle. In order to improve the calculation efficiency and accuracy, a hybrid Tetrahedron-Hexahedron-Pentahedral-Prism mesh strategy was used to discretize the computational domain.

The experimental and numerical results revealed that the realizable k-ε model with hybrid tetrahedron-Hexahedron-Pentahedral prism mesh strategy to discretize the computational domain was proven to be efficient to in simulating the mean flow field around the vehicles. The aerodynamic characteristic is closely related with the rear shape, and different rear shape will induce large different aerodynamic characteristic. The rear negative pressure zone was the main source of aerodynamic drag. Therefore, the drag coefficient of fastback with a more streamlined rear shape was the smallest because of its smaller negative pressure zone. The results also revealed that the more non-streamlined the model was, the stronger the turbulence kinetic in its wake.[4]

4) K.Selvakumar,Dr.K.M.Parammasivam .The aerodynamic drag force of a sedan vehicles is due to separation of flow near the vehicle's rear end is one of the major causes. To delay flow separation, bump-shaped vortex generators are tested on the roof end. Commonly used on aircraft to prevent flow separation, vortex generators themselves create drag, but they also reduce drag by preventing flow separation at downstream. The overall effect of vortex generators can be calculated by sum of the positive and negative effects. Since this effect depends on the shape and size of vortex generators, those on the vehicle roof are optimized .This study presents the optimisation of various aerodynamic characteristics in a Sedan car using computational tools such as CFX and also by using experimental testing using scaled models in the wind tunnel.

The drag force increases with increases with air velocity for both the base and the VG model There is a comparative reduction in the drag coefficient value of the VG model to the base model With implementation of the VG's we obtain negative values of lift coefficient Negative lift coefficient creates negative lift or down force which pushes the car to the Ground. From computational simulation (CFD) and wind tunnel experiment, car's drag coefficient is 0.42 to 0.48, respectively; the difference between the two is around 12.5 percent[5]

4) Bhagirathsinh H Zala, Harilal S Sorathia, Dinesh L. Suthar, Vashant K. Pipalia, This paper is Research about the aerodynamic of sedan car model by comparing experimentally and subsequent validation by computational fluid dynamics (CFD) The experimental investigations was performed on an open circuit suction type wind tunnel having a 30 cm x 30 cm x 100 cm test section, and maximum speed of 33 m/s on a geometrically similar, reduced scale (1:20) Aluminium car models .while the three dimensional computational analysis was carried out using with the help of software tools like ANSYS-CFX to simulate the flow of air around the automobiles ANSYS-CFX, CFD code were used to run the simulation. The main objective of the study is to predict the drag coefficient experimentally as well as computationally. Several factors that influence the drag coefficient such as flow separation, vortex and the effect of pressure coefficient have been studied. Detail velocity profile and pressure distribution plots around the car envelopes have been presented

Drag force of sedan car obtained computationally has higher value than obtained experimentally by 11%. Drag coefficient decreased initially then with further increase in Reynolds number CD attended almost constant value. Distribution of Pressure obtained experimentally is in agreement with prediction that distribution of The pressure remains low down over nose, the back light and on to the trunk because of the continuing curvature or the streamline shape. The front radiator zone displayed higher pressure contours. The pressure contours obtained by CFD at different velocity are identical with those obtained experimentally and thus validate the experimental procedure and result.[6]

5) R. B. Sharma1, Ram Bansal, This work proposes an effective numerical model based on the Computational Fluid Dynamics (CFD) approach to obtain the flow structure around a passenger car with Tail Plates. The experimental work of the test vehicle and grid system is constructed by ANSYS-14.0. FLUENT which is the CFD solver & employed in the present work. In this study, numerical iterations are completed, then after aerodynamic data and detailed complicated flow structure are visualized. In the present work, model of generic passenger car has been developed in solid works-10 and generated the wind tunnel and applied the boundary conditions in ANSYS workbench 14.0 platform then after testing and simulation has been performed for the evaluation of drag coefficient for passenger car. In another case, the aerodynamics of the most suitable design of tail plate is introduced and analysed for the evaluation of drag coefficient for passenger car.

In this analysis, the coefficient of drag has been reduced 3.87% and coefficient of lift is reduced 16.62%. Hence, the Tail Plates is the effective tool to reduce the drag force on vehicle. The drag force can be reduced by using add on devices on vehicle and fuel economy, stability of a passenger car can be improved.[7]

6) Chenguang Lai, Yasuaki Kohama Shigeru Obayashi Shinkyu Jeong, This paper regards experimental and numerical studies to evaluate and analyse the influence of the notchback rear diffuser angle on aerodynamic drag and wake structure. The relationship between drag and rear diffuser angle is summarised, and the flow mechanism are analysed and discussed. A speculation regarding lower trailing vortices is proposed, and is verified in the present research model. Rear diffuser angle is important factor influencing the wake structure, and optimising the rear diffuser angle can contribute to reduce the drag force and improve the wake structure.

The aerodynamic drag of the vehicle can be influenced by the rear diffuser angle. With increasing the rear diffuser angle, the drag first decrease and then increase. By increasing the rear diffuser angle, the mass of the under body flow increase, which can accelerate the velocity recovery in the wake region, and contribute to decrease the drag force. Rear diffuser angle is important factor in developing the rear structure. At large rear diffuser angle, due to the flow separation, the high spanwise pressure gradient on the rear diffuser surface may generate a pair of lower trailing vortices in the wake downstream of the underflow. Therefore optimising the rear diffuser angle can contribute to reduce the drag force and improve the wake structure.[8]

7) Zhu Hui, YU Hao, Yang Zhigang , In this work, the connection between the result of numerical simulation of aerodynamic drag coefficient of hatch back car and the angle change of front and back windshield and hood was analysed. In order to find out how the angle change affect the drag coefficient respectively and comprehensively. By

using CFD method to simulate the drag coefficient of the change of back and front windshield angle and hood angle of MIRA body respectively. And then 3 angle of back and front windshield angle and hood angle were chosen. Totally 27 kinds of car were simulated to find out combined influence and to confirm orthogonal test can be used in aerodynamic drag optimisation or not.

A maximum coefficient of drag was achieved at back windshield angle 35^0 by using MIRA fast back model. And the minimum coefficient of drag was achieved at back windshield angle 48^0 when the angle is bigger than 35^0. When the front windshield angle is below 45 , there is little change in coefficient of drag. When front windshield angle is greater than 45 coefficient of drag increase with increase of front windshield.[9]

8) Sneh Hetawal, Mandar Gophane, Ajay B.K., Yagnavalkya Mukkamala, This paper describes the design and CFD analysis of a Formula SAE car. A numerical study of a rear engine SAE race car is presented. The focus of the study is to investigate the aerodynamics characteristics of a SAE race car with front spoiler, without front spoiler and with firewall vents. To increase the aerodynamic performance of race car, an attempt is made to modify the design of a Formula SAE car. Comparative study is done on three car models by carrying out CFD simulations. The aerodynamics study of the SAE car is made to reduce the drag force. The study was performed using the CFD package. The main goal of this study is to enhance the stability of the vehicle and reduce the drag. With this the track performance will be increased also the resistance of air to the vehicle gets reduced. The CFD analysis is done on full scale model. The aerodynamic study is conducted in the ANSYS Fluent software to perform a turbulent stimulation (using k - ϵ model) of the air flow on the SAE car. The results are graphically shown with co – efficient of drag, velocity contour.

Cutting out the section of firewall and providing wing at front end. Drag co-efficient is found to get reduced from 0.85 for the standard race car to 0.70 for the modified car with front wing, whereas negative lift is increased from 0.2 for standard race car to - 0.25 for the model 3. The pressure at firewall found to be reduced for the modified cars due to providing space to flow the air through cut out section, where flow remains attached and helps to decrease the drag. Thus overall pressure near the driver head region is reduced from 340Pa to 80 Pa. for the modified car with front wing. Velocity of air is found to be increase below the stagnation point of car from 26m/sec to 32m/sec for model 3. Whereas at the rear end more wake region is found for standard race car. Model 3 having wing at the front end and having cut section at firewall shows less drag and lift, shows better aerodynamics characteristics than other two models.[10]

III. CONCLUSION

From the above section we can conclude that CFD is effective tool for the measurement of aerodynamic forces. CFD is one of the best method to find the aerodynamic force which take minimum time to calculate it. To increase the aerodynamic performance of car there should be focus on the various technologies which can reduce the aerodynamic forces. By adding the devices on the car drag coefficient can be reduced to 3 to 4% and hence the performance of car can increase.

REFERENCE

[1] Anu MariaState "INTRODUCTION TO MODELING AND SIMULATION", Proceedings of the 1997 Winter Simulation Conference ed. S. Andradóttir, K. J. Healy, D. H. Withers, and B. L. Nelson
[2] Ravindra A. Viveki, Dr.N.K.Chougule 2015 "Computational Fluid Dynamics Optimization of Roof Curvature for Drag Reduction of Passenger Car" International Engineering Research Journal , 1(5)2015 Page 224-231.
[3] Emmanuel Guilmineau " Computational study of flow around a simplified car body" Journal of Wind Engineering and Industrial Aerodynamics 96, 2008, page 1207–1217.
[4] Y. Wang, Y. Xin, Zh. Gu, Sh. Wang, Y. Deng and X. Yang " Numerical and Experimental Investigations on the Aerodynamic Characteristic of Three Typical Passenger Vehicles" Journal of Applied Fluid Mechanics, Vol. 7, No. 4, pp. 659-671, 2014.
[5] K.Selvakumar, Dr.K.M.Parammasivam " Experimental Investigations On Optimisation Of Aerodynmaic Charactericstics In A Hatchback Model Car 3 Using Vortex Generators" The Eighth Asia-Pacific Conference on Wind Engineering, December 10–14, 2013
[6] Bhagirathsinh H Zala, Harilal S Sorathia, Dinesh L. Suthar, Vashant K. Pipalia 2015 "Comparative assessment of drag force of sedan car model by computational fluid dynamics and experimental method" International Journal of Advanced Engineering Technology 7(1) 2015, page 6-10.
[7] R. B. Sharma1, Ram Bansal "CFD Simulation for Flow over Passenger Car Using Tail Plates for Aerodynamic Drag Reduction" Journal of Mechanical and Civil Engineering 7(5) 2013, page 28-35..
[8] Chenguang Lai, Yasuaki Kohama Shigeru Obayashi Shinkyu Jeong " Experimental and Numerical Investigation on the Influence of Vehicle Rear Diffuser Angle on Aerodynamic Drag and Wake Structure" International journal of Automobile Engineering 2(2011)/47-53.
[9] Zhu Hui, YU Hao, Yang Zhigang "Numerical analysis on effect of BACK/FRONT windshield and hood angle on automotive aerodynamic force" 2010.
[10] Sneh Hetawal, Mandar Gophane, Ajay B.K., Yagnavalkya Mukkamala "Aerodynamic Study of Formula SAE Car" Procedia Engineering 97 (2014) 1198 – 1207