Journal of Chemical and Pharmaceutical Sciences www.jchps.com ISSN: 0974-2115

ANALYSIS OF DRAG AND LIFT PERFORMANCE IN SEDAN CAR MODEL USING CFD

P.N.Selvaraju*, Dr.K.M.Parammasivam, Shankar, Dr.G.Devaradjane
Anna University, Chennai, India
***Corresponding author: E.Mail: selvarajumit@yahoo.co.in**

ABSTRACT

Aerodynamic force plays an important role in vehicle performance and its stability when vehicle reaches higher speed. Nowadays the maximum speed of car has been increased above 180 km/hr but at this speed the car has been greatly influenced by drag and lift forces. So the researchers are mainly focused in reduction of co-efficient of drag and lift in car model at higher speed. Even though the various techniques are found by researchers for improving vehicle performance and its stability still we are in need of further improvement So we are implementing vortex generator as a aerodynamic add on device at rear portion of vehicle . The various yaw angles and location of vortex generator are analyzed to obtain the efficient one to reduce the aerodynamic forces. An approximate outer profile of the typical sedan car body (Hyundai Elantra) which has a Co-efficient of drag value CD (0.35) has been generated in two configurations of with and without vortex generator by using solid modeling software and it has been analyzed using computational fluid dynamics (CFD) tool to reduce the aerodynamic drag and lift forces. Results show good improvement in reduction of above two forces by implementation of vortex generators on the car body.

Keywords: Drag force, Lift force, Side force, CFD, Vortex generator, Yaw angle.

INTRODUCTION

Aerodynamics is a branch of fluid dynamics concerned with studying the motion of air, particularly when it interacts with a moving object. Aerodynamics is also a subfield gas dynamics, with much theory shared with fluid dynamics. Aerodynamics is often used synonymously with gas dynamics, with the difference being that gas dynamics applies to all gases. Understanding the motion of air around an object enables the calculation of forces and moments acting on the object. Typical properties calculated for a flow field include velocity, pressure, density and temperature as a function of position and time. By defining a control volume around the flow field, equations for the conservation of mass, momentum, and energy can be defined and used to solve for the properties. The use of aerodynamics through mathematical analysis, empirical approximation and wind tunnel experimentation form the scientific basis. Aerodynamics and its analysis are basically divided into two major sub-categories, namely the external and internal aerodynamics. External aerodynamics is the study of flow around solid objects of various shapes. Evaluating the lift and drag on an airplane, the shock waves that form in front of the nose of a rocket, or the flow of air over a wind turbine blade are examples of external aerodynamics. On the other hand, internal aerodynamics is the study of flow through passages in solid objects. For instance, internal aerodynamics encompasses the study of the airflow through a jet engine or through an air conditioning pipe and other internal flow conditions.

The vehicle aerodynamic flow process is fall into three types (i) Flow of air around the vehicle. (ii) Flow of air through the vehicle body. (iii) Flow of air within the vehicle machinery.

AERODYNAMIC FLOW FIELD OF A CAR BODY

The Fig. 1.0 shows the streamline of an external flow around a stationary vehicle. When the vehicle is moving at an undistributed velocity, the viscous effects in the fluid are restricted to a thin layer called boundary layer. Outside the boundary layer is the in viscid flow. This fluid flow imposes pressure force on the boundary layer. When the air reaches the rear part of the vehicle, the fluid gets detached. Within the boundary layer, the movement of the fluid is totally governed by the viscous effects of the fluid.

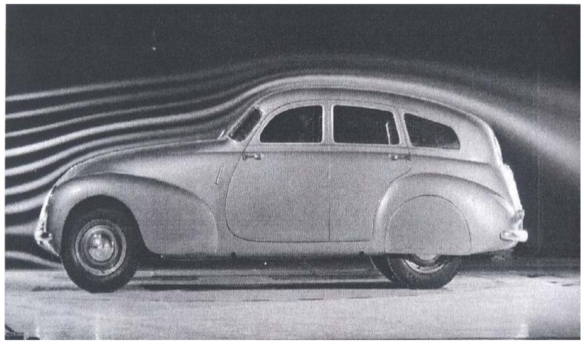

Fig. 1.0 External Flow Field over the Car Body.

The boundary does not exist for the Reynolds Number which is lower than 104. The Reynolds number is dependent on the characteristic length of the vehicle, the kinematic viscosity and the speed of the vehicle. Apparently, the fluid moving around the vehicle is dependent on the shape of the vehicle and the Reynolds number. There is another important phenomenon which affects the flow of the car and the performance of the vehicle. This phenomenon is commonly known as 'Wake' of the vehicle. When the air moving over the vehicle is separated at the rear end, it leaves a large low pressure turbulent region behind the vehicle known as the wake. This wake contributes to the formation of pressure drag, which is eventually reduces the vehicle performance.

FACTORS CONTRIBUTING TO FLOW FIELD AROUND VEHICLE

The major factors which affects flow field around vehicle are the boundary layers, separation of flow field, friction drag and lastly the pressure drag.

(1) Boundary layer: The Aerodynamics boundary layer was first defined by the Aerodynamic engineer 'Ludwig Prandtl' in the conference at Germany. This allows aerodynamicists to simplify the equations of fluid flow by dividing the flow field into two areas: one inside the boundary layer and the one outside the boundary layer. In this boundary layer around the vehicle, the viscosity is dominant and it plays a major role in drag of the vehicle. The viscosity is neglected in the fluid regions outside this boundary layer since it does not have significant effect on the solution. In the design of the body shape, the boundary layer is given high attention to reduce drag. There are two reasons why designers consider the boundary layer as a major factor in aerodynamic drag. The first is that the boundary layer adds to the effective thickness of the body, through the displacement thickness, hence increasing the pressure drag. The second reason is that the shear forces at the surface of the vehicle causes skin friction drag.

(2) Separation: During the flow over the surface of the vehicle, there is a point when the change in velocity comes to stall and the fluid starts flowing in reverse direction. This phenomenon is called 'Separation' of the fluid flow. This is usually occurred at the rear part of the vehicle. This separation is highly dependent on the pressure distribution which is imposed by the outer layer of the flow. The turbulent boundary layer can withstand much higher pressure without separating as compared to laminar flow. This separation causes the flow to change its behavior behind the vehicle and thereby affect the flow field around the vehicle. This phenomenon is the major factor to be considered while studying the wake of the vehicle.

(3) Pressure drag: The blunt bodies like large size vehicle show different drag characteristics. On the rear part of such vehicles, there is an extremely steep pressure gradient which leads to the separation of the flow separation in viscous flow. The front part of the flow field shows high pressure value, whereas on the rear part flow separates leading to a high suction in the area. As we integrate the force component created by such high change in pressure, the resultant is called as 'Pressure Drag'. This factor is affected by the height of the vehicle as well as the separation of the flow field.

(4) Friction drag: Each wall surface or material has a distinct friction which resists the flow of fluids. Due to molecular friction, a stress acts on every surface of the vehicle. The integration of the corresponding force component in the free stream direction leads to a friction drag. If the separation does not occur, then friction drag is one of the main reasons to cause overall drag.

FORCES AND MOMENT ON VEHICLE

When the vehicle is moving at a considerable speed, the air passing over it imposes various forces and moment on the vehicle. The Fig 1.1 shows the detailed sketch view of the various forces and moment acting on the vehicle body.

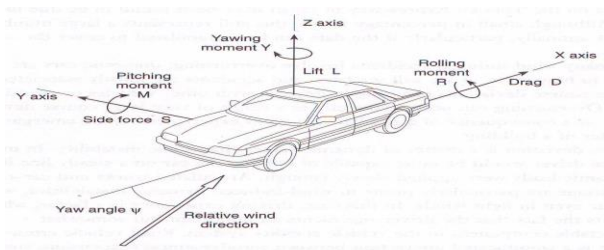

Fig 1.1 Aerodynamic Forces and Moments Acting On a Vehicle

The vertical force acting on the body causes the vehicle to get lifted in air as applied in the positive direction, whereas it can result in excessive wheel down force if it is applied in negative direction. Engineers try to keep this value to a required limit to avoid excess down force or lift. Aerodynamic drag force is the force acting on the vehicle body resisting its forward motion. This force is an important force to be considered while designing the external body of the vehicle, since it covers about 65% of the total force acting on the complete body. Crosswinds produce a side force on a vehicle that acts at the middle of the wheelbase, and when the crosswinds do not act at the middle of the wheelbase a yawing moment is produced.

SCOPE OF AUTOMOBILE AERODYNAMICS

The reduction of vehicle drag is a key motor for improvement of numerical and experimental tools. However, there are many other aerodynamic aspects such as crosswind stability, unsteadiness from passages of tunnels, platforms or other vehicles, ballast projection for high-speed trains, aero acoustics and soiling which require new improved approaches in flow predictions .Advancement in both experimental and numerical techniques has occurred in recent years. More advanced facilities and experimental techniques have been developed on the experimental side .Introduction of time-dependent simulations is the most important improvement in the numerical vehicle aerodynamics. Furthermore, both approaches are today used in development of techniques for improvement of vehicle properties by flow control or aerodynamic shape optimization. Aerodynamics is used by design engineers for cooling the engines, improving the performance of the vehicle, enhancing the comfort of the rider, stabilizing the car in external wind conditions and also increasing the visibility of the rider.

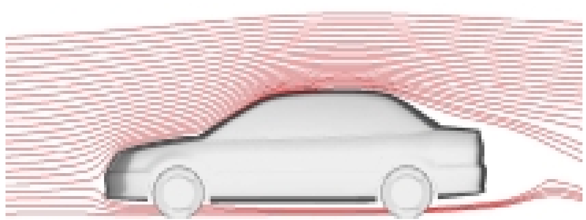

Fig 1.2 Flow Pattern over the Sedan Car Body

Although aerodynamics has so many tasks in its basket, this work concentrates on external devices which affect the flow around the automobile body to reduce the resistance of the vehicle in normal working conditions.

METHODS OF AERODYNAMIC TESTING

The ideal method of testing and exploring car aerodynamics would be road testing in natural conditions. Only then can all factors affecting the aerodynamics, such as rotating wheels, natural atmospheric wind conditions and underbody flow conditions, be completely accounted However, road testing has several challenges to surmount, especially regarding problems with separating drag measurements from rolling resistance, frictional and mechanical losses. Also, the equipment has to be carried with the car, which causes practical problems. Road testing was found impractical for the purposes of this project, and will not be further discussed. The two main alternative test methods, which are the most commonly used in aerodynamic development and testing of cars, are Computational Fluid Dynamics (CFD) and wind tunnel testing.

WINDTUNNEL TESTING

Wind tunnel testing has the big advantage that once the vehicle model is produced and rigged in the wind tunnel test section, it can quickly provide highly accurate data. Data for different boundary conditions, such as different wind speeds and yaw angles, can be acquired quickly. If similar changes in conditions are done on a computer model, the whole simulation has to be run over again for each case.

On the other hand, wind tunnel testing can be both highly costly and time consuming. The wind tunnel itself is a huge investment, and the production of prototypes can be very expensive. Small changes in design will take much more time to implement on a physical prototype than on a computer model. The accuracy of the wind tunnel measurements is affected by several factors, including blockage, scaling effects and the moving road problem, and the reliability and the validity of the data need to be evaluated in each case.

The wind tunnel is used for solving problem of aeronautical, space, automobile and civil engineering structures, are best obtained rapidly, economically and accurately by testing the scaled models, and sometimes actual structure in wind tunnels. The size, speed and other environmental conditions of tunnel are determined by actual users, the speed determines the type of the tunnel namely subsonic, near-sonic, transonic, supersonic and hypersonic. While the speeds of these tunnels are obliviously named with reference to the sonic, velocity, the low speed tunnel which is of our concern is below 300 mph. An alternative definition to the low speed tunnel would be the tunnel where the compressibility of air is negligible.

COMPUTATIONAL FLUID DYNAMICS

Computational fluid dynamics (CFD) is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. Even with high-speed supercomputers only approximate solutions can be achieved in many cases.

INTRODUCTION TO VORTEX GENERATORS

In areas where the body transitions at a rate of more than 12 degrees, vortex generators, diffusers, very short fairings or other devices can be used to "trip the airflow". The idea is that areas like the transition between the roof and front window or rear window on the average car creates a large vortex. Any large vortices effectively grab the car and try to hold it back as it tries to slip through the air. If the air that makes up the vortex can be tripped before it leaves back of the car, it will make smaller vortices, which will have a smaller effect on the overall aerodynamics of the vehicle. Measurement of the effects of these devices at highway speeds has been difficult to obtain.

A well-known example for intensifying the flow separation delaying effect is utilizing a dimple. Adding dimple-shaped pieces can lower the CD to a fraction of its original value. This is because dimples cause change in the critical Reynolds number. There are reported examples of aircraft wings controlling the boundary layer, in which vortex generators successfully delayed flow separation even when the critical Reynolds number is exceeded although the purpose of using VG is to control flow separation at the roof end of a Sedan, it is the same analogy of using VG on aircraft.

PROJECT MODEL

The approximate outer profile of (Hyundai Elantra) sedan car model without any aerodynamic devices by using 3D modeling software is shown

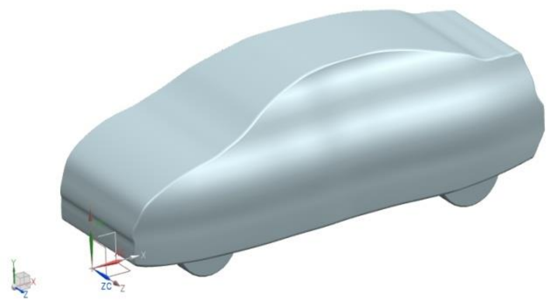

Fig 1.3 Sedan car model without vortex generator

The created model is to be analyzed to determine the current scenario in aerodynamics forces. In order to reduce aerodynamic forces the vortex generator is

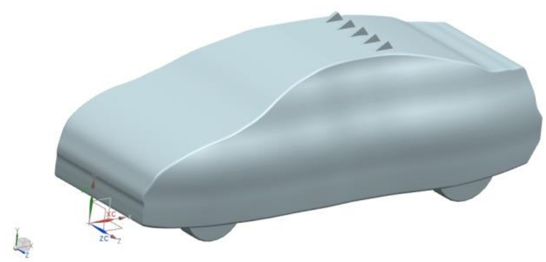

Fig 1.4 Sedan car model with vortex generator adopted then it is to be analyzed to determine the efficient yaw angle of vortex generator which is the aim of this project.

METHODOLOGY

A brief description of the methodology is as follows. Initially, the identification and definition of Problem is the first step of methodology then according to the problem definition approximate model is created using UNI-GRAPHICS modeling software. The created model is to be meshed using for that it was imported into the meshing software ICEM-CFD in the required format. The model which represents a single volume is being segregated into two volumes, fluid and solid. Then the two volumes where meshed separately with Quad elements and the necessary cell zones were specified. In the next step, model in the required format is to be imported into the analysis software then required boundary conditions are to be applied. Finally, the problem is being initialized and an efficient, iterative scheme with solution algorithm was used to solve the problem. The analysis is to be carried out repeatedly until the optimal result is obtained (i.e.) the efficient yaw angle of the vortex generator. In order to reduce lift and drag co-efficient of selected car model.

RESULT AND DISCUSSION

The analysis reports of created sedan model by using CFD in k-ε model is carried out based on two configurations without and with aerodynamic add-on devices mounted on the rear end of vehicle body, the boundary conditions are assumed as for inlet zone velocity of 100 km/hr and outlet zone pressure is set to zero Pascal.The graphical representation of total pressure occurs in different portions of without VG car body. The total pressure contour plot of sedan model with aerodynamic add-on device called VG over the roof of the body.

The results clearly shows that pressure occurs in roof of car model without VG is higher than at model having VG over the body in roof portion so it is confirmed that drag and lift force is reduced to some extent due to vortex layer formed at rear portion of car body by vortex generator. Flow pattern at rear portion of sedan model without and with VG as shown in figure1.7 and figure 1.8.From the figure the flow separation is delayed due to the addition of VG. This causes for drag force reduced to 6% compare with model without VG.

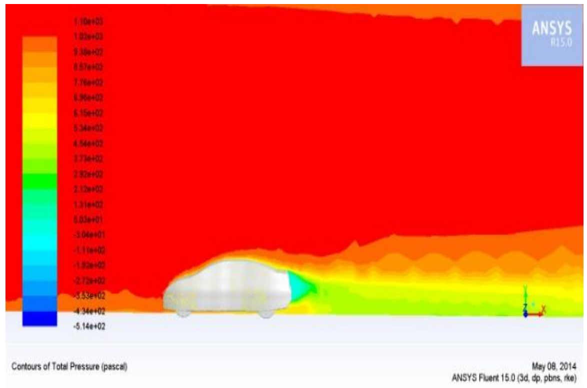

Fig 1.5 Contours of total pressure in the model1

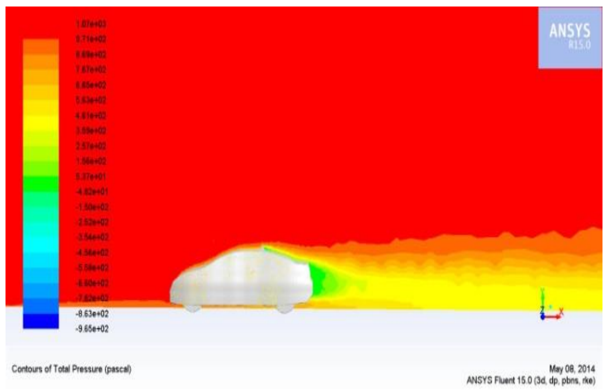

Fig 1.6 Contours of total pressure occurs in model2

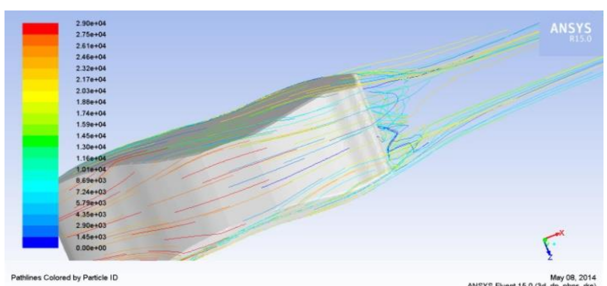

Fig 1.7 flow pattern in the model1.

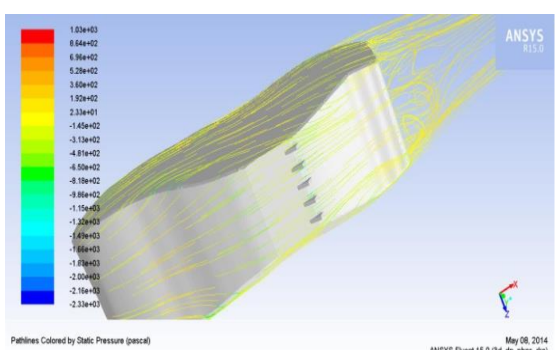

Fig 1.8 flow pattern in the model 2.

Table 1.1 Aerodynamic force performance of car model with and without VG

Model Configuration	VG Yaw angle in degree	Coefficient of drag (CD)	Coefficient of lift (CL)
Model 1	Base model	0.3120	-0.2267
Model 2	0 º	0.2934	-0.2790

CONCLUSION

This project is mainly concentrating on without changing the outer profile of the car body by the adoption of vortex generator as add on device to reduce the drag and lift force of the sedan type car body. The approximate outer profile of the sedan type car body (Hyundai Elantra) is generated by using 3D modelling software. Then it is imported for meshing and analysed for aerodynamic forces with and without vortex generator. Therefore without modifying the car profile the reduction of 6% of drag and negative lift force can be attained by adopting vortex generator as an aerodynamic add on device.

REFERENCES

Angelis, W., D. Drikakis, F. Durst., W. Khier.: Numerical and experimental study of the flow over a two dimensional car model, Journal of wind engineering and industrial engineering. 62, 57-69 (1996)

Baker, C. J., and N. D Humphreys.: Assessment of the adequacy of various wind tunnel techniques to obtain aerodynamic data for ground vehicles in crosswinds". Journal of Wind Engineering and Industrial Aerodynamics, 60, 49–68 (1996)

Cogotti, A.: Evolution of performance of an automotive wind tunnel, Journal of Wind Engineering and Industrial Aerodynamics 96, 667–700 (2008)

Dr. Ing. Thomas Schultz.: Progress in CFD validation in aerodynamics development, auto technology Dissertation University Stuttgart 3, 28-33 (2009)

Drage, P., A. Gabriel., and G. Lindbichler.: Efficient Use of Computational Fluid Dynamics for the Aerodynamic Development Process in the Automotive Industry, Applied Aerodynamics 26, 1-15 (2008)

Emmanuel Guilmineau.: Computational study of flow around a simplified car body, Journal of Wind Engineering and Industrial Aerodynamics 96, 1207–1217 (2008)

Ha, J., S. Jeong., and S. Obayashi.: Drag reduction of a pickup truck by a rear downward flap, International Journal of Automotive Technology, 12(3), 369−374 (2011)

Hasan Ali, M.D., Mohammad Mashud., Abdullah Al Bari and Muhammad Misbah-Ul Islam.: Aerodynamic drag reduction of a car by vortex generation International journal of mechanical engineering, 2(1), 12-21 (2012)