Influence of Wake Characteristics of a Representative Car Model by Delaying Boundary Layer Separation

Gopal P.* and Senthilkumar T.

Department of Automobile Engineering, University College of Engineering (BIT Campus), Anna University, Tiruchirappalli – 620 024, India

Abstract

The main causes of aerodynamic drag for automotive vehicles are the separation of flow near the vehicle’s rear end. By reducing the drag it is possible to increase the fuel economy. To delay flow separation, vortex generators are tested for application to the roof end of a representative car model. It is commonly used on aircraft to prevent flow separation. Vortex generators themselves create drag, but they also reduce drag by preventing flow separation at downstream. In this paper the effect of vortex generators in the flow field and the mechanism by which these effects take place are studied. The paper also illustrates the computational fluid dynamics analysis of vortex generators in the representative car model. Various flow characteristics like pressure and velocity distribution, path line characteristics, vector flow, and wake studies at the rear end are discussed in detail.

Key Words: Aerodynamic Drag, Passive Control Device, Drag Characteristics, Drag Coefficient, Vortex Generators

1. Introduction

The constant need for better fuel economy, greater vehicle performance, reduction in wind noise level, improved road holding and stability for a vehicle on the move, has prompted vehicle manufacturers to investigate the nature of air resistance under various operating conditions. A vehicle with high drag resistance tends to hinder its acceleration marginally but it inhibit its maximum speed and increase the fuel consumption at higher speeds. Even if the goal of motor vehicle aerodynamics is often considered to be essentially the reduction of aerodynamic drag, the scope and the applications of aerodynamics in motor vehicle technology are much wider [1–3]. The important aspects of aerodynamic studies are the reduction of aerodynamic drag, reduction of the side force and the yaw moment, which have an important influence on stability, handling and reduction of aerodynamic noise. Similarly an important issue for acoustic comfort, the reduction of dirt deposited on the windows and lights when driving on wet road also plays a major role in this aspect [4,5]. While focusing for safety this aspect can be extended to the creation of spray wakes that can reduce visibility for other vehicles following or passing the vehicle under study [6].

2. Drag Characteristics in a Car Model

2.1 Boundary Layer

Air has viscosity, that is, there is internal friction between adjacent layers of air, whenever there is relative air movement and consequently energy is dissipated. When air flows over a solid surface a thin boundary layer is formed between the main airstream and the surface. Any relative movement between the main airstream and the surface of the body, then energy dissipation takes place within this boundary layer via the process of shearing of adjacent layers of air. When air flows over any surface, air particles would be in intimate contact with the surface and attach themselves to the surface so that rela-
tive air velocity at the surface becomes zero as shown in Figure 1.

The relative speed of the air layers adjacent to the attached air surface film is very low; however, the next adjacent layer will slide over an already moving layer so that its relative speed would be higher. Hence the relative air velocity further out from the surface rises progressively until it attains the unrestricted free air stream speed.

2.2 Flow Separation and Reattachment

The stream of air flowing over a car’s body tends to follow closely to the contour of the body unless there is a sudden change in shape as shown in Figure 1. The front bonnet (hood) is usually slightly curved and slopes up towards the front windscreen. From here there is an upward windscreen tilt (rake) which is, followed by a curved but horizontal roof. The rear windscreen then tilts downwards where it either merges with the boot (trunk) or continues to slope gently downwards until it reaches the rear end of the car.

An example of flow separation followed by reattachment can be visualized with air flowing over the bonnet and front windscreen as shown in Figure 1. If the rake angle between the bonnet and windscreen is large, the streamline flow may separate from the bonnet and then reattach itself near the top of the windscreen or front end of the roof. The space between the separation and reattachment can be occupied by circulating air which is referred to as a separation bubble, and if this rotary motion is vigorous a transverse vortex would be established [8].

Thus, the drag characteristics of a car is nothing but the resistance force of air (fluid) which is offered to the car (vehicle) to reduce its speed. The study of these characteristics of drag leads to implementation of modern techniques to reduce the resistance and to increase the speed of the vehicle thereby increasing fuel efficiency and economy.

2.3 Vortex Generators as Passive Control Device

Vortex generators are aerodynamic surface, consisting of a small vane or bump that creates a vortex. Vortex generators can be found on many devices, but it is often used in aircraft design. Vortex generators delay flow separation and hence are found on the external surfaces of vehicles where flow separation is a potential problem.

They are typically rectangular or triangular in shape, about 80% as tall as the boundary layer. Vortex generators have many geometrical parameters like general shape, height, length and angle to the main flow direction. Further, it is necessary to specify the chord wise position and span wise spacing on the blade.

Wake reduction can be done by passive control device as well as flow control device. The passive flow control method is chosen in this work to minimize the wake region behind the vehicle. Vortex generators are chosen as passive device which creates vortices, where the laminar flow gets converted into turbulent flow and so the boundary layer separation gets delayed. Due to this, the wake region behind the vehicle is minimized.

3. Experimental Details

3.1 Design of Vortex Generators

A single vane type delta (triangular) shaped vortex generator was chosen. Due to their simplicity, widespread usage and low drag device than any other type makes the vane type more suitable for attaching on the vehicle body. Delta shaped vortex generators are most commonly used on aircraft wings. In connection with the height, the thickness of the boundary layer is measured based on the assumption that the optimum height of the vortex generators would be nearly equal to the boundary layer thickness. Figure 2 shows the velocity profile on the vehicle’s roof considering flow over flat plate concept based on local x position. It is found that the boundary layer thickness at the roof end immediately in front of the separation point is to be about 2 mm [9]. Consequently, the optimum height for the vortex generators are
estimated to be up to approximately 2 mm. The thickness of vortex generators are fixed at 0.5 mm.

Length was taken in proportion of the height of the vortex generators. In the present experiments, vortex generators are installed with a yaw angle of 15° to the air flow direction and with the length to height ratio of 2. Based on this ratio, a single row of vortex generators are positioned on the roof with 8 numbers of vortex generators as shown in Figure 3.

This describes the spacing between vortex generators in a row. A single row of vortex generators are fixed at 5 mm from the roof end. This point was fixed, based on the boundary layer measurements and separation point of the stream line on the roof. The number of rows are limited to one in order to minimize weight and potential manufacturing cost. The delta shaped vortex generators are installed at interval to height ratio of 6. But the airflow direction was found to be different between sideways positions on the roof. The delta-wing-shaped vortex generators must be installed at an angle of 15° against the vehicle centerline for the central position, whereas they must be installed at an angle near 0° for outermost positions. The arrangement of vortex generators are shown in Figure 4.

3.2 Scale Model and Experimental Setup

The 1:15 scaled down test model is used in this study. The length, breadth and height of the scaled model was 0.295 m, 0.108 m and 0.1 m respectively. Thickness of the sheet metal used was 0.5 mm. The vortex generators are cut into pieces from the sheet metal and they were fixed on to a base plate by gas welding process. The base plate with vortex generator was fastened to the roof of scaled model by means of flat headed screw.

To measure the static pressure on the body, 0.2 mm diameter holes were drilled on the centre line of the vehicle body starting from the front end along the roof to the rear end of the vehicle as shown in Figure 5. 15 pressure tappings are connected to micro manometer using pressure tubes. An open circuit wind tunnel with a test section of 0.09 m² was used.

The schematic of the wind tunnel is shown in Figure 6. The total length of the wind tunnel was 6 m and the test section length was 1 m. A 2.5 kW electric motor was used for suction. The wind tunnel tests were conducted at positive and negative yaw angles between ±15°. The frontal area of the scale model of the vehicle is 0.0108 m². The blockage ratio is calculated to be about 9.2%. The relative air speed was measured by using micro
A micro manometer has an accuracy of ±0.5%. Velocity uniformity is ±0.96% which is 1% as given in SAE wind tunnel test procedure.

3.3 Experimental Procedure

The experiment was done with an objective to measurement of drag force, pressure variations and relative speed with varying speeds along the centre line of the vehicle under straight wind conditions. The pressure points are observed on the front, the roof and the rear. The pressure tubes are connected from the model to 20-way single selection box and then to the digital manometer and the pressure difference is observed. The load cell transducer is directly attached to the platform of the vehicle model. The platform is used to measure the drag and lift forces in terms of change in the resistance of the load cell transducer.

4. CFD Simulation on the Wake of a Car Model

4.1 Governing Equations

In fluid dynamics there are three governing equations, which describe the behavior of the flow, they are the continuity, momentum and the energy equations. These equations becomes rather complicated and can’t be solved analytically so numerical simulations are required. Since vehicles travels at relatively low speed, Mach number < 0.3, and constant temperature the flow can be assumed incompressible and isothermal [9], the energy equation can then be neglected.

Navier-Stokes equations are usually expressed for an incompressible Newtonian fluid with constant viscosity. An incompressible fluid is a fluid where the divergence of the velocity is zero, and a Newtonian fluid is a fluid for which stress versus strain rate curve is linear. The Navier-Stokes equations can then be expressed as in Eq. (1), three partial non-linear differential equations, one for each velocity vector [10].

\[
\frac{\rho}{\rho_0} \frac{d \rho}{d \tau} + \frac{\rho}{\rho_0} \left( \frac{d^2 u}{d \tau^2} + \frac{d^2 v}{d \tau^2} + \frac{d^2 w}{d \tau^2} \right) = \rho \frac{d u}{d \tau} \quad \& \quad \frac{\rho}{\rho_0} \left( \frac{d^2 v}{d \tau^2} + \frac{d^2 w}{d \tau^2} \right) = \rho \frac{d v}{d \tau} \quad \frac{\rho}{\rho_0} \left( \frac{d^2 w}{d \tau^2} \right) = \rho \frac{d w}{d \tau}
\]

Since the flow is assumed incompressible the continuity equation will be as in Eq. (2).

\[
\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0
\]

Together with the Navier-Stokes equations the continuity equation gives four unknowns, \( u, v, w \) & \( p \), which would be solved with differential equations.

In order to solve these equation and analyze the flow, the simplest approach is the Reynolds decomposition which is also called Reynolds-Averaged Navier Stokes (RANS) [11]. In the RANS approach the instantaneous velocity and pressure is split into two parts, an average part and a fluctuating part, Eqs. (3) and (4).

\[
\overline{u} = \frac{1}{T} \int u dt
\]

\[
p = \overline{p} + p', \ u = \overline{u} + u', \ v = \overline{v} + v', \ w = \overline{w} + w'
\]

Inserting Reynolds decomposition into Navier-Stokes equation (x-direction) and in the continuity equation will results in new fluctuating terms.

\[
\frac{\partial \overline{u}}{\partial x} + \frac{\partial \overline{v}}{\partial y} + \frac{\partial \overline{w}}{\partial z} = 0
\]
\[
\rho \frac{\partial u}{\partial t} + \rho \left( \overrightarrow{\mu} \frac{\partial u}{\partial x} \cdot \nabla u - \overrightarrow{\mu} \frac{\partial u}{\partial x} \right) + \frac{\partial \left( \overrightarrow{\mu} \frac{\partial u}{\partial y} \cdot \nabla u \right)}{\partial y} = \rho \frac{\partial u}{\partial t} \ (6)
\]

Eq. (6) now consists of new unknown terms like \( \overrightarrow{\mu} \), also called for Reynolds stresses. Since the number of unknowns are greater then the number of equations a so called closure problem is generated, the extra stress terms has to be modeled to get a closed equation system. This is done by using turbulence models.

The \( k-\varepsilon \) model is the most commonly used turbulence model in the industry, this is due to the robustness of the model which gives safe convergence. The \( k-\varepsilon \) model is an Eddy Viscosity model, which means that the turbulence is modeled by adding the turbulence viscosity, \( \mu_t \). This model is a semi-empirical method based on how the kinetic energy, \( k \), is transported and its dissipation rate, \( \varepsilon \). The transport equation for \( k \) is derived exact while the transport equation for \( \varepsilon \) is derived from physical reasoning [12]. The biggest eddies gets their kinetic energy, \( k = (\Delta u^2 + \Delta v^2 + \Delta w^2)^{1/2} \), from the main flow. The energy is transmitted into smaller eddies and ends up as internal energy. Since the \( k-\varepsilon \) model is a RANS-model and is using time averaged terms the model would miss differences in the gradients during very short time steps. In order to achieve a more accurate solution of the flow a new model has been developed called the \( k-\varepsilon \) realizable model [12]. In the realizable \( k-\varepsilon \) model the normal and molecular stresses are taken into account to some extent. The relationship between the kinetic energy, the dissipation rate and the turbulent viscosity is defined in Eq. (7) [13].

\[
\mu_t = C_{\mu} \frac{k^2}{\varepsilon} \ (7)
\]

As mentioned an extra term, \( \overrightarrow{\mu} \), is included in the Navier-Stokes equation. In the \( k-\varepsilon \) model the extra stress term is modeled with a Boussinesq assumption, see Eq. (8).

\[
-\overrightarrow{\mu} \overrightarrow{u} = \mu \left( \frac{\partial u}{\partial x} + \frac{\partial u}{\partial y} \right) - \frac{2}{3} \left( \mu + \mu_t \right) \frac{\partial u}{\partial y} \delta \ (8)
\]

4.2 Creating the Geometry

Geometry operations, such as creating volume and splitting or merging edges or faces are done by using the co-ordinates of the car model. The Figure 7 represents the 1/15th scaled down geometry of the car model used for CFD analysis.

4.3 Meshing the Model

The partial differential equations that govern fluid flow and heat transfer are not usually amenable to analytical solutions, except for very simple cases. Therefore, in order to analyse fluid flows, flow domains are split into smaller sub domains (made up of geometric primitives like hexahedral and tetrahedral in 3D and quadrilaterals and triangles in 2D). The governing equations are then discretized and solved inside each of these sub domains. The sub domains are often called elements or cells, and the collection of all elements or cells is called a mesh or grid. The final meshes consist of 1.1 million fluid cells and the complete domain was divided to half using a symmetry plane to reduce the computational time. Different meshing characteristics were used for the different wind tunnel domains. Although the methods were different, the same configuration of hexahedral core cells are used for both the domains. As the grid dependency test in the \( k-\varepsilon \) turbulence model with hexahedral element in ANSYS Fluent results in a low discrepancy rate of 26% [14]. Hexahedral elements were used in an attempt to provide an accurate estimation of the velocity near the wall, when using the wall function, by keeping the y plus value within an acceptable range (20 to 200) [15]. In addition, these elements easily adjust to the complex bodies used in automobile and aerospace bodies. The mesh of a car model with vortex generators are shown in Figure 8.

Figure 7. Model of a car with vortex generators designed in gambit.
4.4 Simulation

Computer simulation is the discipline of designing a model of an actual or theoretical physical system, executing the model on a digital computer, and analysing the execution output. To understand the reality and all of its complexity, this must be build with artificial objects and dynamically act out roles with them. Computer simulation is the electronic equivalent of this type and it serves to drive synthetic environments as well as virtual worlds. Computer within the overall task of simulation, there are three primary sub-fields namely model design, model execution and model analysis. Hence the modelling of a car with and without vortex generators are executed and analysed the difference by using CFD simulation [16–18]. Numerical simulation was done using a commercial software ANSYS Fluent. The boundary conditions adopted for the numerical simulation are given in Table 1.

5. Results and Discussions

5.1 Validation of CFD Results

For the validation purpose, an experimental value of pressure coefficient of baseline testing at a Reynolds number of $\pm 3.29E5$ (experiments without vortex generators) with a flow velocity of 20 m/s was compared with

<table>
<thead>
<tr>
<th>Table 1. Boundary condition settings</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Magnitude (measured normal to boundary)</strong></td>
</tr>
<tr>
<td><strong>Velocity inlet</strong></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td><strong>Pressure outlet</strong></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td><strong>Wall zones</strong></td>
</tr>
<tr>
<td><strong>Fluid properties</strong></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
</tbody>
</table>
the simulated values as shown in Figure 9. Although the results of the pressure coefficient on the roof surface closely match the experimental results, at the front body absolute pressure shows a slight variation but the pattern of the plot is similar. The maximum value of pressure coefficient is found on the front wind shield of the vehicle model. It is revealed that the maximum deviation between the numerical simulated values and experimental values is about 1.9%.

5.2 Pressure Distribution

The stream of air flowing over a car’s body tends to follow closely to the contour unless there is a sudden change in body shape. At the location of vortex generators, the variation of static pressure values is in the narrow range of -6.10E2 to -5.27E2 and hence the grid resolution can meet the requirement of the turbulence model. The complete domain was divided to the half using the symmetry plane and the z value at the symmetry plane is zero.

From Figure 10 it can be seen that the pressure at the front end of the car is found to be maximum (Transition Point-1), because of the direct collision of the air flow with the vehicle. Thus the pressure is to be at its maximum. Whereas it tends to be decreased towards the surface and it again get increased with its collision at the wind-shield of the car (T.P-2). The flow of the air gets transformed at the front end of the roof (T.P-3) and the rear end of the roof (T.P-4). After the transition point 4, the air tends to flow at a very minimum pressure and thus creates wake and results in drag.

Figure 11 shows the variation of static pressure around the car with vortex generators. The vortex generators have been added to the rear end of the roof, to energize the flow and thus the separation of flow would be delayed and reduces the drag. From the analysis it can be observed that the flow is energized because of the vortex generators at the transition point 4 and the separation of flow takes place at a relatively longer distance that shows reduction in drag.

5.3 Velocity Distribution

The relative air movement at the rear of the boot can be slow to car speed, conversely its pressure would have again risen to the surrounding atmospheric pressure conditions. Thus allowing the random network of distorted molecules to move closer together to a more stable con-
dition in velocity distribution. As the air moves beyond the roof into the diverging wedge region it decelerates to cope with the enlarged flow space.

From the Figure 12 it can be inferred that the velocity at the front end of the car is very minimum whereas at the rear end of the car it is found to be higher than that of the front end. This shows the pressure is very low at the rear end of the car. The velocity at the rear end of the car could be minimized as if the pressure drop is reduced, thus it has been achieved with the help of the inclusion of vortex generators, and the drag has been minimized as well.

The Figure 13 shows that the velocity at the rear end has been reduced wake when compared to that of the model without vortex generators. Thus invariantly causes the reduction in the drag of the vehicle.

5.4 Vector Flow

Vector flow is the study of the direction of the flow of air around the car model. Vectors are represented by small arrows that denote the flow of the air around the car model. In general the velocity vector flow has been studied. From the Figure 14 it can be inferred that the air flow at the front end of the car is directly opposite towards the motion of the car. It causes direct collision and results in low velocity profile at the front end of the car. Similarly, at the rear end of the car the direction of flow changes and flows towards the motion of the car. It results in high velocity profile. It is visualised that there is a change in the direction of flow.

The Figure 15 shows that the directional change of the air at the rear end of the car tends to be minimal when compared to that of the one without vortex generators.
Thus in turn provides a low velocity change at the rear end of the car, causes the reduction in wake formation and drag of the vehicle.

5.5 Pressure Coefficient

Figure 16 shows the observations of the pressure coefficient along the surface of the car without vortex generators. It is observed that the value of pressure coefficient varies from -7 to 3 along the centre line of the vehicle. At a X/L ratio of 0.9 the pressure coefficient value is found to be around -0.75. This indicates the low pressure intensity at the rear end of the vehicle. It is also evident that the variation of pressure coefficient along with the position of the vehicle from its front end to the rear end. It is also inferred that the variations indicate the low pressure intensity at the rear end of the car model. An observations of the pressure coefficient along the surface of the car with vortex generator as shown in Figure 17. Here the pressure coefficient varies from -10 to 2.5. At X/L ratio of 0.9 the pressure coefficient value is around 0.9. Thus the values showed that the pressure coefficient is higher at the rear end of the car model when compared to that of the model without vortex generators.

5.6 Coefficient of Drag

From the observations of pressure coefficient, the overall drag coefficient has been derived. The overall drag coefficient of the car model without the vortex generators is thus found to be 0.086. The overall drag coefficient of the car model with the vortex generators is found to be 0.079. The results showed that the overall drag coefficient of the car model has been reduced significantly and it improves the overall efficiency of the vehicle.
Figure 14. Vectors of velocity at 20 m/s without vortex generators.

Figure 15. Velocity vectors at 20 m/s with vortex generators.

Figure 16. Variation of pressure coefficient along with x/L (without vortex generators).
6. Conclusion

The pressure variations around the car model shows that the pressure is found to be maximum at the front end of the car, whereas it reaches the minimum towards the rear end of the car. This results in increase of wake region at the rear end of the car model, and thus in turn leads to the increase in the drag of the vehicle. The drag coefficient of the representative car model in this case is found to be about 0.086. This drag is not desirable, as it may influence directly on the efficiency of the vehicle. Thus to reduce the drag of this vehicle, a series of vortex generators has been introduced in order to minimize the wake region at the rear end of the car model.

It is found that the drag is created in this model also, but this drag is reduced by preventing flow separation at downstream. The overall effect of vortex generators can be calculated by summarizing the positive and negative effects. The reduced wake region has its direct implication on the overall drag coefficient of the car model. In the car model with vortex generators, the overall drag coefficient is found to be about 0.079. The drag coefficient around the car model has been reduced significantly and thereby improves the vehicle’s efficiency.

References


*Manuscript Received: Dec. 12, 2012
Accepted: Jun. 5, 2013*