ABSTRACT: The aim of this research paper is to reduce the drag of SUV by using a vortex generator and to calculate the pressure and turbulence profile across the vehicle. The Ahmed Reference Model is taken as a benchmark test. Computational fluid dynamics (CFD) simulation with and without vortex generator is performed at different velocities across the SUV similar to TATA Sumo. The performance of Vortex generator is analyzed at different velocities to obtain the particular velocity at which it will have the minimum value of drag. The end results are henceforth analyzed and a comparative study has been performed with the experimental data given by Gopal and Senthikumar on SUV. And finally it is found that the 10% of drag reduction is achieved using vortex generator.

KEY WORDS: aerodynamics, Ahmed body, computational fluid dynamics, SUV, drag reduction.

1 INTRODUCTION

Aerodynamics these days is considered as an essential parameter for vehicular design. With the rise in inflation, fuel prices are ascending because of very limited resources of oil. And most of the South Eastern countries import a major proportion of oil needs. So it became necessary to incorporate aerodynamic concepts while designing the vehicle. Coefficient of drag value can be minimized up to a certain extent that results in more fuel efficiency. Drag coefficient plays very important role in the design of aerodynamic vehicle. Drag coefficient value depends upon the pressure variation on the vehicle. Because of this variation in the pressure, the flow separates from the rear end of the vehicle instantly resulting in obstruction to the forward motion of the vehicle. It is a challenging job for an engineer to design the rear end of the vehicle in such a way that it will delay the flow separation and produce less value of drag. There are numbers of passive devices available like Flaps, Deflector, Vortex Generator [1], Spoiler and Rear wings [2] by which we can reduce the value of coefficient of drag. Computational Fluid Dynamics (CFD) methodology was adopted by Xingjun Hu et al. [3] to minimize the value of drag at different diffuser angle of a sedan.

*Corresponding author e-mail: vir.singh@hotmail.com
The underline theme of this research article is to minimize the coefficient of drag of a vehicle by using vortex generator. A famous benchmark that is commonly used in the ground vehicle industry is the Ahmed body [4, 5]. The first part of this research paper is to compute the coefficient of drag of Ahmed body by using CFD and compare this value with the experimental value given by Deepak Kumar Kalyan and A.R. Paul [4]. The second part focuses on the reduction of coefficient of drag value for SUV by using Vortex Generator and compares this value with the experimental result given by [1]. For the analysis CFD package, Ansys 2014 is used to obtain the result. In a fixed domain we will calculate the value of drag at different Reynolds number and will analyze the flow pattern at different speeds. Variation of coefficient of pressure is also studied at different speeds. And finally the results we obtained are compared with the experimental result given in Ref. [1].

2 Numerical Method and Turbulence Modeling

Reynolds-averaged Navier-Stokes (RANS) [6, 8] is the earliest known methodology for turbulence modeling and is cost effective than Large Eddy Simulation (LES) [7, 8]. Using this approach, selective transport equations with new Reynolds stresses can also be solved. This approach is based on the decomposition of the flow variables into mean and fluctuating parts followed by time. In cases where density is not constant, it is advisable to take density weight to the velocity variations. This introduces a tensor of unknown variables of second order that need to be solved. Examples of RANS methods are K-ε methods, Mixing Length Model, and the Zero Equation model [8].

2.1 K-ε Model

Among two equations eddy viscosity models, the k-ε turbulence model is highly recommended. In comparison to Spalart-Allmaras model [8], the k-ε turbulence model involves complex mathematical calculations and is mostly used for low Reynolds number. The turbulence kinetic energy, k, and its rate of dissipation ε, are obtained from the following transport equations:

\begin{align*}
(1) \quad & \frac{\partial}{\partial t} (\rho k) + \text{div}(\rho k V) \text{div} \left( \frac{\mu_t}{\sigma_k} \text{grad} k \right) + 2 \mu_t S_{ij} - \rho \varepsilon \\
(2) \quad & \frac{\partial}{\partial t} (\rho \varepsilon) + \text{div}(\rho \varepsilon V) \text{div} \left( \frac{\mu_t}{\sigma_k} \text{grad} \varepsilon \right) + C_1 \varepsilon \left( \frac{\mu_t}{k} (2 \mu_t S_{ij} S_{ji}) \right) - C_2 \rho \varepsilon^2 \frac{\varepsilon}{k}
\end{align*}

\{Rate of change of k or \( \varepsilon \}\} + \{Transport of k or \( \varepsilon \) by convection\} = \{Transport of k or \( \varepsilon \) by diffusion\} + \{Rate of production of k or \( \varepsilon \)\} - \{Rate of destruction of k or \( \varepsilon \)\}
3 GEOMETRIC MODELING AND NUMERICAL SIMULATION

Ahmed body with 25 degree slant angle is taken as test sample and simulated using CFD. Dimensions of the flow field are $7 \times 3$ m. In order to minimize the estimated time, a two-dimensional model is taken as shown in Fig. 3.1.

Unstructured triangular mesh is used to calculate the variable at different nodes. To accurately measure the variables, meshing of the domain area close to the surface of the vehicle is coarser as shown in Fig. 3.2. Total numbers of triangular elements are 25,566. Figure 3.1 shows the zone name and boundaries of Ahmed body and flow domain.

Unstructured triangular mesh is used to calculate the variable at different nodes. To accurately measure the variables, meshing of the domain area close to the surface of the vehicle is coarser as shown in Fig. 3.2. Total numbers of triangular elements are 25,566. Figure 3.1 shows the zone name and boundaries of Ahmed body and flow domain.

Fig. 3.1. (Color online) Ahmed body with 25 degree slant angle with flow domain.

Fig. 3.2. (Color online) Triangular mesh of the domain of flow.
SUV similar to TATA Sumo is taken as a model to find out the drag reduction in a vehicle with and without vortex generator shown in Fig. 3.3. The model has a 1:15 scale ratio. The length and height of the scaled model are 0.295 m and 0.1 m, respectively. This model is analyzed in a flow domain shown in Fig. 3.4 having dimension of $7 \times 3$ m. Total numbers of triangular elements are 37,487 for unstructured mesh used for simulation.

The problem considered is unsteady turbulent flow. The problem is solved by commercial software package Ansys 14 and a post processor (Fluent) is used to examine the results. PISO is the prudent technique used for pressure measurement and QUICK approach is used for measuring the momentum, turbulent kinetic energy and turbulent dissipation.

Initial boundary conditions are given in Table 1.
Reduction of Drag of SUV Similar to TATA Sumo Using Vortex Generator

Table 1.

<table>
<thead>
<tr>
<th>Sample No</th>
<th>Input variable</th>
<th>Ahmed Body</th>
<th>SUV similar to TATA Sumo</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Inlet velocity</td>
<td>40 m/s</td>
<td>2.42, 3.47, 4.42 and 7.12 m/s</td>
</tr>
<tr>
<td>2</td>
<td>Outlet pressure</td>
<td>0 Pascal (reference)</td>
<td>0 Pascal (reference)</td>
</tr>
<tr>
<td>3</td>
<td>Viscous model</td>
<td>K-epsilon model</td>
<td>K-epsilon model</td>
</tr>
<tr>
<td>4</td>
<td>Turbulence intensity (%)</td>
<td>2.93047</td>
<td>2.93047</td>
</tr>
<tr>
<td>5</td>
<td>Turbulence viscosity ratio</td>
<td>10</td>
<td>10</td>
</tr>
<tr>
<td>6</td>
<td>Space</td>
<td>2 D</td>
<td>2 D</td>
</tr>
<tr>
<td>7</td>
<td>Convergence criteria</td>
<td>1e-03</td>
<td>1e-03</td>
</tr>
<tr>
<td>8</td>
<td>Symmetry</td>
<td>Top</td>
<td>Top</td>
</tr>
</tbody>
</table>

4 RESULTS AND DISCUSSION

The results obtained from CFD simulations are collected and plotted graphically. The turbulence and pressure contours show the turbulence viscosity and pressure at different points in the flow field. In the same manner, the coefficient of pressure is plotted against the x-coordinate of the SUV.

4.1 CFD ANALYSIS FOR AHMED BODY

4.1.1 RESIDUALS PLOTS

The residual plots are the variations in the characteristics (for example x-velocity or y-velocity or k, etc.) between two iterations. This implies that if we iterate, significant increase in low residual will have little impact on end results. The final results are obtained iteratively. During the iteration, the value of variable like velocity, turbulent viscosity and turbulent dissipation rate varies. To find the convergent point and to calculate the error in solution, we have to calculate the residuals of each variable. The Residual plot of variables of Ahmed body is shown in Fig. 4.1. This figure shows that the residual value is less than 0.01 that means the error is approximately 0.01.

4.1.2 TURBULENCE VISCOSITY CONTOURS

Figure 4.2 shows that the flow gets separated at the back end of the vehicle. As a result of which a wake zone gets created at the rear end. This wake or vortex opposes the frontward motion of the vehicle. In the present work, the value of wake is less as compared to the reference paper work.
4.1.2 TURBULANCE VISCOSITY CONTOURS

Fig. 4.1 shows that the flow gets separated at the back end of the vehicle. As a result of which a wake zone gets created at the rear end. This wake or vortex opposes the frontward motion of the vehicle. In the present work, the value of wake is less as compared to the reference paper work.

4.1.3 STATIC PRESSURE CONTOURS

Fig. 4.3 shows the difference in static pressure in the flow domain. This difference indicates how the flow is deaccelerating. It is clear from Fig. 4.3, that static pressure is highest (1340 Pa) at the front indicated by red colour and lowest (-1460 Pa). Whereas velocity exhibits opposite behaviour, i.e. minimum at front and maximum at top. At the rear end, pressure value is 1304 which is not same as that at the front. So this pressure gradient causes drag.

4.1.4 COEFFICIENT OF DRAG VS. FLOW TIME

Figure 4.4 shows the variation of coefficient of drag with flow time for Ahmed body. As the time passes, there is a decrement in the value of $C_d$. This occurs because during initial flow time air does not come in contact with the surface of vehicle, so
Reduction of Drag of SUV Similar to TATA Sumo Using Vortex Generator

4.1.4. COEFFICIENT OF DRAG VS FLOW TIME

Fig. 4.4 shows the variation of coefficient of drag with flow time for Ahmed body. As the time passes, there is a decrement in the value of $C_d$. This occurs because during initial flow time air does not come in contact with the surface of vehicle, so $C_d$ becomes high. Solution starts converging after a flow time of 0.05 s. At 0.2 s of flow time, solution converges. If we increase the flow time, solution converges more but variation in $C_d$ will be very small after a flow time of 0.2 s.

The calculated value of $C_d$ at 25 degree slant angle and at 40 m/s velocity is 0.2805 which is almost same as achieved by the researcher [4] using both computational fluid dynamics and by experimental approach.

4.2. SUV ANALYSIS WITH AND WITHOUT VG

4.2.1. TURBULENCE VISCOSITY CONTOURS

It is observed in Fig. 4.5, turbulence zone is less at the back of SUV without VG as compared to with VG at different velocities. This turbulence zone is due to ad-
verse pressure gradient at the rear end. The flow, which was laminar at the front end gradually gets transformed into turbulent as it moves through the rear of the vehicle, this develops awake region near the back end of the vehicle. Formation of eddies at the rear end of the vehicle because of this turbulence effects the boundary layer formation. More turbulence in air leads to forward positioning of boundary layer
transitions, but the increased turbulence can also help to keep the flow intact. The minimum and maximum value of turbulence viscosity without VG is 8.41 m²/s and 1260 m²/s and with VG is 826 m²/s and 2160 m²/s, respectively for a velocity of 7.18 m/s. The turbulence viscosity with VG is approximately double as compared to turbulence viscosity without VG. So higher turbulence helps in sticking the flow to the surface for more time and delays boundary layer formation thus results in less value of drag.

### 4.2.2 Pressure Coefficient

Figure 4.6 shows the variation in pressure coefficient ($C_p$) with the $x$ coordinate of the vehicle and compared with the experimental result [1] at velocity equal to 2.42, 3.47, 5.47 and 7.14 m/s, respectively. The simulation is done on different velocity so as to analyze how the pressure coefficient varies with velocity and in $x$ direction of the vehicle.

From Fig. 4.6 it is evident that the value of pressure is maximum at the front.
This occurs because the velocity of air reaches zero when it hits at the front resulting in maximum pressure coefficient. The pressure coefficient gets reduced and flow velocity reaches its zenith at the mid point during the passage of flow over the vehicle. The pressure coefficient starts accelerating as the flow advances from mid-point to rear end. As shown in Fig. 4.6, the pressure coefficient at the mid-point is lowest e.g. at 7.14 m/s velocity, maximum pressure coefficient is -160 and minimum is -20 at the front. At the rear end of the vehicle value of pressure coefficient is 100. Due to loss in pressure energy of the flowing air over the vehicle, the pressure at the rear end does not coincide with the value at the front. This variation in pressure results in drag at the rear end, which is responsible for backward motion of the vehicle.

4.2.3 Comparison of $C_d$ using CFD approach vs. experimental

The drag coefficient value evaluated by numerical approach is prone to errors because of absence of vehicular dimensions in the reference paper [1]. As per the graph there is a slight difference in drag coefficient value. This happens because minor changes in vehicular dimensions leads to major variations in drag coefficient. When the flow passes from mid point to the rear end of the vehicle the pressure coefficient again starts increasing.

![Graph showing experimental vs. simulated results of drag coefficient without VG](image)

Fig. 4.7. (Color online) Experimental vs. CFD results of drag coefficient without VG.

4.2.4 Comparison of $C_d$ with and without VG

Figure 4.8 shows the effect of Vortex Generator on SUV. As observed from the graph, the installation of VG reduced the drag value up to a certain extent at different velocities. If we consider a velocity equal to 7.14 m/s, $C_d$ value is reduced by 10% as given
Reduction of Drag of SUV Similar to TATA Sumo Using Vortex Generator

This decline in value is because of the fact that VG tries to stick the boundary layer to the surface. The vortex is generated at a point close to the downstream edge of the bump, which causes the vortex to interfere with the bump and lose its strength. VG itself creates some drag but overall reduces the value of drag. The effect of VG is estimated that the separation point is shifted to downstream, which in turn narrows the flow separation region. Installation of VG also helps in delay the flow separation time, so the flow remains attached with the surface more time hence resulting in low coefficient of drag as compared to without VG.

5 CONCLUSION

Numerical Investigation of the Ahmed body and SUV similar to TATA Sumo for reduction of drag is presented using Ansys design modeler and Fluent (commercial CFD Package). After verifying the CFD setup and mesh settings, the results produced along with the original SUV model were then utilized to install the vortex generators. From the study it was observed the value of Drag coefficient reduces as the flow velocity accelerates. Setting up of VG at the top of vehicle results in pressure fall at the back end. This fall simultaneously leads to rise in air velocity which successfully helps in delaying the air flow detachment by shifting the separation point further back. This results in drag coefficient reduction VG itself creates drag but overall it reduces the value of drag coefficient. There is a particular value of velocity equal to 7.14 m/s for which drag coefficient is minimum with VG.
REFERENCES


