International Journal of Innovative Technology and Exploring Engineering (IJITEE)
ISSN: 2278-3075, Volume-8, Issue-6S3, April 2019

Aerodynamic Performance Enhancement of a Generic Sedan Model

Abhishek Saha, Sai Kiran Reddy, Srinivas G

Abstract: The design phase of developing an automobile emphasizes on the importance of vehicle aerodynamics which directly affects the overall performance and efficiency. The drag coefficient (Cd) plays a huge role in this aspect and its reduction is crucial in ensuring stability of the vehicle. Our project focuses on the flow analysis over a generic sedan model with an aim to reduce the drag acting on it. The model is created on CATIA and then imported to ANSYS for analysis. It is tested initially as per the prescribed boundary conditions and their contours (pressure and velocity) are studied in order to understand the airflow around the body. A new set of boundary conditions is incorporated following the baseline analysis. The mesh characteristics are also varied to study its effect on drag reduction. This project discusses, in detail, the various experimental, theoretical and numerical processes involved in the computational fluid dynamic study of a sedan model.

Index Terms: aerodynamic drag, drag coefficient, contour, efficiency, pressure, velocity.

I. INTRODUCTION

Nowadays, engineers and designers over the world are focusing on reducing the aerodynamic drag in automobiles to ensure better performance and efficiency at the same time. At higher speeds, aerodynamic drag plays a more significant role. Technical improvements are being carried out every day to meet the objective of designing vehicles with low drag coefficients and reduced lift at high speeds. This article is concerned with the CFD analysis over a sedan model and the various improvements that can be deduced over the course of the research. The analysis shall help to observe and study the flow over the car body. Various trials shall be carried out and the necessary beneficial changes are to be incorporated to the sedan to improve its overall performance and aerodynamics. Better aerodynamics means lower fuel consumption and hence, improved economy. The average modern-day automobile achieves a drag coefficient varying between 0.25 and 0.30. Sedans generally have good drag coefficient values (Cd) because of their ability to further cull the flow of air over the car once it has passed through the roof and thus, create a more efficient release over the trunk space. In addition, sedans are more centered towards the ground and have a perfect weight distribution.

Revised Manuscript Received on December 22, 2018.

Abhishek Saha, Under Graduate student, Dept of Aero & Auto Engg, Manipal Institute of Technology, Manipal Academy of Higher Education (MAHE), Manipal, Karnataka, India.

Sai Kiran Reddy, Under Graduate student, Dept of Aero & Auto Engg, Manipal Institute of Technology, Manipal Academy of Higher Education (MAHE), Manipal, Karnataka, India.

Srinivas G, Assistant Professor (Sr Scale), Dept of Aero & Auto Engg, Manipal Institute of Technology, Manipal Academy of Higher Education (MAHE), Manipal, Karnataka, India.

II. LITERATURE REVIEW

Mass conservation, Newton's second law and the laws of thermodynamics govern the fluid flow over the sedan. Flow domains are defined using either Eulerian or Lagrangian approaches. The finite control volume, an imaginary closed volume within the control region of the flow, forms the basis for the validity of the laws of physics. The continuity equation in the non-conserved form can be represented as follows:

$$D\rho / Dt + \rho\ \nabla . \vec{V}\ = 0 \quad (1)$$

Where ρ represents density, V represents velocity vector field and Dρ/Dt is the change in density with respect to time. Equation (2) represents the continuity equation in the conserved form:

$$\partial \rho / \partial t + \ \nabla \cdot (\rho \vec{V}\ ) = 0 \quad (2)$$

According to the continuity equation, the mass is conserved in the system, which means that the rate at which mass enters the system is equal to the rate at which it leaves. During the process, conservation of energy takes place too as shown in (3).

$$\partial u / \partial t + \ \nabla .\ q = 0 \quad (3)$$

Where u is the local energy density and q is the energy flux, the amount of energy per unit cross sectional area per unit time, as a vector. While setting the appropriate boundary conditions, we opt to keep a stationary wall, hence, ensuring no-slip. In such a case, the following equation holds true.

$$u = v = w = 0 \quad (4)$$

Equation (4) is applicable at the surface to all viscous flows. u, v and w are the velocities in the x, y and z-directions respectively. The Navier-Stokes equation, depicted in (5), comprises of partial differential equations that help to describe the motion of viscous fluid substances. Mass, momentum and energy are all conserved in the process.

$$\rho \frac{Du}{Dt} = -\ \nabla \bar{p} + \mu \nabla^{\ 2} u + \frac{1}{3} \mu \nabla (\ \nabla \cdot u) + \rho g \quad (5)$$

Where ρ is density, g is the acceleration due to gravity, p is pressure, μ is the dynamic viscosity, u is flow velocity, ∇u is the stress tensor and its divergence is equal to ∇^2u.

All these properties form an important basis for deriving the solution.

III. METHODOLOGY

The aerodynamic characteristics are examined using computational fluid dynamics (CFD). The modelling was

Published By:
Blue Eyes Intelligence Engineering & Sciences Publication

done on CATIA- V6. The sedan has dimensions of 4370 mm length, 1700 mm width and 1475 mm height [1]. The model created is a generic one which does not include side mirrors, bumpers or any spoiler. The idea is to study and observe the flow characteristics affected by the shape contours of the vehicle. The bottom-up approach was incorporated, wherein the vertices are initially made and connected to form the edges. The edges join each other to form the faces which assemble to form a volume.

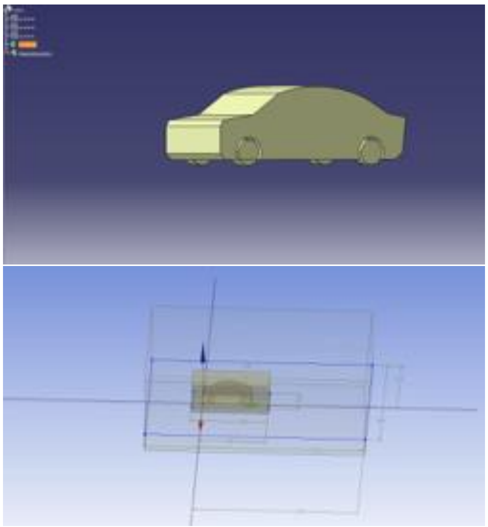

Fig. 1. (a) Sedan car modelled on CATIA; (b) Geometry of the car model

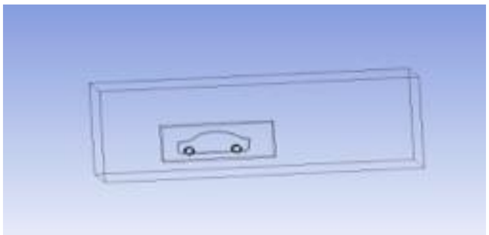

Fig. 2: 3D model of the sedan car within its rectangular domain.

Using ANSYS Meshing, a rectangular shaped mesh is created around the model. The automatic method which includes patch conforming sweep was considered for the process [2]. Meshing plays a crucial role in attaining favorable results. Finer mesh size helps to achieve lower drag coefficient values. Mesh quality depends on three important factors: skewness, smoothness and aspect ratio. It is preferable to maintain a gradual and smooth change in shape for better quality.

Fig. 3: Generated mesh of the vehicle.

Body sizing was used to decide the mesh size. The average skewness of the constructed mesh is around 0.21. Meshes with a skewness varying between 0.20 and 0.25 are said to be of excellent cell quality. An unstructured grid, beneficial for complex geometries is created in which cells are arranged in an arbitrary manner. This grid also allowed changing the resolution over the domain so that the model could be applied over a large domain and the resolution be refined wherever fine scales are important, without affecting the computational time by a large margin. The mesh generated over the sedan model is of tetrahedral form.

IV. AERODYNAMIC ANALYSIS

The flow analysis is carried out on ANSYS Fluent using the pressure-based settings. The standard k-epsilon model is considered for the simulations. An inlet velocity of 25 m/s and outlet gauge pressure of 0 Pa are fixed for the purpose [3]. The fluid material is chosen as air while aluminum as the solid. The energy equation is also turned on. The default position of (0,0,0) along the X, Y and Z axes are taken as the reference frame of the solid. The turbulent intensity is set to 1% and the turbulent viscosity ratio to 10 for the inlet. For the outlet, the corresponding values are set to 5% and 10 respectively [4]. A stationary wall is chosen with a roughness height and constant fixed to a value of 0 and 0.5 respectively. The frontal area of the sedan model is chosen as the reference area. The convection-diffusion equations such as momentum, turbulent kinetic energy and turbulent dissipation rate are chosen for energy. Standard pressure was selected over the other available options. Among the pressure-velocity coupling schemes, SIMPLE was preferred. The momentum value is brought down to 0.3 from the default value of 0.7. Calculations are performed once the solution has been initialized. A convergence criterion of 1e-06 is applied to all residuals, indicating that their values have to drop below that for the solution to converge. Further improvements are carried out testing the model across different velocities, turbulent models and mesh domains.

A. Different Turbulent Models

Apart from the standard k-epsilon model, the model is tested using various other turbulence models such as k-omega, Transition SST (4-equation), laminar (1-equation) and Reynolds shear stress (7-equation). The standard k-omega model, similar to the standard k-epsilon comprises of two equations to solve for two variables, k and omega. Reynolds stress model, on the other hand, involves a complex approach consisting of seven equations to solve the solution and obtain a converged one, thus, increasing computational time, but at the same time producing highly precise results.

B. Different Inlet Velocities

The inlet velocities are also varied and the model is analyzed across velocities of 20, 30, 35, 40 and 45 m/s apart from the earlier prescribed value of 25 m/s. This analysis was conducted for the standard k-epsilon model to maintain uniformity and test the vehicle solely across a range of speeds. A relationship between the drag and inlet velocity could be deduced from this study.

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication

C. Different Far Fields

A rectangular shaped domain was considered during the initial analysis. Later on, the flow analysis is conducted over other types of domains. This helped to monitor the effects that different far fields have on drag reduction. Cylindrical, triangular prism and semi-capsule shaped mesh domains are created around the model. The initial geometry is taken and a cylinder is created along an axis and extruded over the length of the vehicle. A Boolean is created and subtracted from the total surface to produce the desired domain [5]. Similarly, the triangular prism and semi-capsule are also generated. The semi-capsule shaped domain comprises of a cylindrical and rectangular faced inlet and outlet respectively. For the triangular prism, an equilateral triangle is created along any axis and similarly extruded as in the case of the other domains. The analysis is done at an inlet velocity of 25 m/s.

V. RESULTS AND DISCUSSIONS

The initial solution converged at 1436 iterations. All the residuals including continuity, x-velocity, y-velocity, z-velocity, energy, k and epsilon dropped their value below 1e-06, hence, indicating that the solution has been converged as illustrated in Fig. 4.

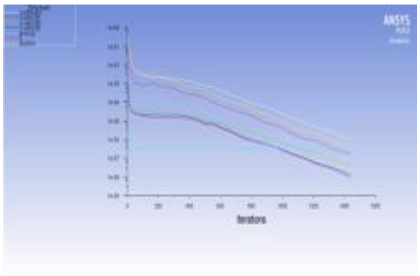

Fig. 4: Convergence plot depicting all the residuals converging at 1e-06.

The drag coefficient obtained is 0.148. The pressure contours are generated over both the plane and the car body. A maximum positive pressure of 390 Pa and a minimum negative pressure of magnitude 712.1 Pa are produced. From Fig. 5, it can be observed that a high-pressure region is created in the front end of the car. The pressure tends to reduce near the streamlined edges of the vehicle. Flow separation and eddy formation near the end of the car cause the pressure to drop further. In addition, a negative pressure region is formed behind the car.

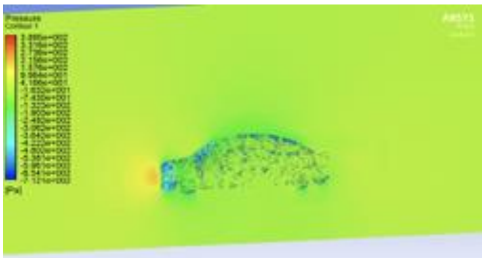

Fig. 5: Pressure contour across the plane

The velocity contour is generated in a similar manner as depicted in Fig. 6. A maximum value of 32.5 m/s is achieved. The velocity reaches its peak value over the top of the car, mainly due to the uniformity in shape and surface. The high-speed air gets separated and does not follow the contour of the vehicle. The low-pressure region towards the rear end due to the flow separation induces a pressure drag on the vehicle. The velocity dips towards the back of the rear end of the vehicle. This is a result of the sudden change in the contour of the vehicle.

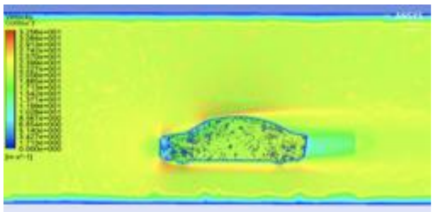

Fig. 6: Velocity contour across the plane.

The variation in drag coefficient across different turbulent models has been depicted in Fig. 7 as a graphical representation. Table I lists the values of drag coefficient, minimum and maximum pressure, minimum and maximum velocity.

Table I. Drag coefficient values obtained for different turbulent models.

Type of Turbulent Model	C_d	Min. P (Pa)	Max. P (Pa)	Min. v (m/s)	Max. v (m/s)
Standard k-epsilon *	0.148	-723.26	400.76	0.32	40.51
Standard k-omega	0.014	-721.61	400.73	0.20	40.55
Transition SST	0.169	-725.36	400.79	0.249	40.65
Laminar	0.437	-723.16	396.01	0.23	40.73
Reynolds Stress	0.04	-725.52	400.98	0.28	40.67

* indicates initial boundary condition considered during baseline analysis.

On comparing the different results obtained, it is deduced that the k-omega model produces the best drag coefficient at the same inlet velocity of 25 m/s. All other models exhibit good aerodynamic characteristics too. The Transition SST model produces a Cd of 0.169. The laminar model attains a value of 0.437, slightly higher than the other models. The Reynolds stress model achieves a low drag coefficient of 0.035.

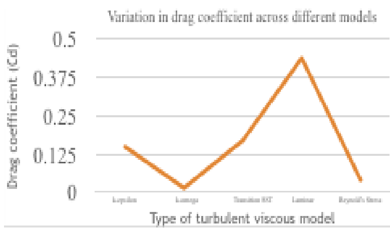

Fig. 7: Graph displaying the variation in C_d evident across different models.

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication

Table II lists the values of drag coefficient and other variables at different inlet velocities. The Cd values showed a linear increment with velocity. Its value increased with a rise in velocity as represented in Fig. 8. From the table, it can be inferred that the magnitude of pressure increases with inlet velocity.

Table II. Drag coefficient values for different inlet velocities.

Inlet Velocity(m/s)	C_d	Min. P (Pa)	Max. P (Pa)	Min. v (m/s)	Max. v (m/s)
20	0.09	-462.57	256.65	0.27	32.4
25*	0.148	-723.26	400.76	0.32	40.51
30	0.22	-1042.03	576.83	0.37	48.3
35	0.30	-1418.89	784.83	0.43	56.75
40	0.39	-1853.8	1024.76	0.48	64.87
45	0.48	-2346.92	1296.62	0.53	73.00

The drag coefficient achieved is found to be directly proportional to the inlet velocity. A drag coefficient of 0.09 is computed at 20 m/s. This proves to be a highly favorable result, indicating an improvement over the earlier baseline analysis.

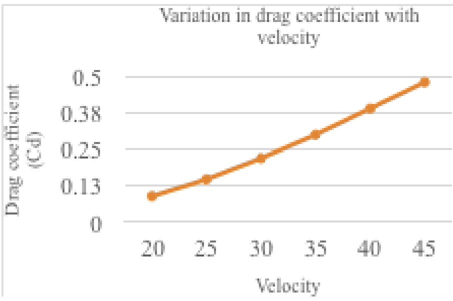

Fig. 8: Variation in drag coefficient with inlet velocity.

Table III depicts the values of the variables obtained when tested across the various far fields. No specific pattern was observed in the variation of drag coefficient. Fig. 9 illustrates a graphical representation of the variation in drag coefficient with different far fields.

Table III. Drag coefficient values obtained for different far fields.

Farfield	C_d	Min. P (Pa)	Max. P (Pa)	Min. v (m/s)	Max. v (m/s)
Rectangle*	0.148	-723.26	400.76	0.32	40.51
Cylinder	0.27	-1199.67	389.95	0.07	43.86
Triangular Prism	0.13	-820.01	388.45	0	36.65
Semi-Capsule	0.66	-3373.30	1929.83	0	71.19

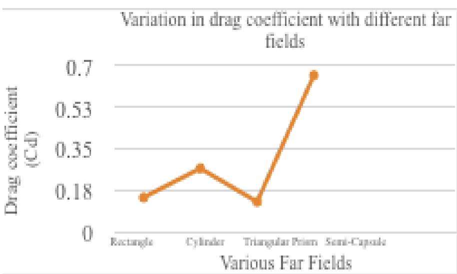

Fig. 9: Variation in C_d across various far fields.

The triangular prism and semi-capsule domains produce a minimum velocity of zero. The semi-capsule domain brings about the highest value of both maximum and minimum pressure. The triangular prism domain achieves a drag coefficient value of 0.13, the lowest among the various domains tested.

VI. CONCLUSION

The aim of the project is to reduce the aerodynamic drag acting on the vehicle. During the course of the research, the obtained results indicate an improvement. Therefore, it could be concluded that a noticeable reduction in drag was obtained upon testing it across a varied set of conditions. The research has further scope. Active aerodynamics, the ability of the vehicle to minimize drag by changing its shape based on the operating conditions, could be implemented to further enhance resultant characteristics. Shape shifting has gained tremendous importance recently. It shall continue to have a profound impact on drag reduction. This research could be further worked upon by using shape shifting spoilers, wings, fins or flaps. Vehicles could be able to change shape, hunkering and elongating for maximum aero-efficiency.

REFERENCES

1. Sneh Hetawal, Mandar Gophane, Ajay B.K., Yagnavalkya Mukkamala, “Aerodynamic Study of Formula SAE Car”.
2. Francesco Mariani, Claudio Poggiani, Francesco Risi, Lorenzo Scappatici, “Formula SAE Racing Car: Experimental and Numerical Analysis of External Aerodynamics”.
3. S.M. Rakibul Hassan, Toukir Islam, Mohammad Ali, Md. Quamrul Islam, “Numerical Study on Aerodynamic Drag Reduction of Racing Cars”.
4. P Ramya, A Hemanth Kumar, Jaswanth Moturi , N.Ramanaiah, “Analysis of Flow over Passenger Cars using Computational Fluid Dynamics”.
5. Mahmoud Khaled, Hicham El Hage, Fabien Harambat, Hassan Peerhossaini, “Some innovative concepts for car drag reduction: A parametric analysis of aerodynamic forces on a simplified body”.