Design and analysis of a new Sedan Car: Analysis from Aerodynamic Perspective

Roopsandeep Bammidi\textsuperscript{a}, B.V.Ramana Murty\textsuperscript{b}

\textsuperscript{a}Department of Mechanical Engineering, RAGHU Engineering College, Dakamarri Village Rd, Dakamarri, Andhra Pradesh 531162, India
\textsuperscript{b}Department of Mechanical Engineering, Gayatri Vidya Parishad College of Engineering (A), Madhurawada, Visakhapatnam-530048, India

(Manuscript Received: 12 May 2016; Revised: 11 Jun 2016; Accepted: 16 Jun, 2016)

Abstract

People demand more efficient road vehicles by the effect of global climate change and increasing the fuel prices. The road vehicles must be obtained with much more performance pertaining within the standard rules. The reduction of drag coefficient is a major target in the industries of automobiles due to its inter-relation with the reduction of fuel consumption. The fuel economy is major concern in the development of automobiles, for the resources of energy conservation and global environment protection. The drag force reduction is a necessary process in road vehicle aerodynamics for the improvement of fuel consumption and also driving performance of the vehicles. In the present scenario, there are plenty of software tool packages that made it easy for the students, engineers and the designers to display the concepts and to describe an idea of imagination in better sense and quality. In this research paper an attempt is made to study the Aerodynamic properties on passenger car model, the Sedan. The attention is paid only to the exterior design and interior is not modeled. The part modeling is done in CATIA V5 R19 which is a backbone tool used for product development processes in industries. Analysis is performed on this model at various speeds such as 60 kmph, 80 kmph, 130kmph and 165kmph. The study is focused on the coefficients and forces in case of lift, side and drag, velocity streamlines and turbulent flows. The ANSYS CFX 15.0 which is a commercial CFD tool used for the analysis. The main source of air resistance of this model is drag which is to reduce it for better fuel consumption. The Numerical calculations have been carried. This work gives the brief idea about aerodynamic properties of Passenger car which seems to be simple automobile design. The research work fetched in geometrical corrections and improvement of aerodynamics to develop a new model which might achieve a better fuel economy.

Keywords: CFD; Passenger Car; Sedan; Fuel Consumption; ANSYS; CATIA

1. Introduction

Fuel consumption of automobile is related to its aerodynamic drag which is highly influenced by the separation flows around its shape. The human activities became major cause in increasing the greenhouse effect of gases and also the average global temperature. The activity that includes transportation sector where number of automobile is rapidly increasing and also increases the fuel consumption. It tends to create harmful effects on environment by increasing air pollution. Because of these problems, the automobile industries try to minimize the greenhouse effect by using efficient fuel consumption. The automotive industry uses the CFD package which is a commercial tool. The CFD is not only used for the improvement of vehicle aerodynamics, but also for domain optimization like brake cooling, lighting, engine cooling, fuel system and airbags. While the product development of new road vehicles, leads to understanding the phenomenon of flow behavior and also the aerodynamic forces that are influenced by the changes in aesthetic shape of the vehicle body. The CFD is best way for the designers to obtain the results in a short interval of time. The object of aerodynamic research is to minimize the resistance of the aerodynamics to allow the faster running for the same energy consumption or the lower consumption for the same speed. The performance of vehicle aerodynamics is mainly determined by the co-efficient of the drag which directly effects the fuel consumption and engine requirements.

In the industries of motor vehicles, aerodynamics plays a very important role and the car manufacturers around the world had been fascinated and influenced by the various improvements in the aerodynamics. There had been a constant effect to incorporate changes in road vehicles, not just as an aesthetic design feature but also contribute to improve vehicle handling and the fuel economy. Aerodynamics is the study related to flow of air particles around a body such as airplanes, cars and motorcycles at lower or higher speeds. The forces of aerodynamics will be generated if there is a flow of air over the body. The forces of aerodynamics include pressure distributions and shear stress distributions over the body. With respect to pressure distribution over body, drag force and lift force are occurring. Theory of Bernoulli says that the flow of high speeds on the area makes low pressure on that area and flow of low speeds on the area makes the area have high pressure.

Air flow over the Ahmed body is investigated by means of transient RANS turbulence models [1]. The performance of several RANS turbulence models has been compared. It has been found that Durbins k-\varepsilon model is more accurate than the other turbulence models for the wall- bounded cases with separation and reattachment.

Tastan [2] studied reliability and the turbulence model performances by CFD software to determine the passenger car aerodynamics that are tested and compared. In this analysis,
the drag forces that are acting on the car, pressure and velocity distributions and wake flow patterns which are determined by using several turbulence models with a commercial tool called software Fluent. The results are compared to the experimental values and observed that, the turbulence models gave relatively reliable results. Among the models of turbulences, standard $k-\omega$ models and RNG $k-\varepsilon$ models stand one step ahead of the other models in accordance with the results. Damjanović et al. [3] designed a car by using the software tool Autodesk 3ds Max 2010 which represents a new solution of conceptual design and Initial design that is free form sketches of an imaginary car. A tool Mental Ray is used for digital image generation. The attention is only given to the exterior design of car and FLUENT software is used for the side contours of the road vehicles. The possible correction of geometry of the car is made in the purpose of improving the design in the terms of air resistance reduction and aerodynamics improvement.

Morden [4] studied the lift forces and investigated in accuracy of the computational fluid dynamics during the external flow over the race vehicles. In this work validation of full scale results of wind tunnel experimentation with the data is collected from the track testing.

Katz [5] reviewed the significance of aerodynamic negative lift and the way how it improves the performance of the race car. The various methods for the generation of down force such as vortex generators, diffusers and inverted wings are discussed. The vehicles complex geometry made the interactions of aerodynamics between the components of entire body which is significant that result in the vortex flows and surface shapes lifting and also unlike the traditional airplane wings. Also discussed the typical design tools like as computational fluid dynamics, track testing and wind tunnel testing and their relevance to the development of the race car. Several examples covers the wide range of the vehicle shapes from stock cars to the open-wheel race cars and are presented for the demonstration of fluid flow nonlinear nature.

Mohamed et al. [6] investigated road vehicle aerodynamics and study the drag, lift coefficient and the pressure distributions on the surface. PROTON Saga CAD model of small sedan car is produced by using the scanned data and the surface refinement with CATIA. External flow analysis is done with the speed limit of 120 kmph. The lift and drag coefficients value are averaged and compared. The efficiency of the body shape is and coefficient of pressure is analyzed to observe the relation between the drag and lift. The overall velocity distributions are also focused to visualize the fluid flow.

2. CATIA Modeling

CATIA V5R19 takes the real time rendering of one step that is further to allow the users and also to experience product as if it was the real, to provide the real time testing the material configuration and to leverage perceived quality of final product. The workflow solution of unified industrial design is to imagine, creating, to share and to experience. This modeling tool is easy, fast, robust, user oriented and user friendly and in the automotive industry, it is one of the leading CAD-systems which ensure a smooth transition from concept phase to production phase. It is powerful tool and every manufacturer has its own CAE tool landscape.

The geometrical dimensions of the model used to develop part modeling in CATIA V5 R19 is tabulated in Table 1. The rough sketch of this model in CATIA tool is shown in Fig. 1. Also, the sketch developed into part modeling in software tool is presented in Fig. 2.

3. CFD Analysis

The Analysis is carried out at various speeds such as 60kmph, 80kmph, 130kmph and 165kmph in Ansys-CFX and the results such as contours, vector plots, and turbulent kinetic energy and streamlines plots. The surface pressure contour is also observed in the analysis. The analysis includes process of five steps such as:

1) Geometry
2) Mesh
3) Setup
4) Solution
5) Results

Geometry: The part modeling of CATIA file is saved in .stp format and then imported to CFX

Mesh and Setup: The imported file geometry undergoes meshing after that the physics is defined to the external domain. The fine mesh is considered for good results (Fig. 3).
Figure 1. Geometrical Representation of Hatch-Back Car in CATIA V5 R19

Figure 2. Part Modeling of Hatch-Back Car in CATIA V5 R19

Figure 3. Meshed Part of Sedan Car
Solution and Results: After applying boundary conditions, the solution and results are as follows.

Analysis at 60kmph: For performing analysis on the body, the boundary conditions are shown in Table 2.

Table 2. Boundary Conditions of Sedan Car at 60kmph

<table>
<thead>
<tr>
<th>Input Velocity</th>
<th>60kmph</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mesh Nodes</td>
<td>181878</td>
</tr>
<tr>
<td>Mesh Elements</td>
<td>982601</td>
</tr>
<tr>
<td>Domain Type</td>
<td>Fluid</td>
</tr>
<tr>
<td>Domain Motion</td>
<td>Stationary</td>
</tr>
<tr>
<td>Heat Transfer Model</td>
<td>Isothermal</td>
</tr>
<tr>
<td>Fluid Temperature</td>
<td>2.5000e+01 [°C]</td>
</tr>
<tr>
<td>Turbulence Model</td>
<td>k epsilon</td>
</tr>
<tr>
<td>Turbulent Wall Functions</td>
<td>Scalable</td>
</tr>
<tr>
<td>Boundary Type</td>
<td>Inlet</td>
</tr>
<tr>
<td>Flow Regime</td>
<td>Subsonic</td>
</tr>
<tr>
<td>Mass and Momentum (Inlet)</td>
<td>Normal Speed</td>
</tr>
<tr>
<td>(outlet) (Wall Boundary)</td>
<td>Static Pressure</td>
</tr>
<tr>
<td>Relative Pressure</td>
<td>0.0000e+00 [Pa]</td>
</tr>
<tr>
<td>Wall Roughness</td>
<td>Smooth Wall</td>
</tr>
</tbody>
</table>

![Figure 4. Pressure Contour of Sedan Car at 60kmph](image)

![Figure 5. Velocity Contour of Sedan Car at 60kmph](image)

![Figure 6. Turbulent Kinetic Energy Contour of Sedan Car at 60kmph](image)

![Figure 7. Pressure Vector of Sedan Car at 60kmph](image)

![Figure 8. Velocity Vector of Sedan Car at 60kmph](image)
After CFD analysis, the obtained pressure contour, velocity contour, turbulent kinetic energy contour, pressure vector plot, velocity vector plot, turbulent kinetic energy vector plot, velocity stream lines and pressure distribution around the Sedan Car are presented.

**Analysis at 80kmph:** For performing analysis on the body, the boundary conditions are shown in Table 3.

<table>
<thead>
<tr>
<th>Input Velocity</th>
<th>80kmph</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mesh Nodes</td>
<td>181873</td>
</tr>
<tr>
<td>Mesh Elements</td>
<td>982578</td>
</tr>
<tr>
<td>Domain Type</td>
<td>Fluid</td>
</tr>
<tr>
<td>Domain Motion</td>
<td>Stationary</td>
</tr>
<tr>
<td>Heat Transfer Model</td>
<td>Isothermal</td>
</tr>
<tr>
<td>Fluid Temperature</td>
<td>2.5000e+01 [C]</td>
</tr>
<tr>
<td>Turbulence Model</td>
<td>k epsilon</td>
</tr>
<tr>
<td>Turbulent Wall Functions</td>
<td>Scalable</td>
</tr>
<tr>
<td>Boundary Type</td>
<td>Inlet</td>
</tr>
<tr>
<td>Flow Regime</td>
<td>Subsonic</td>
</tr>
<tr>
<td>Mass and Momentum (Inlet)</td>
<td>Normal Speed</td>
</tr>
<tr>
<td>(outlet) (Wall Boundary)</td>
<td>No Slip Wall</td>
</tr>
<tr>
<td>Relative Pressure</td>
<td>0.0000e+00 [Pa]</td>
</tr>
<tr>
<td>Wall Roughness</td>
<td>Smooth Wall</td>
</tr>
</tbody>
</table>

After CFD analysis, the obtained pressure contour, velocity contour, turbulent kinetic energy contour, pressure vector plot, velocity vector plot, turbulent kinetic energy vector plot, velocity stream lines and pressure distribution around the Sedan Car are presented.
Analysis at 130kmph: For performing analysis on the body, the boundary conditions are shown in Table 4.
Table 4. Boundary Conditions of Sedan Car at 130kmph

<table>
<thead>
<tr>
<th>Input Velocity</th>
<th>130kmph</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mesh Nodes</td>
<td>181873</td>
</tr>
<tr>
<td>Mesh Elements</td>
<td>982578</td>
</tr>
<tr>
<td>Domain Type</td>
<td>Fluid</td>
</tr>
<tr>
<td>Domain Motion</td>
<td>Stationary</td>
</tr>
<tr>
<td>Heat Transfer Model</td>
<td>Isothermal</td>
</tr>
<tr>
<td>Fluid Temperature</td>
<td>2.5000e+01 [°C]</td>
</tr>
<tr>
<td>Turbulence Model</td>
<td>k epsilon</td>
</tr>
<tr>
<td>Turbulent Wall Functions</td>
<td>Scalable</td>
</tr>
<tr>
<td>Boundary Type</td>
<td>Inlet</td>
</tr>
<tr>
<td>Flow Regime</td>
<td>Subsonic</td>
</tr>
<tr>
<td>Mass and Momentum (Inlet)</td>
<td>Normal Speed</td>
</tr>
<tr>
<td>(outlet) (Wall Boundary)</td>
<td>Static Pressure</td>
</tr>
<tr>
<td></td>
<td>No Slip Wall</td>
</tr>
<tr>
<td>Relative Pressure</td>
<td>0.0000e+00 [Pa]</td>
</tr>
<tr>
<td>Wall Roughness</td>
<td>Smooth Wall</td>
</tr>
</tbody>
</table>

After CFD analysis, the obtained pressure contour, velocity contour, turbulent kinetic energy contour, pressure vector plot, velocity vector plot, turbulent kinetic energy vector plot, velocity stream lines and pressure distribution around the Sedan Car are presented.

Figure 20. Pressure Contour of Sedan Car at 130kmph

Figure 21. Velocity Contour of Sedan Car at 130kmph

Figure 22. Turbulent Kinetic Energy Contour of Sedan Car at 130kmph

Figure 23. Pressure Vector of Sedan Car at 130kmph

Figure 24. Velocity Vector of Sedan Car at 130kmph
3. Analytical calculations of various forces on New Sedan Car

By giving these equations in the CFX simulations, got the coefficient values and there by performing reverse interpolation, forces are calculated.

Drag Force, \(D = C_d \left( \frac{1}{2} \right) \rho AV^2\)

Here, \(C_d\) is Drag Coefficient
\(\rho\) is Density of Air, 1.225 kg/m\(^3\)
A is Frontal Area, 0.02053305 m\(^3\)
V is Flow Velocity, kmph

At 60kmph, \(D = C_d \left( \frac{1}{2} \right) \rho AV^2\)
\[= 0.581877 \times \frac{1}{2} \times 1.225 \times 0.02053305 \times 60^2\]
\[= 26.34469952\text{ N.}\]

At 80kmph, \(D = C_d \left( \frac{1}{2} \right) \rho AV^2\)
\[= 0.475093 \times \frac{1}{2} \times 1.225 \times 0.02053305 \times 80^2\]
\[= 38.24002463\text{ N.}\]

At 130kmph, \(D = C_d \left( \frac{1}{2} \right) \rho AV^2\)
\[= 0.464014 \times \frac{1}{2} \times 1.225 \times 0.02053305 \times 130^2\]
\[= 98.62280409\text{ N.}\]

At 165kmph, \(D = C_d \left( \frac{1}{2} \right) \rho AV^2 = 0.459163 \times \frac{1}{2} \times 1.225 \times 0.02053305 \times (165)^2\]
\[= 157.215127\text{ N.}\]

Lift Force, \(L = C_l \left( \frac{1}{2} \right) \rho AV^2\)
Here, \(C_l\) is Lift Coefficient
\(\rho\) is Density of Air, 1.225 kg/m\(^3\)
A is Frontal Area, 0.02053305 m\(^3\)
V is Flow Velocity, kmph

At 60kmph, \(L = C_l \left( \frac{1}{2} \right) \rho AV^2\)
\[= 1.20191 \times \frac{1}{2} \times 1.225 \times 0.02053305 \times 60^2\]
\[= 54.41692627\text{ N.}\]

At 80kmph, \(L = C_l \left( \frac{1}{2} \right) \rho AV^2\)
\[= 1.18047 \times \frac{1}{2} \times 1.225 \times 0.02053305 \times 80^2\]
\[= 95.01550617\text{ N.}\]

At 130kmph, \(L = C_l \left( \frac{1}{2} \right) \rho AV^2\)
\[= 1.25177 \times \frac{1}{2} \times 1.225 \times 0.02053305 \times 130^2\]
\[= 266.0546179\text{ N.}\]

At 165kmph, \(L = C_l \left( \frac{1}{2} \right) \rho AV^2\)
\[= 1.2623 \times \frac{1}{2} \times 1.225 \times 0.02053305 \times 165^2\]
\[= 432.2052405\text{ N.}\]

Side Force, \(S = C_s \left( \frac{1}{2} \right) \rho AV^2\)
Here, \(C_s\) is Side Coefficient
\(\rho\) is Density of Air, 1.225 kg/m\(^3\)
A is Frontal Area, 0.02053305 m\(^3\)
V is Flow Velocity, kmph

At 60kmph, \(S = C_s \left( \frac{1}{2} \right) \rho AV^2\)
\[= 0.0461893 \times \frac{1}{2} \times 1.225 \times 0.02053305 \times 60^2\]
\[= 4.18247578\text{ N.}\]

At 80kmph, \(S = C_s \left( \frac{1}{2} \right) \rho AV^2\)
\[ S_f = C_s \left( \frac{1}{2} \right) \rho AV^2 \]

At 130 kmph, 
\[ S_f = 7.2597 \times 10^{-5} \cdot \frac{1}{2} \cdot 1.225 \cdot 0.02053305 \cdot 130^2 \]
\[ = 0.015430007 \text{ N}. \]

At 165 kmph, 
\[ S_f = 7.1322 \times 10^{-5} \cdot \frac{1}{2} \cdot 1.225 \cdot 0.02053305 \cdot 165^2 \]
\[ = 0.0244204 \text{ N}. \]

4. Results and Discussion

The exterior profile design of different automobile shapes are developed and analysis is carried out to find different characteristics of aerodynamics such as drag coefficients, lift coefficients, side force coefficients, drag forces, lift forces, side forces, contours and vectors, turbulent kinetic energy, velocity streamline flows at various speed limits such as...
60kmph, 80kmph, 130kmph and 165kmph. The results that observed in different cases are as follows (Table 5).

<table>
<thead>
<tr>
<th>Speed</th>
<th>Drag Coefficient, $C_d$</th>
<th>Lift Coefficient, $C_l$</th>
<th>Side Force Coefficient, $C_s$</th>
<th>Drag Force, $D$, N</th>
<th>Lift Force, $L$, N</th>
<th>Side Force, $C_s$, N</th>
</tr>
</thead>
<tbody>
<tr>
<td>60kmp</td>
<td>0.581</td>
<td>1.201</td>
<td>0.046</td>
<td>26.344</td>
<td>54.416</td>
<td>4.182</td>
</tr>
<tr>
<td>80kmp</td>
<td>0.475</td>
<td>1.18</td>
<td>7.99e-05</td>
<td>38.24</td>
<td>95.015</td>
<td>0.004</td>
</tr>
<tr>
<td>130kmp</td>
<td>0.46</td>
<td>1.25</td>
<td>5.7e-00</td>
<td>50.26</td>
<td>226.205</td>
<td>0.01</td>
</tr>
<tr>
<td>165kmp</td>
<td>0.45</td>
<td>1.26</td>
<td>7.13e-05</td>
<td>157.215</td>
<td>432.205</td>
<td>0.024</td>
</tr>
</tbody>
</table>

From the table the Sedan Car shows that the drag coefficient decreases with increase in speed and the average value of 0.49 and is well agreement with the standard limit of 0.1 to 0.5. The drag coefficient at various speed of different models are compared and presented.

**Lift Coefficient:** The lift coefficient at various speeds is shown below. As the lift coefficient increases stability and safety will be decreased.

**Side Force coefficient:** The side force coefficient is almost negligible.

**Drag Force:** Minimum drag force is observed for Sedan car and required to overcome the wind resistance. Hence the Sedan car design is aerodynamically efficient and gives minimum fuel consumptions.
Lift Force: The work in the paper is the effect of aerodynamics on fuel economy that comes through drag force only. However drag is not only the aerodynamic effect, of other forces and moments generated aerodynamically one is interested in lift force. The lift coefficient of today’s road vehicle is ranging between 0.0 to 0.7 and or more based on the frontal area. The lift forces at various speeds are presented.

Side Force: Sedan car almost requires negligible side forces.

Wake Formation: From the analysis it has been observed that the wake formation is less which leads to less dust formation on rear end of the car from velocity flow patterns.

5. Conclusion

Aerodynamics plays a vital role in fuel economy, stability and emissions of any road vehicles. The road vehicle aerodynamics is studied and inspired to design a new conceptual car which initiated to design only exterior profile of the vehicle. In this research work, the profile design of new Sedan car is modeled in CATIA V5 R19 and analysis is carried out in ANSYS-CFX 15.0.

* The main objective of this research work concentrated only on the drag reduction.
* The Sedan car is aesthetic and fuel efficient.
* The drag coefficient 0.45 with consideration of only exterior profile is equal to or less than 0.3 drag coefficients for entire car with considerations of interior design, required accessories and also including the add-on devices.
  * The wake formation in sedan car is less.
  * The dynamic pressure distribution is more at the wind shield.
  * The drag force of sedan car is 157.215 N at 165kmph is less force that required in overcoming the wind resistance.
  * Computational analysis of this new exterior profile design of sedan car experienced the aerodynamic study at different high way speeds.
  * A significant change occurred in the drag coefficient, lift coefficient and side force coefficient of the model when streamline shape is adopted.
  * The reduction of wind shield angle and more streamline body of sedan car made it as a fuel efficient car.

6. Future Scope

The aesthetic design of aerodynamics of road vehicles is in the concern of reduction in case of drag which is an area and where a lot of modifications and improvements that will appear in near future. The aerodynamics of road vehicles that had been described in a way to achieve motions about reduction of drag resistance, and even theoretically ideal techniques had been recommended. However, designing an automobile had to deal with a lot of styling issues, performances and functionality.

The future work recommendations for this research work are as follows:

By adding spoiler to the Sedan car, the drag 0.45 is reduced and then it can be a fuel efficient car and cross wind studies is to be applied for all the models and also unsteady analysis is to be investigated for good understanding of wake flow properties.

By adding the accessories and add-on devices, the computational analysis is to be done for getting good drag coefficient. Pitching moment, yawing moment and rolling moments is to be investigated for better stability of the vehicle. Wind Tunnel Experimentation is to be done for all these models and then to be compared.

And finally if wheels, mirrors, doors and head-lights are added to my sedan car it might look to be as a concept car that is as follows.

References


[2] Umurtastan, “Investigation of turbulence models used in automotive industry”. A thesis submitted to the graduate...
school of natural and applied sciences of Middle East technical university


[10] www.cfd-online.com