Journal of Advanced Engineering Research
ISSN: 2393-8447
Volume 6, Issue 2, 2019, pp.59-63

CFD Analysis on Various Commercial Vehicles to Evaluate the Aerodynamic Characteristics - A Comparative Study

J.V. Muruga Lal Jeyan, Krishna S Nair*, Godwin Vincent

School of Mechanical Engineering, Lovely Professional University, Punjab, India

*Corresponding author email: krishnanair98s@gmail.com

ABSTRACT

Performance, handling, safety and comfort of an automobile vehicle are significantly affected by its aerodynamics. High brake power of the engine is not enough to evaluate the performance of vehicle. Aerodynamics plays a significant role in vehicle design and development especially while designing the stability aspects after certain speed. Drag forces has significant impact on efficient drive of the vehicle. In fact, the fuel economy is majorly affected by the vehicle design. In the present study, flow analysis has been done on various commercial cars. The prime objective of an automobile company is to improve the fuel efficiency by minimising the drag. Different scaled models of the different cars have been selected which includes Audi Q7, Jeep Compass, Toyota Altis and Volkswagen Polo. A CFD analysis has been performed on all these models using ANSYS FLUENT. Pressure and velocity distribution have been computed for three different sir velocities 30m/s, 50m/s and 75m/s. Stagnation points have been identified along with the velocity distribution on the windshields. Some of the preventive measures have been identified while designing such vehicles. A comparative analysis has been done for the different vehicles and various conclusions have been framed.

Keywords: AZ91C Aerodynamic Efficiency, ANSYS, Automobile, Fuel Economy, Aerodynamics.

1. INTRODUCTION

The aerodynamic analysis of automotive models has helped the automotive industry significantly to reduce the aerodynamic drag. The strength of the resistance is determined by the flow of air over a vehicle, which has an impact on the performance and efficiency of the vehicle. The dragging force acting on the machine depends on various parameters, such as the drag coefficient, the projected frontal area, the speed at which the machine is navigating and the atmospheric conditions in which the machine operates [1-5]. The efficiency of a car can be increased with the reduction in air resistance as the energy lost due to friction is less. Therefore, aerodynamic design of vehicle helps in reducing the drag coefficient, thus increasing the fuel efficiency [6-9]. The main aim of automotive development is to save energy and protect global environment by reducing consumption of fuel. Drag reduction plays an essential role in fuel consumption and driving performance. An aerodynamically refined body adds to increased vehicle appeal. Therefore it is important that the vehicle possesses a body shape that is aerodynamic, not the ideally aerodynamic shape of a fish or of birds. A separation of the flow in the back is unavoidable with such a body shape. Rolling resistance dominates aerodynamic drag at lower speeds. The increase in resistance due to the increase in the speed square causes the aerodynamic drag to exceed three quarters of the total engine power when operating in higher speeds. Therefore, aerodynamic drag is overcome with the use of maximum power generated by the engine. This increases the load on engine which causes a further increase in rate of fuel consumption [10-12]. In addition, the use of CFD tools and high-end configuration computers has helped automakers in reducing their investment costs for automobiles and also take full advantage of the modeling and analysis tool in making the necessary changes to the vehicle geometry. Furthermore, the problems of interaction of the fluid structure influence each other. The force of the fluid deforms the structure and the displacement of the structure is followed by the fluid. This interaction does not only mean the velocity of the fluid equalling the structure at the interface, but also the fluid's domain change as a result of the structure’s movement. Now, numerical resolution of these complex problems is allowed by the latest technologies and super computers.

Therefore, in the present study, different scaled models of the different cars have been selected which includes Audi Q7, Jeep Compass, Toyota Altis and Volkswagen Polo. A

CFD analysis has been performed on all these models using ANSYS FLUENT. Pressure and velocity distribution have been computed for three different sir velocities 30m/s, 50m/s and 75m/s. Stagnation points have been identified along with the velocity distribution on the windshields. Some of the preventive measures have been identified while designing such vehicles.

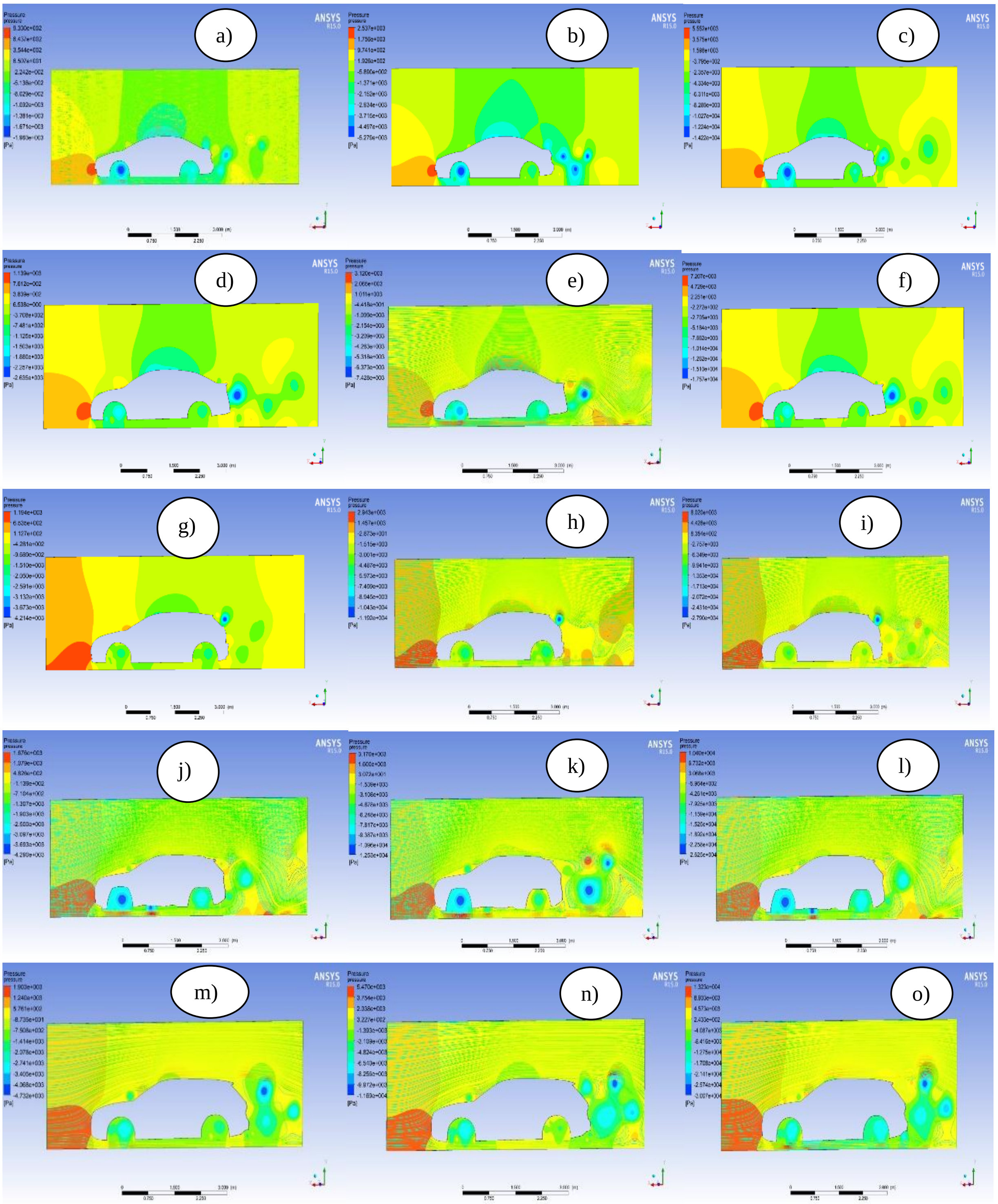

Fig. 1Pressure distribution over five different car designs (a-20m/s, b-50m/s, c-75m/s) Toyota Altis, (d-20m/s, e-50m/s, f-75m/s) Nissan Sunny, (g-20m/s, h-50m/s, i-75m/s) Volkswagen Polo, (j-20m/s, k-50m/s, l-75m/s) Jeep Compass and (m-20m/s, n-50m/s, o-75m/s) Audi Q-7.

2. METHODOLOGY

The resistive force that resists a solid body to move forward through a fluid is known as aerodynamic drag. There are two components for drag force, pressure and friction. The aerodynamic drag of any form is standardized by a dimensionless number called a resistance coefficient. Any design can only be acceptable if its form drag is reduced.

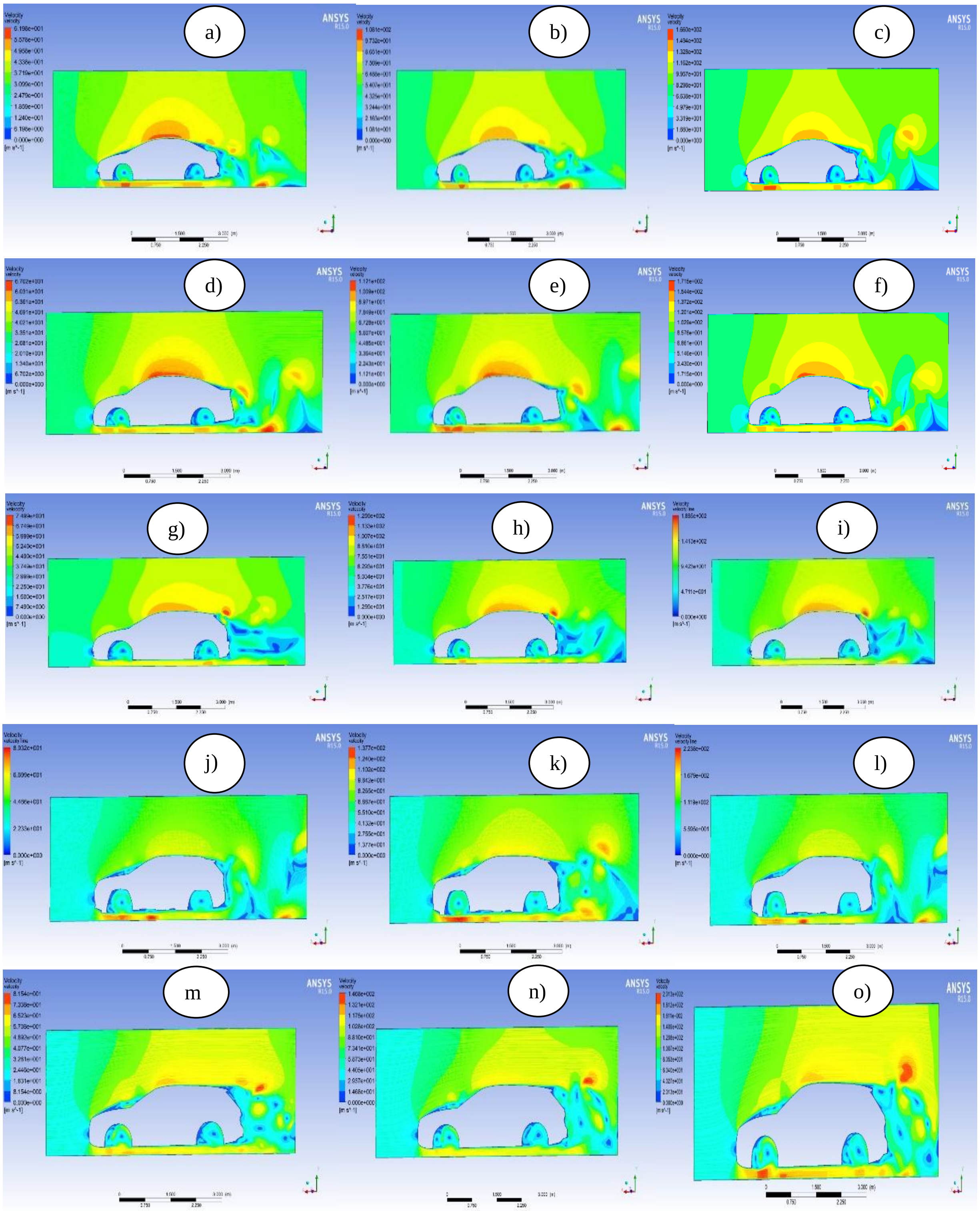

Fig. 2Velocity distribution over five different car designs (a-20m/s, b-50m/s, c-75m/s) Toyota Altis, (d-20m/s, e-50m/s, f-75m/s) Nissan Sunny, (g-20m/s, h-50m/s, i-75m/s) Volkswagen Polo, (j-20m/s, k-50m/s, l-75m/s) Jeep Compass and (m-20m/s, n-50m/s, o-75m/s) Audi Q-7.

The main objective of automotive aerodynamics study is to lessen the aerodynamic drag by governing and studying the exterior flow and thereby increasing the fuel efficiency. For this, 6 scaled models of the different cars are taken which includes Sedans, SUVs and Hatchbacks. These models are tested in the wind tunnel to study the flow properties. This is followed by finding coefficient of drag and the pressure distribution around the samples. Based on results, vehicles are upgraded and comparison is made. The scaled models are placed in the wind tunnel to conduct several tests. Using Ansys Fluent for SUV, sedan and hatchback car models at 3 different velocities 30m/s, 50m/s and 75m/s.

Table 1 Dimensions of the Vehicles

Models	Length (mm)	Width (mm)	Height (mm)	Ground Clearance (mm)
Toyota Corolla Altis	4620	1775	1475	175
Nissan Sunny	4495	1695	1515	161
Volkswagen Polo	3971	1682	1469	165
Jeep Compass	4395	1818	1640	178
Audi Q7	5052	1968	1740	240

Based on results, vehicles comparison is made. The main output of this research is to make easy way to compare the flow parameters of Sedans, Hatchbacks and SUVs, thus helping the manufacturers to modify the upcoming models and increase the fuel economy of the car by developing the aerodynamics of the frontal area. The results obtained from the experimental research allow specifying complications and reviews the improvement of vehicle aerodynamics based on the development of scale models.

3. RESULTS AND DISCUSSIONS

The car models were modeled in SolidWorks considering only its exterior geometry. Then, the models were brought to the ANSYS workbench as a 2D sketch to perform the exterior flow analysis on to the model. The exterior flow for the car was made assuming that the car moves at a speed of 30m/s, 50m/d and 75m/s. From the analysis the aerodynamic characteristics has been studied in vibrant.

Fig. 1 shows the pressure distribution over five different car designs including two Sedan (Toyota-Altis and Nissan Sunny), 2 Hatch Back (Volkswagon-Polo and Jeep) and one SUV (Audi-Q7). The models were tested for three velocity inputs 30m/s, 50m/s and 75m/s. It has been noticed after the analysis that the maximum pressure is acting on frontal area and stagnation point is located at the front bumper for all five designs. It has been observed that the magnitude of the pressure is found to high for higher velocities and the same trend has been observed for all designs. It can also be observed that the pressure drop at the upper part of the models is found to be larger for Toyota Altis than other designs. It is quite obvious as this model consists aerodynamic air flow over the body. The separation of the flow leaves a negative pressure behind the car model.

Fig. 2 shows the velocity distribution over five different car designs including two Sedan (Toyota-Altis and Nissan Sunny), 2 Hatch Back (Volkswagen-Polo and Jeep) and one SUV (Audi-Q7). Models are being tested using ANSYS Fluent for three different velocities 30m/s, 50m/s and 75m/s. It can be seen from the Figure 2 that the velocity is higher at the top of the vehicle and it is quite zero at the front bumper. Reverse flow can also be observed at the rear part of the vehicle which is quite obvious as flow separation leads to negative pressure which results in back flow.

4. CONCLUSION

The performance of the car changes due to the aerodynamics. Relative study has been performed on five models by carrying out CFD simulations using ANSYS Fluent. Computational analysis of the Sedan, SUV and Hatchback models suggests that the sedan models have less resistance over the full range of speed and pressure than the Hatch Back and SUV models. Therefore, it can be resolved that the sedan car model is more agile and aerodynamically effective than the Hatch Back and SUV vehicle models. For the Sedan, SUV and Hatchback models, the strength of resistance increases with increasing speed, while the resistance coefficient decreases slightly as the Reynolds number increases.

REFERENCES

[1] J. Howell, C. Sherwin, and G. Le Good, The aerodynamic drag of a compact SUV as measured

on-road and in the wind tunnel Aerodynamic Drag of a Compact SUV as Measured On-Road and in the Wind Tunnel, *SAE Technical Paper, 2002-01-0529,* 2002.

[2] K.S. Song, S.O. Kang, S.O. Jun, H.I. Park, J.D. Kee, K.H. Kim and D.H. Lee, Aerodynamic design optimization of rear body shapes of a sedan for drag reduction, *International Journal of Automotive Technology,* 13, 2012, 905–914.

[3] M. Gophane, G. Salvi, G. P. Pradeep and K. Ravi, Effect of Aerodynamic Forces over the Bus Body and Design of Conceptual Bus for Enhanced Performance, *International Journal of Engineering Trends and Technology,* 11 (4), 2014, 159–162.

[4] B. Prajwal and D. Unune, Modelling, simulation and validation of results with different car models using wind tunnel and Star-CCM+, *Journal of the Serbian Society for Computational Mechanics,* 9 (1), 2015, 46–56.

[5] A. G. Model, Analysis of External Aerodynamics of Sedan and Hatch Back Car Models Having Same Frontal Area by Computational Method, *International Journal of Research in Mechanical Engineering & Technology,* 6 (1), 2016, 55-61.

[6] G. Dias, N.R. Tiwari, J.J. Varghese and G. Koyeerath, Aerodynamic Analysis of a Car for Reducing Drag Force, *IOSR Journal of Mechanical and Civil Engineering,* 13 (3), 2016, 114–118.

[7] G. Siva and V. Loganathan, Design and Aerodynamic Analysis of a Car to Improve Performance, *Recent Innovations in Engineering, Technology, Management & Applications*, 24, 2016, 133–140.

[8] S.S. Shinde and P.M.D. Shende, Enhancement of Aerodynamic Drag Reduction of Passenger Vehicle using CFD analysis - Review, *International Journal of Innovative Research in Science, Engineering and Technology*, 6 (1), 2017, 437–448.

[9] V. Yakkundi, S. Mantha and V. Sunnapwar, Hatchback Versus Sedan – A Review of Drag Issues, *SSRG International Journal of Mechanical Engineering*, 4 (1) 2017, 5-13.

[10] S. Kant, D.D. Srivastava, R. Shanker, R.K. Singh, and A. Sachan, A Review on CFD Analysis of Drag Reduction of a Generic Sedan and Hatchback, *International Research Journal of Engineering and Technology*, 4 (5), 2017, 973–982.

[11] A. Anish, P.G. Suthen and M.K. Viju, Modelling And Analysis Of a Car For Reducing Aerodynamic Forces, *International Journal of Engineering Trends and Technology*, 47 (1), 2017, 1–16.

[12] B. Zala, P.P. Rathod and S.S. Arvind, Aerodynamic performance assessment of sedan and hatchback car by experimental method and simulation by computational fluid dynamics - A review, *Journal of Engineering Research and Studies*, 3 (1), 2012, 91-95.

[13]B. Deepanraj, P. Lawrence and G. Sankaranarayanan, Theoretical analysis of gas turbine blade by finite element method, *Scientific World*, 9 (9), 2011, 29-33.

[14] J.V. Muruga Lal Jeyan and M. Senthil Kumar, Performance Evaluation of Yaw Meter With the Aid of Computational Fluid Dynamic, *International Review of Mechanical Engineering*, 8 (2), 2014, 445-459.