Annals of R.S.C.B., ISSN:1583-6258, Vol. 25, Issue 6, 2021, Pages. 4501 - 4507
Received 25 April 2021; Accepted 08 May 2021.

Drag Reduction on Passenger Car

M. Hariharan[1], E. Harish Babu[2], S. Kirubakaran[3], S. Gopalakrishnan[4]
[1,2,3,4]M.Kumarasamy College of Engineering, Thalavapalayam, Tamil Nadu, India.

ABSTRACT

This work proposes an effective numerical model using the Computational Fluid Dynamics (CFD) to obtain the flow structure around a passenger car with spoiler in a different angles. The computational/numerical model of the passenger car and mesh was constructed using ANSYS Fluent which is the CFD solver and employed in the present work. In this study, numerical iterations are completed, and then aerodynamic data and detailed complicated flow structure are visualized. In the present work, a model of generic passenger car was developed using solidworks, generated the wind tunnel, and applied the boundary conditions in ANSYS workbench platform, and then testing and simulation have been performed for the evaluation of drag coefficient for passenger car. In another case, the aerodynamics of the most suitable design of spoiler in a different angle called 10, 15, 20 degrees are introduced and analysed for the evaluation of drag coefficient for passenger car. The addition of these spoilers reduces the drag-coefficient and lift coefficient in head-on wind. Rounding the edges partially reduces drag in head-on wind but does not bring about the significant improvements in the aerodynamic efficiency of the passenger car, and it can be obtained. Hence, the drag force can be reduced by using spoilers on vehicle and fuel economy, stability of a passenger car can be improved.

KEYWORDS

Flexible-Fuel Vehicles, Para Solid-kernel, Gear Train.

Introduction

A car (or automobile) is a wheeled motor vehicle used for transportation. Most definitions of car say they run primarily on roads, seat one to eight people, have four tires, and mainly transport people rather than goods. Cars came into global use during the 20th century, and developed economies depend on them. The year 1886 is regarded as the birth year of the modern car when German inventor Karl Benz built his Benz Patent-Motorwagen. Cars became widely available in the early 20th century. One of the first cars that were accessible to the masses was the 1908 Model T, an American car manufactured by the Ford Motor Company. Cars were rapidly adopted in the US, where they replaced animal-drawn carriages and carts, but took much longer to be accepted in Western Europe and other parts of the world. Cars have controls for driving, parking, passenger comfort and safety, and controlling a variety of lights. Over the decades, additional features and controls have been added to vehicles, making them progressively more complex. Examples include rear reversing cameras, air conditioning, navigation systems, and in car entertainment. Most cars in use in the 2010s are propelled by an internal combustion engine, fuelled by the combustion of fossil fuels. This causes air pollution and also contributes to climate change and global warming. Vehicles using alternative fuels such as ethanol flexible-fuel vehicles and natural gas vehicles are also gaining popularity in some countries. Electric cars, which were invented early in the history of the car, began to become commercially available in 2008. There are costs and benefits to car use. The costs include acquiring the vehicle, interest payments (if the car is financed), repairs and maintenance, fuel, depreciation, driving time, parking fees, taxes, and insurance. The costs to society include maintaining roads, land use, road congestion, air pollution, public health, health care, and disposing of the vehicle at the end of its life. Road traffic accidents are the largest cause of injury-related deaths worldwide.The benefits include on-demand transportation, mobility, independence, and convenience. The societal benefits include economic benefits, such as job and wealth creation from the automotive industry, transportation provision, societal well-being from leisure and travel opportunities, and revenue generation from the taxes. The ability for people to move flexibly from place to place has far-reaching implications for the nature of societies. It was estimated in 2014 that the number of cars was over 1.25 billion vehicles, up from the 500 million of 1986. The numbers are increasing rapidly, especially in China, India and other newly industrialized countries.

Modelling Methodology

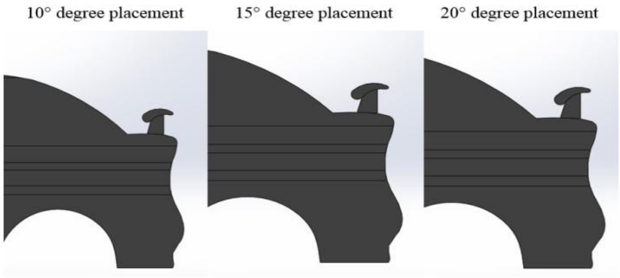
SolidWorks is a solid modeler, and utilizes a parametric feature-based approach to create models and assemblies. The software is written on Para solid-kernel.

Parameters refer to constraints whose values determine the shape or geometry of the model or assembly. Parameters can be either numeric parameters, such as line lengths or circle diameters, or geometric parameters, such as tangent, parallel, concentric, horizontal or vertical, etc. Numeric parameters can be associated with each other through the use of relations, which allows them to capture design intent.

Design intent is how the creator of the part wants it to respond to changes and updates. For example, you would want the hole at the top of a beverage can to stay at the top surface, regardless of the height or size of the can. SolidWorks allows the user to specify that the hole is a feature on the top surface, and will then honor their design intent no matter what height they later assign to the can.

In an assembly, the analog to sketch relations are mates. Just as sketch relations define conditions such as tangency, parallelism, and concentricity with respect to sketch geometry, assembly mates define equivalent relations with respect to the individual parts or components, allowing the easy construction of assemblies. SolidWorks also includes additional advanced mating features such as gear and cam follower mates, which allow modeled gear assemblies to accurately reproduce the rotational movement of an actual gear train.

Different Spoiler Geometries Used for Analysis Purposes



Solving the CFD Problem

1. Reading the file.The reading the file should clear as case file or data file or case and data file.In this we have to read case and data file.
2. Scaling the grid.
3. Checking the grid.
4. Defining the models.Modelshoulddefinewhetheritissteadyorunsteadyandwhetheritis viscous.The model is defined here is steady and viscous.
5. Defining the materials.
6. Defining the boundary condition
7. Controls
8. Initialize
9. Monitor
10. Iterate

The component that solves the CFD problem is called the Solver. It produces the required results in a non-interactive/batch process. A CFD problem is solved as follows:

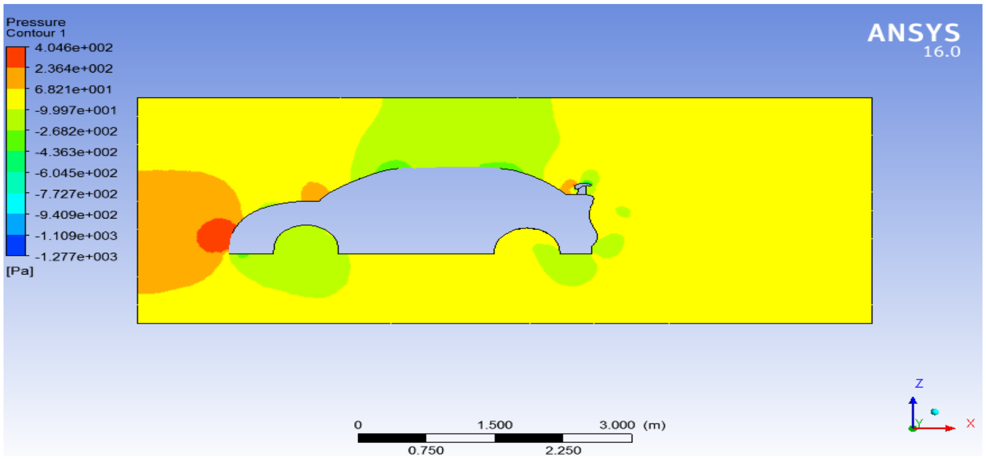
The partial differential equations are integrated over all the control volumes in the region of interest. This is equivalent to applying a basic conservation law(for example, for mass or momentum) to each control volume These integral equations are converted to a system of algebraic equations by generating a set of approximations for the terms in the integral equations. Thealgebraicequationsaresolvediteratively.Aniterativeapproachis requiredbecauseofthenon-linearnatureoftheequations,andasthesolution approaches the exact solution, it is said to converge. For each iteration an error, or residual, is reported as a measure of the overall conservation of the flow properties.

How close the final solution is to the exact solution depends on a number of factors, including the size and shape of the control volumes and the size of the final residuals.Complexphysicalprocesses,suchascombustionandturbulence,are often modeled using empirical relationships. The approximations inherent in these models also contribute to differences between the CFD solution and the real flow.

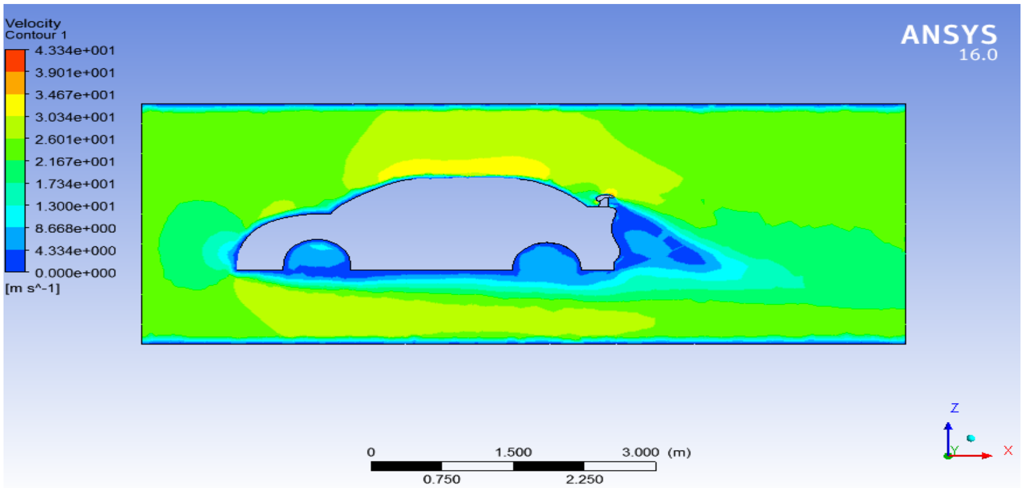
The solution process requires no user interaction and is, therefore, usually carried out as a batch process. The solver produces a results file which is then passed to the post-processor.

Results of Car with Spoiler of 10°

Pressure Distribution

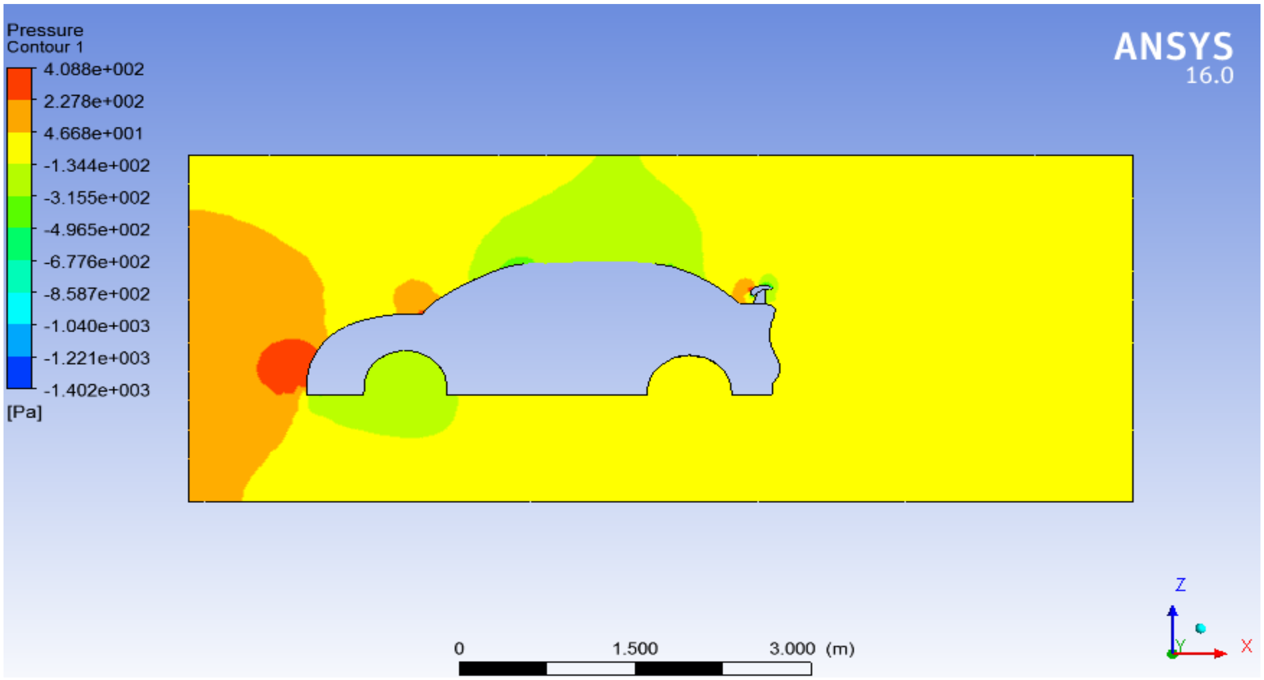


Velocity Distribution

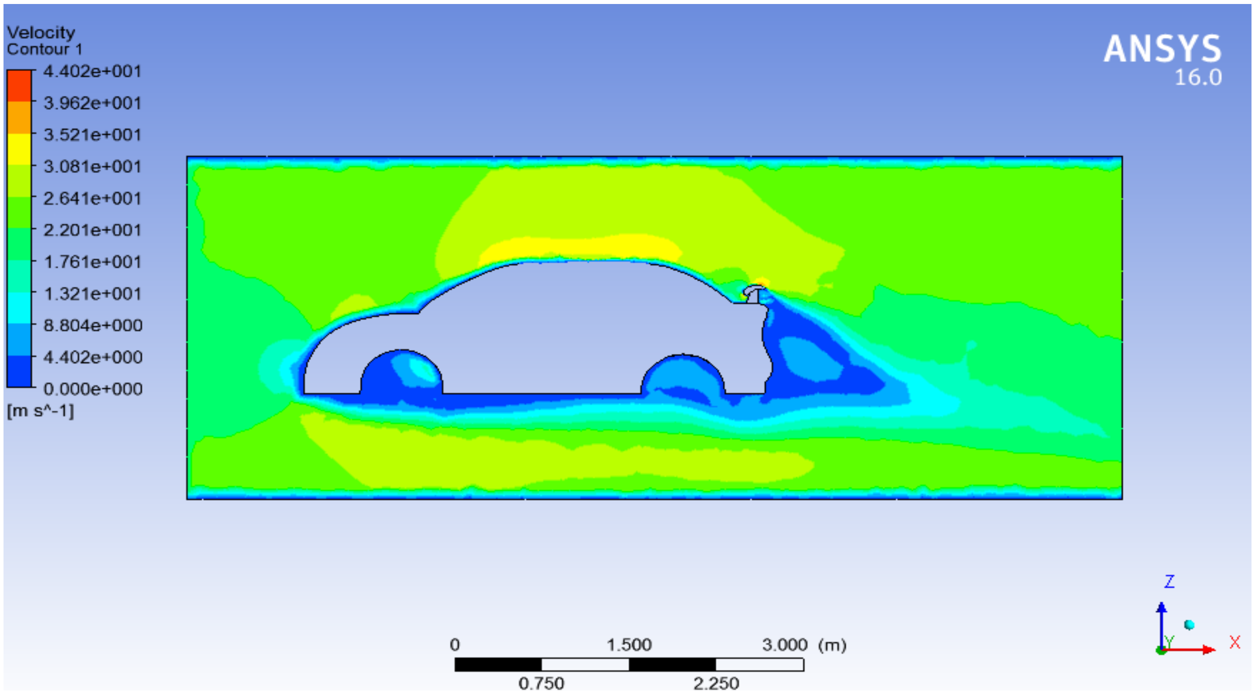


Results of Car with Spoiler of 15°

Pressure Distribution



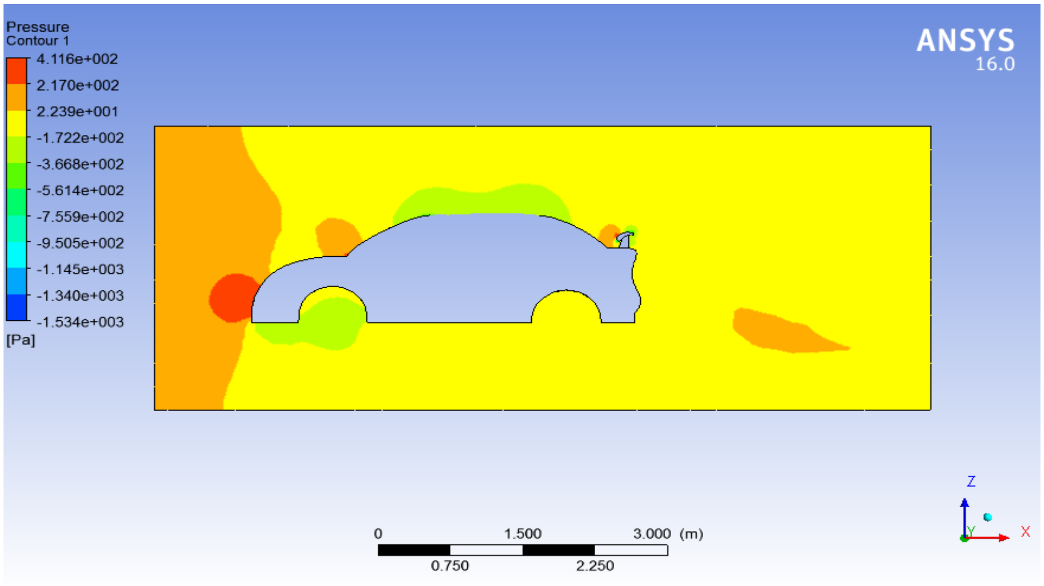
Velocity Distribution



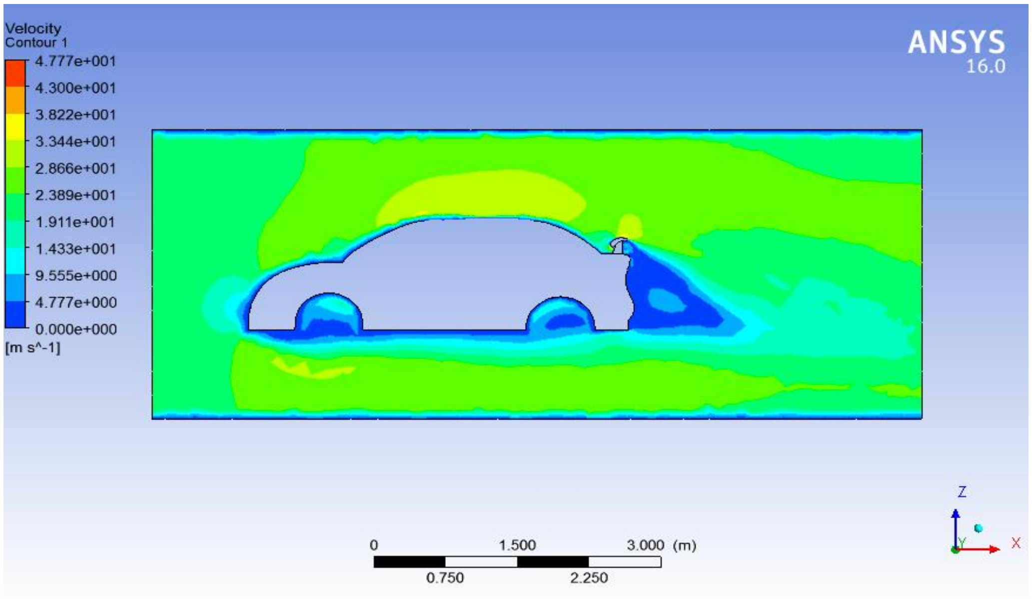
Results of Car with Spoiler of 20°

Received 25 April 2021; Accepted 08 May 2021.

Pressure Distribution



Velocity Distribution



Tabulated Result

MODEL	Velocity (m/s)		Pressure (pa)	
	Min	Max	Min	Max
10° spoiler	0	43.3	-1.27e3	4.04E2
15° spoiler	0	44	-1.40e3	4.08E2
20° spoiler	0	47.7	-1.53e3	4.11E2

Conclusion

Computational fluid dynamics (CFD) simulations of the steady flow field around passenger car models with and without spoilers both were presented comparing the simulated data to each other. The ANSYS Fluent with the - ε steady model is used for the simulations of aerodynamics. In this analysis, the coefficient of drag and coefficient of lift is reduced due to the addition of spoiler at an angle of 10°. Hence, the spoiler at specified angle is the effective tool to reduce the drag force on vehicle.

The effects of aerodynamic on different spoiler on flow and its structure over a generic passenger car is analyzed using CFD approach. The objective is to reduce aerodynamic drag acting on the vehicle and thus improve the fuel efficiency of passenger car. Hence, the drag force can be reduced by using spoiler on vehicle and fuel economy, and stability of a passenger car can be improved.

References

[1] Hans_Herman B & Ulrich S, handbook of automotive engineering (SAE international, Warrendale, Pennsylvania USA) 2005, 5-27.

[2] Aider JL, Dubuc L Hulin G & Elena L, Experimental and numerical investigation of the flow over a simplified vehicle shape,*In Proc 3rd MIRA Int Vehicle Aerodyn Conf(Rugby, UK)*2000.

[3] Stapleford WR, Aerodynamic improvements to the body and cooling systems of a typical small saloon car,*J Wind Eng Ind Aerodyn, 9*(1981) 63-75.

[4] McCallen R, Browand F, Leonard A, & Rutledge W, Systematic approach to analyzing and reducing aerodynamic drag of heavy vehicles, *Annu Auto Tech Dev customers Coord Meeting (Dearborn, Michigan)* 27-30 October 1997.

[5] Frederique muyl, Laurent Dumas & Vincent Herbert, Hybrid method for aerodynamic shape optimization in automotive industry, *Comp & Fluid, 33* (2004) 849-858.

[6] Hucho W H, Aerodynamic of road Vehicles (Butterworth, London) 1997.

[7] Katz J, Race Car Aerodynamics (Robert Bently Publishing) 1995.

[8] Manan Desai, SA Channiwala and HJ Nagarsheth, A Comparative assessment of two experimental methods for aerodynamic performance evaluation of a car, *Journal of Scientific & industrial research*, vol. 67, july 2008.

[9] Hucho, W. H. *Aerodynamics of Road Vehicles,* 1997 (Cambridge University Press, Cambridge, UK).

[10] Lachmann, G. V. *Boundary Layer and Flow Control,* Vols I and II, 1996 (Pergamon Press, Oxford, UK).

[11] Goldstein, S. *Modern Developments in Fluid Mechanics,* Vols I and II, 1968 (Oxford University Press, Oxford, UK).

[12] Chang, P. K. *Separation of Flow,* 1970 (Pergamon Press, Oxford, UK).

[13] Schlichting, H. *Boundary Layer Theory,* 1968 (McGraw Hill Book Company, Maidenhead, UK).

[14] Rosenhead, L. *Laminar Boundary Layers,* 1966 (Oxford University Press, Oxford, UK).

[15] Lissaman, P. B. S. and Lambie, J. H. Reduction of aerodynamic drag of large highway truck. *Proceedings of the conference/workshop on The Reduction of the Aerodynamic Drag of Trucks, California Institute of Technology,* 10–11 October 1974, pp. 89–120 (National Science Foundation RANN Document Centre, Washington DC).

[16] Kirsh, J. W. and Bettes, W. H. Feasibility study of the S3 air vane and other truck drag reduction devices. *Proceedings of the conference/workshop on The Reduction of the Aerodynamic Drag of Trucks, California Institute of Technology,* 10–11 October 1974, pp. 89–120 (National Science Foundation RANN Document Centre, Washington DC).

[17] Montoys, L. C. and Streers, L. L. Aerodynamic drag reduction tests on a full scale tractortrailercomlayer. with several add on devices. *Proceedings of the conference/workshop on The Reduction of the Aerodynamic Drag of Trucks, California Institute of Technology*, 10– 11 October 1974, pp. 65–88 (National Science Foundation RANN Document Centre, Washington DC).

[18] Singh.S.N.,Veeravalli,S.V.,Bhatnager,A., Puri, P. Effect of boundary layer control using momentum injection on the lift characteristics of a NACA airfoil. *Proceedings of the 2nd International conference and 29th national conference of Fluid Mechanics and Fluid Power*, IIT Roorkee, India, 13–15 December 2002, Vol. I, pp. 252–259 (Ajay Printers and Publishers, Roorkee).

[19] Modi, V. J., Mokhtarian, F., and Yokomizo, T. Effect of moving surfaces on the airfoil boundary layer control. *J. Airc.*, 1990, 27(1), 42–50.

[20] Modi, V. J., Mokhtarian, F., and Yokomizo, T. Bound vortex boundary layer control with application to V/STOL airplanes. *Fluid Dyn. Res.*, 1 September 1988, 3(1–4), 225–230.

[21] Shubham Sawant and Deep Prajapati. A Review on Zero Emissions Vehicles. *International Journal of Mechanical Engineering and Technology, 8*(2), 2017, pp.198–202.

[22] Apoorv Prem, Articulated Vehicle Systems, *International Journal of Mechanical Engineering and Technology, 5*(7), 2014, pp. 36–41.

[23] Hoerner, S. F. *Fluid Dynamic Drag,* 1965 (US Library of Congress catalog card no. 64- 19666).