International Journal for Modern Trends in Science and Technology
Volume: 09, Issue No: SI02, March 2023
ISSN: 2455-3778
http://www.ijmtst.com

Comparison of CFD Drag analysis on Heavy Motor Vehicle between paraboloc draft at Top and Decremental rectangular surface

P Venugopal[1] | N SyambabuT[2]

[1]Automobile Engineering,Godavari Institute of Engineering and Technology,Rajahmnudry,Andhrapradesh,India
[2]Mechanical Engineering,Kakinada Institute of Technology and Science,Divili,Andhrapradesh,India

ABSTRACT

In this era of fast-depleting natural resources, the hike in fuel prices is ever- growing awareness among people about conservation of non-renewable energy like fossil fuels researchers and scientists are investigating a lot on how to reduce the consumption of these fossil fuels either by using alternative fuel or by improving the performance of the gadgets like vehicles, aeroplane and other energy converting devices. There are many papers publish in the past in field of aerodynamics of vehicles intended to optimize the geometry of these vehicles to reduce the fuel consumption and maximize the performance by reducing the aerodynamic drag. Researches in this field are still growing in numbers. This paper also intends to reduce the aerodynamic drag by studying the flow field around the heavy Motor Vehicles. This paper compares the results obtained from CFD analysis of the existing model and the modified model with a duct added to the roof of a sedan model car. The analysis is carried out using ANSYS Fluent and the car 3D model is created using CATIA V5.

KEYWORDS: *Keyword 1, Keyword 2, Keyword 3, Keyword 4, Keyword 5*

I. INTRODUCTION

When the vehicle is moving at a particular velocity, the viscous effects in the fluid are observed only to a thin layer called boundary layer. Outside the boundary layer viscous effects are not felt i.e. flow is inviscid. This fluid flow imposes pressure on the edge of boundary layer. When the air reaches the rear end of the vehicle, the fluid gets separated. Within the boundary layer, the motion of the fluid is governed by the viscous property of the fluid. There does not exist any Boundary layer for Reynolds number lower than 104. Reynolds number depends on the characteristic length of the body, the kinematic viscosity and the velocity of the vehicle. In other words, the fluid moving around the vehicle depends on the shape of the vehicle and the Reynolds number. Another phenomenon which affects the flow of fluid past the vehicle body and the performance of the vehicle which known as 'Wake'. When the air moving over the vehicle gets separated at the rear end, it creates low pressure turbulent region behind the vehicle known as the wake. This contributes to the pressure drag, which reduces of the vehicle performance.

II. LITERATURE SURVEY

Abdul Razzaque [1] studied about the design of a vehicle to minimize the drag force to get at economic model with optimized the performance. He analysed the 3D design model of Tata Indica car designed using Pro-E 5.0 and used ANSYS Fluent platform to analyse the flow field around the car. His work was concerned and intended on reduction of the coefficient of drag & drag force on car body by optimizing the exterior shape using CFD software. This analysis was done to calculate the coefficient of drag and drag resistance. Different design parameter of car was analysed using a suitable turbulence model and comparison of coefficient of drag of the car model was carried out using CFD software he validated his experimental result using CFD analysis/experimental studies. The experimental results were found to agree with the results found through CFD analysis which shows that CFD analysis is an effective tool to study the aerodynamic design of cars.).

Rubel Chandra Dasa [2] made a detailed study about the aerodynamical shape of the car which uses approximately 3% of fuel to resist the drag force in urban driving, while it uses 11% of fuel during the highway driving. He observed that during highway driving fuel consumption is considerably high; this encouraged him to improve the aerodynamics of the vehicle using optimal design changes. He was also observed that at high speeds the vehicle experience large lift force which leads to a large number of accidents. As a result he proposed the usage of add-on devices, which could be assembled to the existing vehicle without changing the body. His proposal was based on the design, developments and numeral calculation of the effects of spoiler mounted at the boot of the vehicle. The analysis results like lift, drag, and pressure distributions due to spoiler assembly were investigated and reported using Autodesk Simulation Shreenidhi R Kulkarni [3] he analysed and simulated the existing model Tata 1613 truck in ANSYS FLUENT. He used CATIA V5 to model the original truck and he analysed the same drag coefficient. He determined and evaluated aerodynamic drag and fuel utilization. He made modifications in truck and again analysed for the drag and fuel consumption. He observed that the results were promising and optimization in drag force was achieved and hence fuel consumption was reduced by 43%. J Abinesh he modified [4] the outer body and structure of the bus in order to reduce the drag force on the bus which in turn minimizes utilization of fuel for the vehicle. He created two models of the external bus surface and used CFD to analyse the model. The two models used for analysis were the existing Volvo intercity bus model and its modified version. He observed that drag force was minimized as a result the performance of the bus improved and fuel consumption was also reduced. The overall reduction in aerodynamic drag force was observed as 10%. Toukir Islam et.al [5] .made a detailed study on the various kinds of drag forces that considerable in case of cars. He calculated the aerodynamic coefficient of drag and suggested various techniques and design considerations to minimize drag force.given a copyright form and the form should accompany your final submission.

III. Methodology

PROJECT STATEMENT This paper focuses understanding the flow around the sedan car model. The aerodynamic drag on the car external body can be used by streamlining the car geometry or by using add-ons which can be attached to car's external body. The cars performance depends on the resistive forces acting on the car's external body surface due to the flow of air around the car body. There are two types of forces that act on the car the aerodynamic drag aerodynamic Lift. This paper focuses on minimizing drag force on the car body thereby reducing fuel consumption using an air duct fixed at the roof thereby increasing the performance of the car. Analysis is done on the existing model and the modified model using CFD Fluent 14.5 and the results obtained for both models are compared and the best model is suggested. The car is modelled using CATIA V5. The project focuses on reducing the wake area at the rear end of the car which is formed due to the flow reversal near the wall. SOLID MODELING As we discussed earlier for aerodynamics analysis we require either a scaled prototype of the car for wind tunnel testing or a computer model to carry out simulation in ANSYS. As wind tunnel testing is expensive, requires large set-up and time consume therefore this project relies on CFD analysis technique for carrying out analysis. For analysis in CFD we require a 3D model which can be modelled using surface modelling softwares like Creo, CATIA V 5, and Solidworks etc. any of these softwares can be used depending on one's skill. For this project work CATIA V5 Surface.

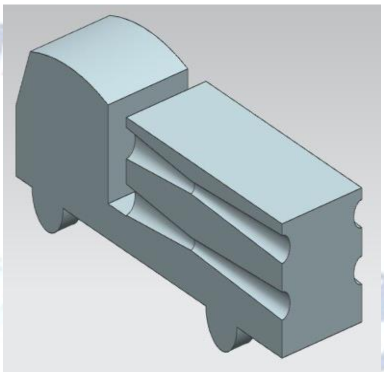


Figure-1 with Convergent-Divergent Nozzle

MESHING

The process of discretization model into number of elements is known as meshing. It is the most important step in a FEA analysis. The quality of mesh defines the accuracy of the result. We must require refining the meshing to get crisp and clear elements with uniformity. The meshing is done in ANSYS Mesh Generator. Before creating the mesh a fluid enclosure is created using CREATE>Enclosure.

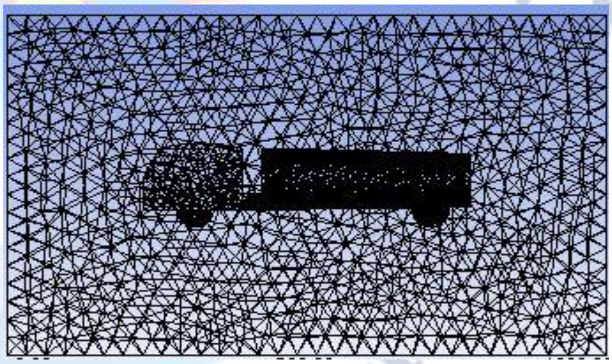


Figure-2 Meshing

CFD SOLVER (SET UP)

This is the main part of CFD where our model solved and analysed according to the given boundary. Special care should be taken while defining the steps in the solver. The setup of the solver varies with the type of analysis we are performing. It takes up the most part of time of CFD analysis. It is here where the size of generated comes into play. Finer the mesh more the time will it consume to solve our problem because the equations are solve for each and every element, smaller the element size more will be the number of elements for a given model. The solver setup includes following steps:-

1. Type of analysis: For our aerodynamic analysis we carry out 3D model analysis which pressure based with relative velocity formulation. We do steady state analysis.

2. Models: here the type of numerical model used for the analysis is defined. For aerodynamic analysis Viscous based numerical model. The type of viscous based model used is realizable k-epsilon and scalable wall function model

3. Materials: here the material for the model is defined. In this work the analysis is carried out on Air which is the surrounding fluid. Here properties of the material can be retrieved from ANSYS Library.

4. Boundary conditions: this is the core part of ANSYS Solver here the initial boundary conditions are defined. Without defining these variables we cannot initialize the solution that is why they are called initial boundary values. The initial boundary values for our analysis are :- a. Inlet fluid velocity: - 50km/hr. b. Wall: - Stationary

5. Solution methods :- a. Pressure-Velocity, Coupling b. Scheme: Simple c. Gradient: - Least Square Cell Based d. Pressure: - second order e. Momentum: - second order upwind

6. Monitors :- from the required Cd and Cl After carrying out all these initial set-ups run the calculation with required number of iterations. More the number of iterations the obtained result will be more close to the actual values. The accuracy of the results can be determined by the convergence of the Cd and Cl plots.

IV.RESULTS AND DISCUSSIONS

All the designed Heavy motor vehicles are models with different and combined aerodynamic devices are simulated in ANSYS™ 17.0 Fluent. The results of the drag coefficients obtained are discussed below. Figure-3 depicts the streamlines plot derived from the simulation for different cases. As seen from the streamlines plot in the figure, minimum flow separation is favorable, which subsequently leads to lesser turbulence. The amount of turbulence created behind the rear region of the car determines the magnitude of the drag force. In the case of Rectangular Draft is found more drag forces when compared to Paraboloic draft on the top side.

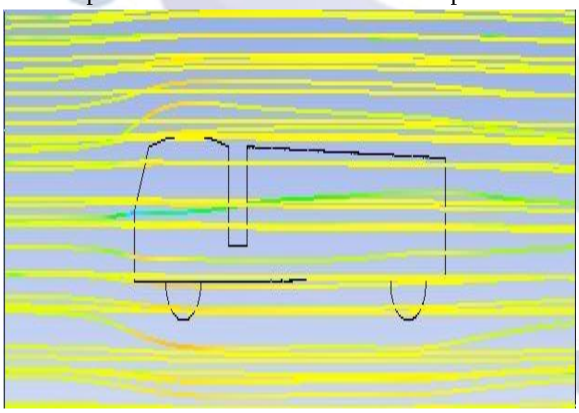


Figure-3 Vehicle surface with decremental rectangular draft

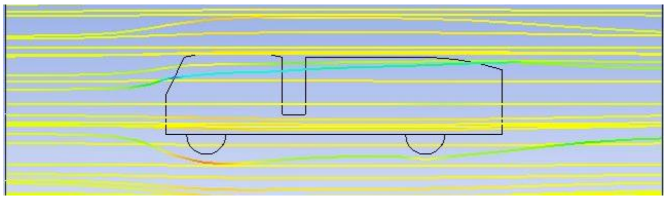
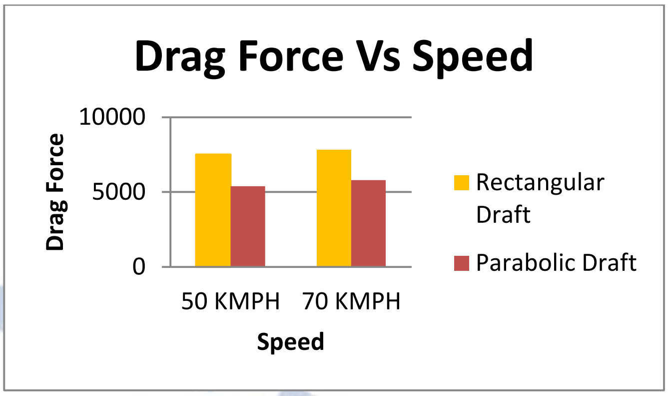


Figure-4 Vehicle with Parabolic draft at topside



IV. CONCLUSION

The constant evolution in the history of vehicle aerodynamics has led to the development of certain devices which led to the enhancement of the overall aerodynamic characteristic of the vehicles. Not only it improves the efficiency of the vehicle but also reduces fuel consumption. The analysis of the baseline Rectangular Draft with different add on aerodynamic devices was studied by using numerical simulation in this paper. It has been found that aerodynamic drag can be influenced by using different add on devices. In consideration to reduce drag, it is favorable that the flow is attached to the vehicle's body as long as possible. A streamlined body would result in less flow separation, which would cause less turbulence. In the case of GT Spoiler with Diffuser, maximum drag reduction of 16.53% is observed. Although other devices like fins also reduced drag to a much extent, they may pose a different functionality such as high-speed stability by channeling flow at rear accordingly. Wings have altogether a different function. It indeed increased the drag but its prime function is to provide downforce at the cost of increased drag, and it is much like a trade-off. Diffusers on the other hand decreased the drag whenever applied in different cases. In conclusion, it may be regarded as proper optimization can lead to better aerodynamics of the vehicle in different scenarios

REFERENCES

[1] Abdul Razzaque Ansari "CFD Analysis Of Aerodynamic Design Of Tata Indica Car", International Journal of Mechanical Engineering and Technology (IJMET) Volume 8, Issue 3,pp. 344–355, March 2017

[2] Rubel Chandra Dasa, Mahmud Riyada, "CFD Analysis of Passenger Vehicle at Various Angle of Rear End ", Science Direct, pp.160-165, 2017.

[3] Shreenidhi R Kulkarni, Abhishek Kothari, Chetan Pavate, Chinmay Dodmani, "Aerodynamic Simulation of A Truck To Reduce The Drag Force" , International Journal of Engineering Research ISSN:2319-6890 Volume No.4, Issue No.11, pp. : 613-617 , Nov. , 2015

[4] J Abinesh and J Arunkumar, "CFD Analysis of Aerodynamic Drag Reduction and Improve Fuel Economy", IJMERR, ISSN 2278 – 0149 Vol. 3, No. 4, October, 2014.

[5] Toukir Islam, "Numerical Study on Aerodynamic Drag Reduction of Racing Cars", Science Direct, pp.308 – 308 , 2016

[6] V. Naveen Kumar, K. Lalit Narayan, L. N. V. Narasimha Rao and Y. Sri Ram, "Investigation of Drag and Lift Forces

over the Profile of Car with Rea spoiler using CFD”, International Journal of Advances in Scientific Research,ISSN: 2395-3616 ,pp.331-339,2015

[7] Simon McBeath, “Competition Car Down force” , 1998 Joseph Katz, “Race Car Aerodynamics- Designing for Speed”, ed., Bently Publishers, 1995.

[8] Dan Barbut, Eugen Mihai Negrus, “CFD analysis for road vehicles - case study”, Incas Bulletin, 3(2011) 15-22.

[9] VDM Verlag Dr. Müller, “Aircraft Performance Analysis” , 2009

[10] Prof. Tamás Lajos, “Spoiler Basics of Vehicle Aerodynamics”, Budapest University of Technology and Economics.J. Wang, “Fundamentals of erbium-doped fiber amplifiers arrays (Periodical style—Submitted for publication),” *IEEE J. Quantum Electron.*, submitted for publication.