

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Issue 5, May 2018

CFD Analysis of Aerodynamics of Car

R.Varun¹, Dr. S.Sankar², Rajiv Varma³, Sreejith K.V.⁴

M.Tech Scholar, Dept. of ME, NCERC, Pampady, Kerala, India¹

H.O.D, Dept. of ME, NCERC, Pampady, Kerala, India²

Asst. Professor, Dept. of ME, NCERC, Pampady, Kerala, India³

Asst. Professor, Dept. of ME, NCERC, Pampady, Kerala, India⁴

ABSTRACT: With 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 published 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 car. 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: ANSYS CFD, Aerodynamics & Drag Force.

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. RELATED WORK

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.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Issue 5, May 2018

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.

Naveen Kumar[6] observed that forces like drag & lift, weight, side thrust and thrust acts while a vehicle is on motion, this considerably increases the consumption of fuel. He observed that the aerodynamic drag is result of flow of air around the vehicle and almost 60% of total drag is produced at the rear end. He stated Reducing drag force at the rear end will optimize the fuel consumption. His work was intended to minimize the drag force which will optimize the fuel consumption and ensure cleanliness of environment. He modelled a sedan car with different types of spoilers to minimize the drag. He used CATIA-2010 for design of sedan car and the analysis was carried out in ANSYS-(fluent). The analysis was intended to evaluate the drag and lift forces at various velocities and spoiler angles. To obtain the flow structure around the car He proposed an effective numerical model based on the computational fluid dynamics (CFD) approach with a rear spoiler.

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 modeller has been used to develop a 3D-Model of the car body. Various modelling tools have been used like Multi-

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Issue 5, May 2018

section, Fill, 3D-curve, Connect curve, Split curve, Sweep section, Point etc. A sedan car 3D surface model has been created and then it is converted to a solid model using ANSYS Design Modeller using Fill by Cavity Tool.

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.

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: - 100km/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 C_d and C_l

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 C_d and C_l plots.

IV. RESULTS AND DISCUSSIONS

The existing model was analysed in ANSYS and the following data were found:-

1. The coefficient of drag was found to be 1.02
2. The pressure at the frontal area was to be 200 Pa along the flow direction
3. The pressure at the rear end was found to be 107 Pa opposite to the direction of flow.
4. The pressure difference was found to be 93 Pa opposite to the direction of flow.

From the data we can infer that a pressure difference of 93 Pa acts opposite to the direction of flow which acts as resistive force to the motion of car. The results obtained from the analysis of the existing models were compared with the results obtained from the modified model with a duct fitted at the roof. Still, some small non-text objects are detected.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Issue 5, May 2018

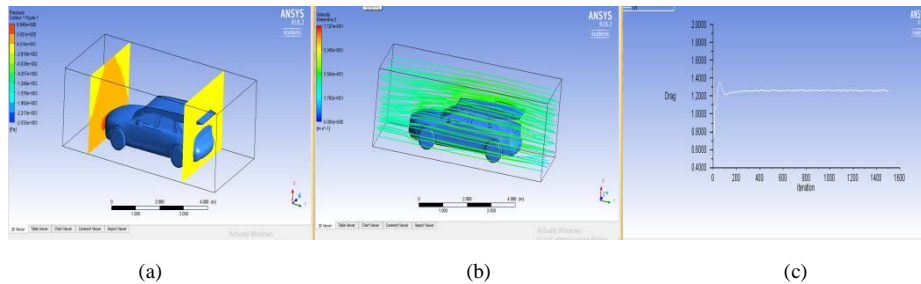


Fig.1.Pressure Contour Plot at the front and rear end of the existing model (a) Original image (b) Streamline over the existing car model (c) Iterations vs. coefficient of drag plot for existing model

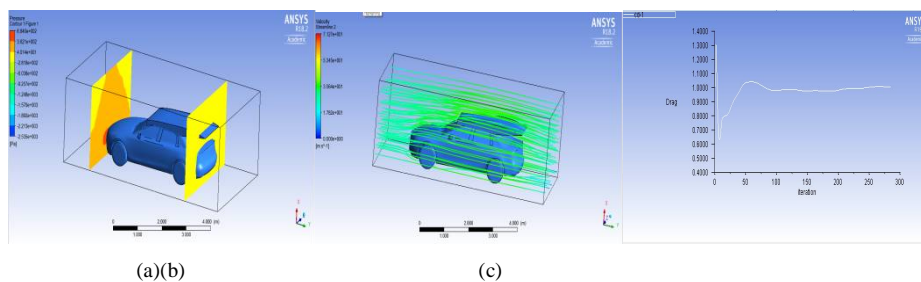


Fig. 2Pressure Contour plot at the rear and the front end of the modified model (b) Streamline around the modified model (c) Iterations vs. coefficient of drag plot for the modified model

Duct with the below mentioned dimensions is introduced at the roof:-

1. At Front:-
 - Height= 147mm, Width= 1220mm
2. At the Rear:-
 - Width = 1220 mm, Height = 235mm
3. Thickness=10mm, Length = 1630mm

Duct is diverging duct which allows the air to flow through it and compensate the wake formed behind the vehicle by allowing and guiding the air with high pressure to fill the wake created and thereby decreasing the pressure drag. The modified is analysed using ANSYS fluent to understand the flow through the car and whether the introduction of the duct reduces the drag force. From the analysis following data about the aerodynamic parameters are calculated as mentioned below:-

1. The coefficient of drag was found to be 1.196.
 2. Pressure at the front of the car was calculated and found to be 253 Pa along the direction of flow or opposite to the motion of car.
 3. The pressure at the rear end was found to be 197 Pa along the direction opposite to the flow or along the motion of car.
 4. The pressure difference between the rear and the back end is 52 Pa.
- This pressure difference produces a resistive force opposite the motion of car.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Issue 5, May 2018

VI. CONCLUSION

From the analysis we compared the results of the existing and the modified model which is shown in tabular column given below:-

Table 1: Showing the comparison of results obtained from CFD analysis of existing and modified model.

Model	Existing	Modified
Drag	1.20	1.10
Pressure at the front end	200 Pa	253 Pa
Pressure at the rear end	107 Pa	197 Pa
Pressure difference	93 Pa	56 Pa

It can be seen clearly that the modified has a low pressure drag as the pressure difference at front and the rear end is low however there is no significant deviation in the coefficient of drag has therefore the modified model is the optimized model.

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
- [8]. Joseph Katz, "Race Car Aerodynamics- Designing for Speed", ed., Bently Publishers, 1995.
- [9]. Dan Barbut, Eugen Mihai Negrus, "CFD analysis for road vehicles - case study", Incas Bulletin, 3(2011) 15-22.
- [10]. VDM Verlag Dr. Müller, "Aircraft Performance Analysis" , 2009
- [11]. Prof. Tamás Lajos, "Spoiler Basics of Vehicle Aerodynamics", Budapest University of Technology and Economics.