International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 3 Issue 6, June - 2014

Aerodynmic Improvement of Car Body

Yogendra Kumar Vishwakarma[1], Mohit Raj Saxena[2], Dr. P. K. Nagrajan[3]
M.Tech Scholor
*Proffesor.
Scme Galgotias University
City-Greater Noida Uttar Pradesh

Abstract **- Now days the demand of high speed car is continuously increasing. At high speed the vehicle stability is major concern and many forces act on the vehicle when moving on road like drag force which directly affects the fuel consumption. The drag force is produced by relative motion between air and vehicle. About 60% of total drag produced at the rear end and significant amount of power is degrading to overcome the drag force. In vehicle body development, reduction of drag is important issue in order to keep lower fuel consumption, better driving characteristic and to protect the environment. In this research paper sedan car model was modified with two adds i.e. deck lid spoiler and grooves in the back, to reduce the aerodynamic drag force. The designing of sedan car have done on SOLIDWORKS-2010 and analysis have done by using ANSYS-15.0(FLUENT) with boundary condition at inlet velocity is 30 m/s (108kmph) and at outlet pressure is zero to made virtual wind tunnel.**

1. INTRODUCTION

Now a day's environmental issue is a major concern of automobile industries. Main objective of automotive industries is to reduce the fuel consumption and emissions and also the demand of faster and fuel efficient car increases day by day. The importance of aerodynamic of car is to determine the drag coefficient to know the how much car perform on road against the air resistance. It affects the fuel consumption, vehicle stability, driving and noise characteristics of car. In aerodynamic, shape and size of vehicle has direct affect on aerodynamic drag. In car aerodynamic generally apply for the reducing drag and improve the flow around the vehicle.

Carr.G.W.et al (1982), investigated the effects of streamlining the front end of the rectangular bodies in ground proximity. Experiments shown a stream lined front end with low leading edge resulted in a drag coefficient of 0.21[1]. Manan Desai et al(2008) did the comparative assessment of two distinct experimental strategies of aerodynamic predictions by conventional wind tunnel approach and its subsequent validation with advanced computational procedures, carried out as a part of design process of a small hybrid car proposed to be named as ADRENe [2]. Aniket A Kulkarni et al (2012) did the analysis of flow over a convertible car; it involves comparison of the coefficient of drag for two configurations of the convertible car with the roof and without the roof [3]. Mohan Jagadeesh Kumar et al (2013) used rear, bump-shaped vortex generators at the roof end of a car are tested in this paper for two different types of car models Sedan and Hatchback [4].

2. METHODOLOGY OF WORK

In this research work solid model of sedan car and the modified model is made on Solidworks-2010 version. First model is obtained by making grooves at the rear end of the car.

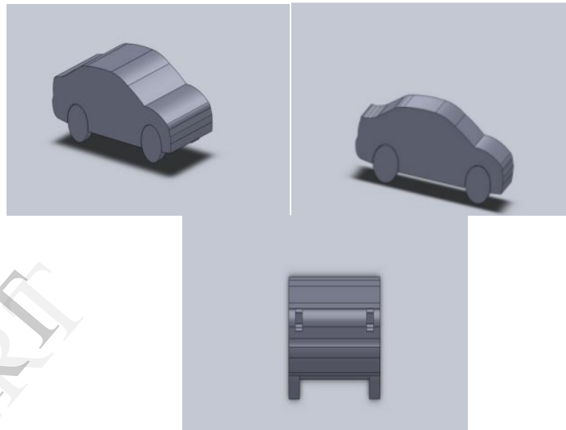

Fig. 1 Solid model of base and their modified models

This analysis is carried out in an enclosure made in ANSYS 15.0 of dimensions 7.695 m x 8.5709 m x 28.202 m .

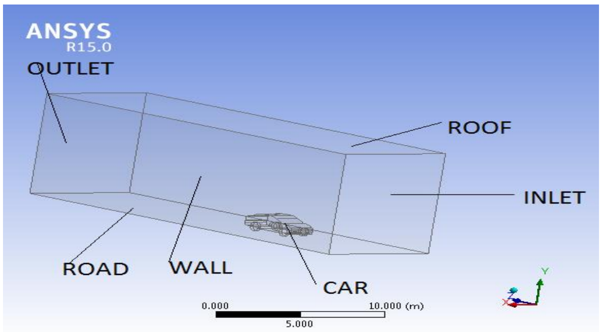

Fig.2 Car inside the enclosure

The analysis is carried out in FLUENT with realizable k-∈ turbulence model. The velocity of streamline is only given at the inlet of the enclosure. The velocity given at the inlet is 108km/hr. Simple algorithm is applied for the solution of momentum, kinetic energy and dissipation rate.

3. RESULT AND DISCUSSION

Figure3 shows the pressure in the front of the car is highest and decreases back along the back of the car. It means that there is favorable pressure gradient**.**

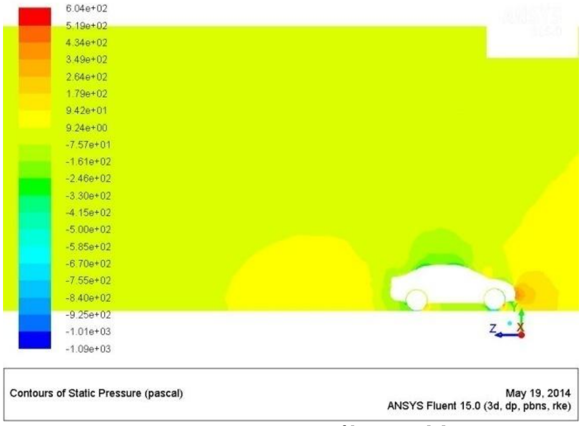

Fig.3 Pressure contour of base model

Figure4 shows that there is large stagnant zone in the front of the car and the large dead zone is created in the back of the car. Due to this large dead zone the vortex generated in the back of the car. It means turbulence region is created in the back of the car. Figure5 shows the turbulence intensity in the base model. High turbulence created in back retard the velocity of the vehicle. The coefficient of drag calculated for this model is 0.457.

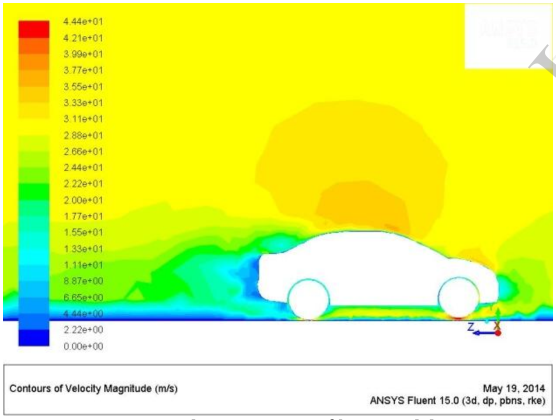

Fig.4 Velocity contour of base model

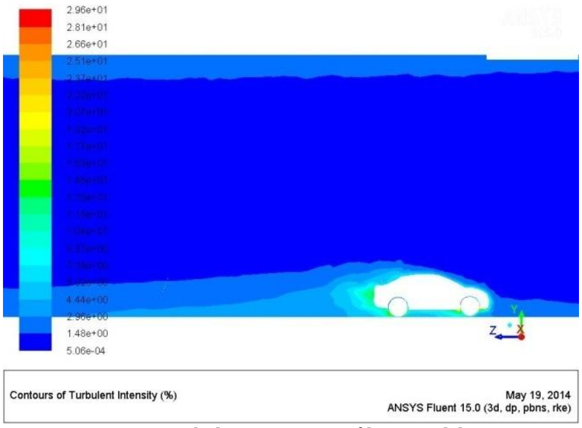

Fig.5 Turbulence intensity of base model

The pressure in the front of the car is less in base model in comparison to first modified model, it can be easily understand from figure 6.

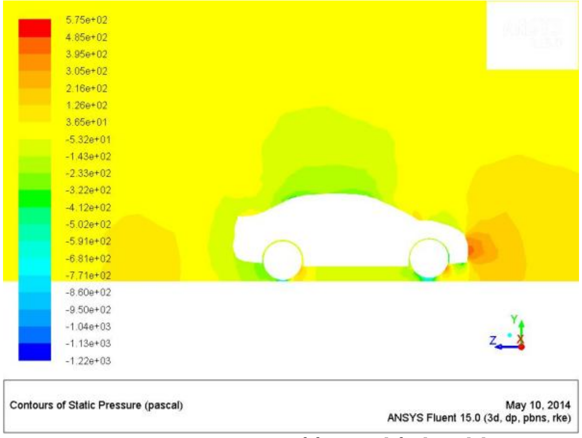

Fig.6 Pressure contour of first modified model

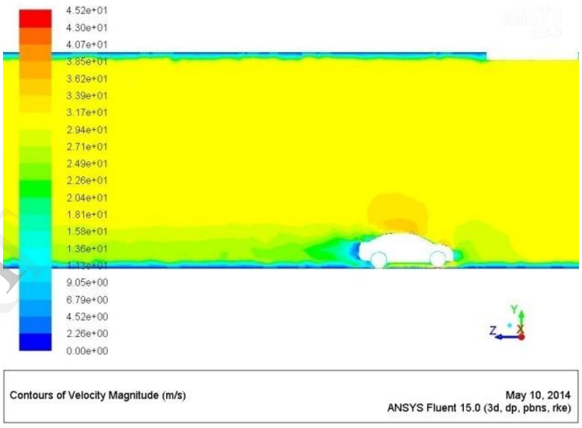

Fig.7 Velocity contour of first modified model

The vortices generated in the back of first modified model are less as compared to base model.

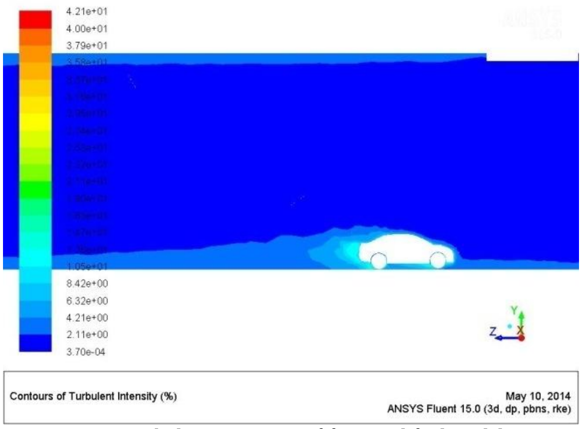

Fig.8 Turbulence intensity of first modified model

Figure9 shows that pressure is less at the front in comparison to base model so the turbulence created in the back of the car is less. So the drag produced in second model is less as compare to base model. The calculated drag coefficient for second model is 0.447. Drag coefficient in this model is reduce by 2.18%.

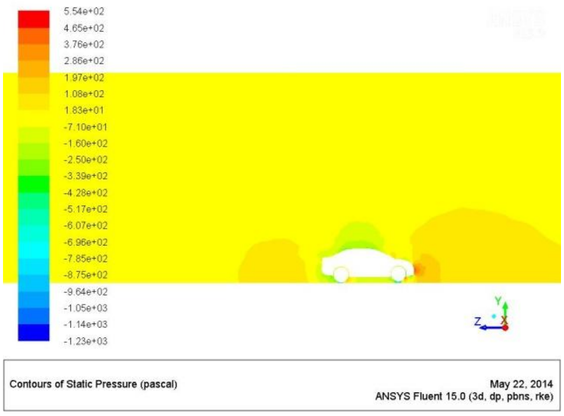

Fig.9 Pressure contour of second modified model

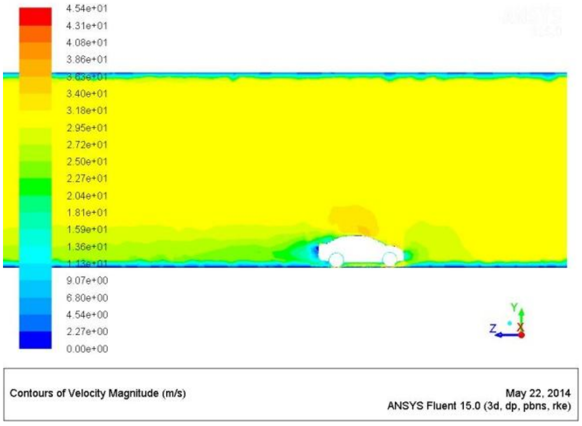

Fig.10 Velocity contour of second modified model

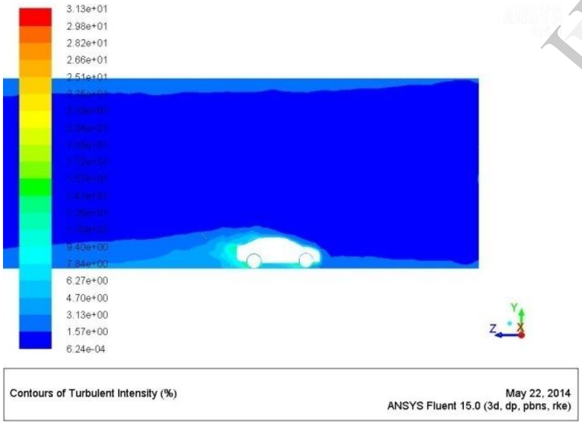

Fig.11 Turbulence intensity of second modified model

4. CONCLUSION

This computational analysis shows that there is possibility of improving the aerodynamic performance of car by modifications in exterior design of car body. These modifications are helpful in reducing the coefficient of drag i.e. C_d which effects the fuel consumption. By these modifications the coefficient of drag is reduced by approximately 2.18%

5.REFERENCE

[1] Carr.G.W., (1982), “The aerodynamics of basic shapes of road vehicles, part 1, Simple rectangular bodies”, MIRA report No.1982/2.
[2] MANAN DESAI et.al “experimental and computational aerodynamic investigations of a car ISSN: 1790-5087 Issue 4, Volume 3, October 2008
[3] Aniket A Kulkarni et.l Analysis of Flow over a Convertible Computer-Aided Design & Applications, PACE (2), 2012, 69-75,2012 CAD Solutions, LLC
[4] Mohan Jagadeesh Kumar M et.al Effect of Vortex generators on Aerodynamics of
[5] a Car: CFD Analysis vol-2 issue 2 april-2013.ISSN-2319-1058