Research Article

Comparing Aerodynamic Design stability of Passenger Cars Manufactured for Indian Conditions

Prince Kumar[1]

Abstract

The primary objective of this study is to compare the aerodynamic impact and stability of passenger vehicle which are used in India. To measure and compare the aerodynamic drag of two car models widely used on Indian roads in north India, computational fluid dynamics simulation using *FLUENT* software was used to visualize airflow around the car geometry. CFD analysis achieved was used to analyse the critical places in geometry which result in unfavourable aerodynamics of moving car. The aerodynamic drags on the baseline models at different sides of the cars (i.e., front faring, side skirting and gap filling) were measured for both the vehicle operating at same speeds and yaw angles, and with different combinations. The results show that the different car geometry has notable impact on aerodynamic drag and stability.

Keywords: Passenger vehicles, Aerodynamic drag, CFD analysis

Introduction

In automobile sector or and many other industries one of the objective of any work measurement technique is to reduce the work content and effort to improve the productivity of the process [6]. The challenge is to provide just in time, mass customization to maximize value @ design to manufacture to assembly of engineered products [4]. The external car body shape plays an important role in the automotive industry as it influences vehicle's performance. The comfort, fuel performance etc all are influenced by vehicles external shape [5]. For this purpose the car shapes are analysed for stability using various techniques [1]. Conventionally wind tunnel experiments were used which were cumbersome and time consuming but now-a-days, CFD analysis has also become more popular as it helps to reduces number of experiments significantly [3]. Thus computational and experimental investigation finds wide applications in engineering fields for carrying out analysis now-a-days [9]. The importance of aerodynamic shape can be seen from simple example that if velocity of racing cars is increased from 250 km/h to 300km/h without altering its shape, we need increase its power from 400 hp to 550 hp. An alternate solution to this is decreasing coefficient of drag [7]. Thus vehicle designs are altered for design in such a way that they have reduced drag coefficient [8]. It therefore makes a case for the present work as the accent on newer car development embraces intently the dwindling resources and/or constraints, on various fronts, including design or materials, or even concepts around which cars were developed [10]. When vehicle moves in air there is two main concerns area is front area with increased pressure and rear vacuums which is need to be minimum for minimum drag forces [11]. Few sites of pressure distributions, which must be taken care of while designing are shown in figure 1.

[1]PG Student, Mechanical Engineering Department, PEC University of Technology, Chandigarh.

E-mail Id: pkgoyal94@gmail.com

Orcid Id: 0000-0001-5257-6456

How to cite this article: Kumar P. Comparing Aerodynamic Design stability of Passenger Cars Manufactured for Indian Conditions. *J Adv Res Prod Ind Eng* 2017; 4(1&2): 3-7.

ISSN: 2456-429X

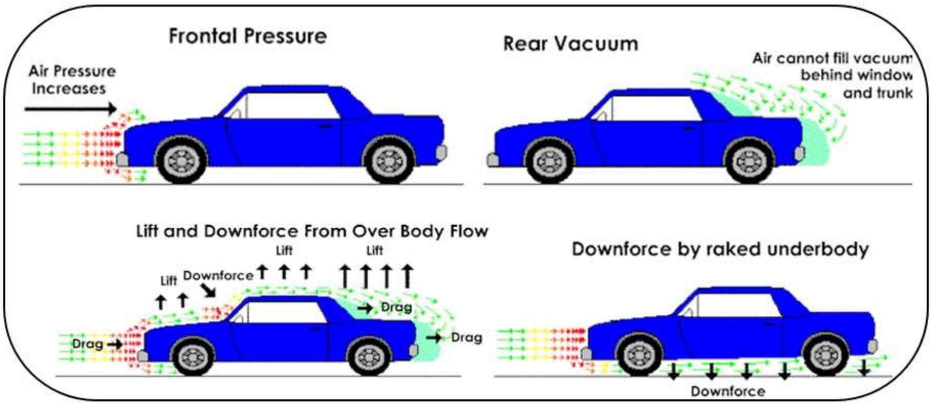

Figure 1.Aerodynamic Drag in Car

Lift is upward force on the body during moving of the body in fluid. Another term is down force.

Objectives of Study

The objective of the work is to compare aerodynamic stability of two different cars widely sold in India in year 2016-17 using CFD analysis. The software's GAMBIT and ANSYS FLUENT are employed for analysis of aerodynamics flow. Maruti Suziki Baleno and Hyundai Creta are the two Indian cars are selected for the study. Side contours of both car models are generated using design specifications of these cars to analyse that which car has better external shape in terms of drag or aerodynamic stability.

Methodology

There are two ways, as I have already stated. to analyze external shape any mechanical device: The wind tunnel and computational fluid dynamics (CFD). Due to financial limitation and better efficiency we will go for CFD. This analysis also gives us some visual pictures to understand fluid flow. We will use gambit software for designing and meshing, ansys fluent as solver and post processing. Following are the steps followed:

Step 1: The first step in this project is to create 2D models of both the cars. Both the models are prepared using software gambit with actual dimensions

Step 2: Mesh generation: After drawing the model the next step was to obtain mesh. Meshing consists of quads and triangular elements. Small meshing space was given near the contour to obtain accurate results.

Step 3: Specifying boundary conditions: The boundary conditions for this problem were the contour of car bodies were set as WALL, the upper and lower edges of wind tunnel were also set as WALL, the inlet of our virtual box is set as VELOCITY and outlet as PRESSURE OUTLET.

Step 4: Exporting the file in Ansys FLUENT: After specifying boundary conditions the problem was now to be solved in Ansys FLUENT. For this we exported the model files from GAMBIT to FLUENT by creating "**.msh**" file.

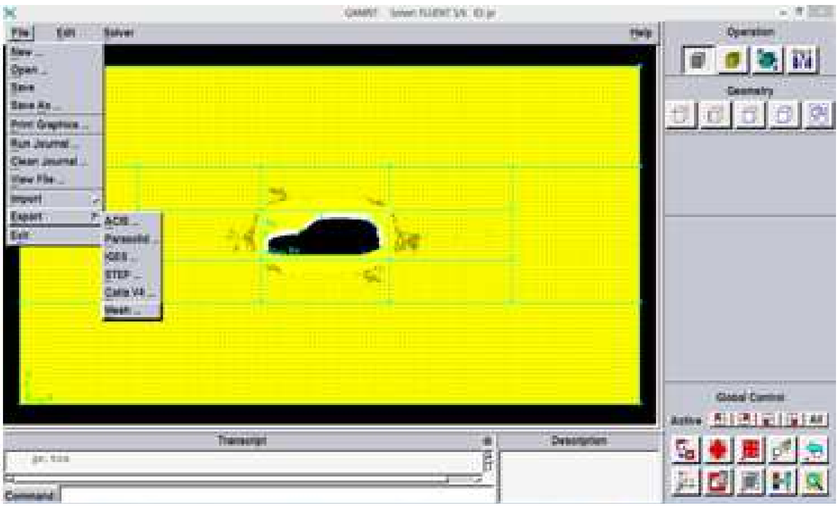

Figure-2.Car design with mesh generation and boundary condition

Step 5: Defining problem in FLUENT: The boundary conditions for the certain lines of domain were configured in GAMBIT. Velocity of the air at the inlet boundary condition is set in FLUENT with value of 100 km/h and with temperature of 300 K. The outlet boundary condition is set to pressure outlet with atmospheric pressure. Car contour, top and bottom of virtual wind tunnel is set as wall. The density of air is set as 1.225 kg/m3 and viscosity of air is 1.7894 10^{-5} N-s/m^2.

Results

Computational fluid dynamics simulation using *FLUENT* software was used to visualize airflow around the car geometry. CFD analysis achieved was used to analyse the critical places in geometry which result in unfavourable aerodynamics of moving car. Figure 3 and 4 shows the velocity contours of two Indian cars having different geometry. Similarly Figure 5 and 6 shows pressure contour of the same cars. Both these factors play vital role in car stability. Turbulence contour of these cars are also generated as shown Figure 7 and 8 as turbulence generally affects the "rear vacuum" portion of the car. The turbulence created by this detachment can then affect the air flow to parts of the car which lie behind the mirror.

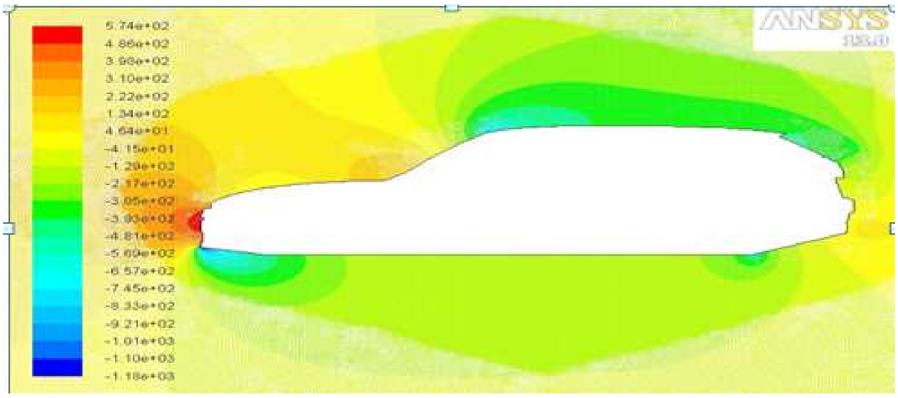

Figure 3.Velocity contour for CRETA

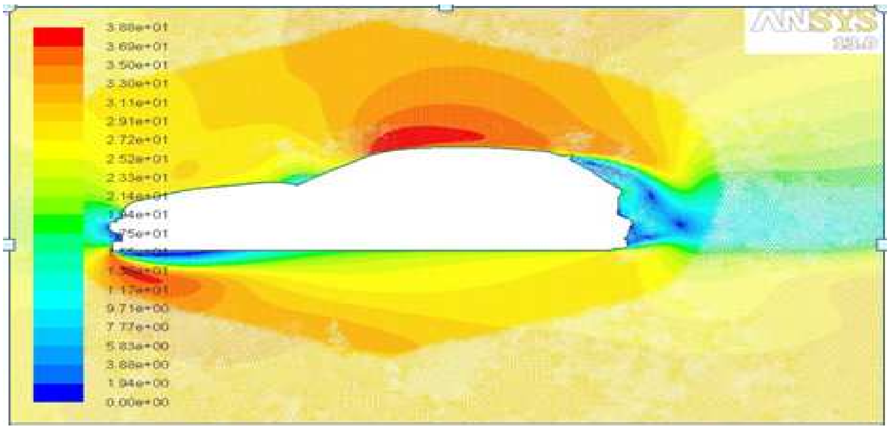

Figure 4.Velocity contour of BAlENO

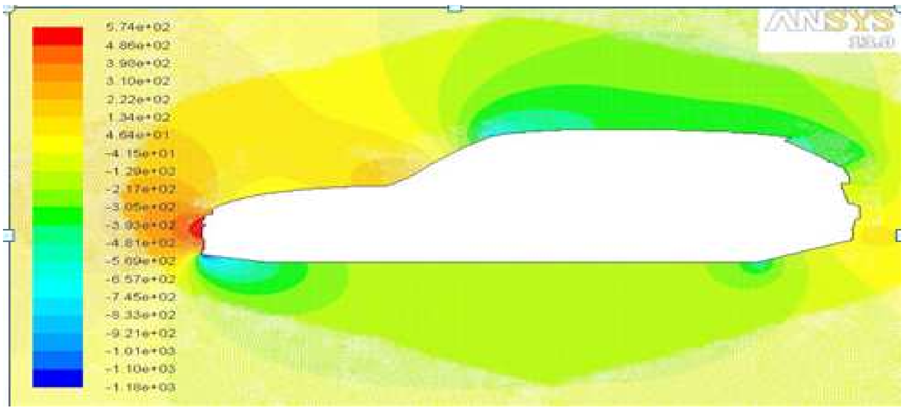

Figure 5.Pressure contour of CRETA

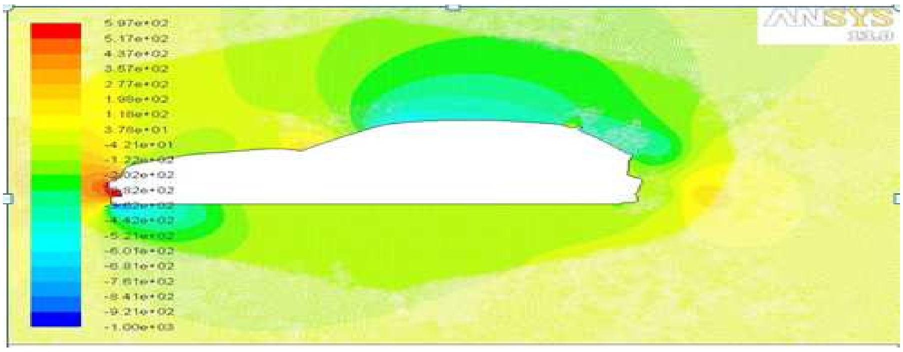

Figure 6.Pressure contour of BALENO

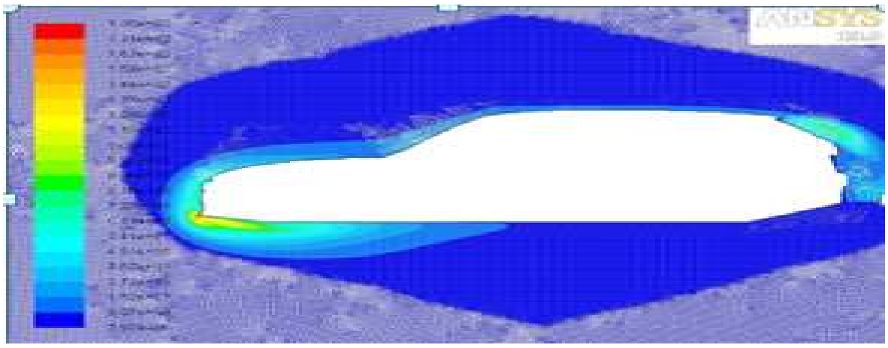

Figure 7.Turbulence contour of CRETA

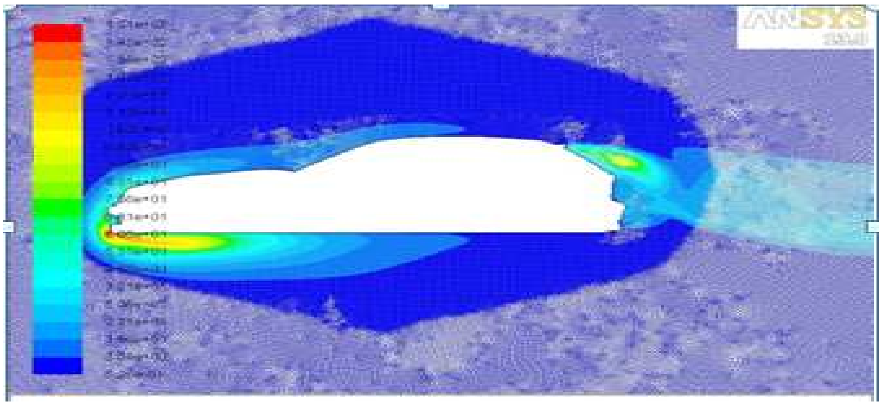

Figure 8.Turbulence contour of BALENO

Conclusion

Figure 3 and 4 show velocity contours. Air velocity than increases away from the car front. By comparing Figure 3 and 4, it can be seen that in case of contour plot of Baleno car (refer figure 4) the velocity magnitude increases with a higher gradient, which means that air resistance is smaller, and in case of Hundai Creta car (refer figure 3) it is increasing at low rate which means high aerodynamic drag in case of Hyundai Creta.

Figure 5 and 6 shows pressure contour .As the air flows over the car hood, pressure is decreasing, but when reaches the front windshield it briefly increasing. When the higher pressure air in front of the windshield travels over the windshield, it accelerates, causing the decreasing of pressure (see figure-5). This lower pressure literally produces a lift force on the car roof as the air passes over it. It is also shown that the baleno (figure-6) car geometry has smaller amount of pressure in front of car than that of Crete (figure-5).

Figure 7 and 8 shows turbulence intensity contours for both case of car geometry. It is seen that Turbulence intensity in front of the car in case of Baleno (figure 8) is less than that of Creta (figure 7).As it is presented from contour colors, it can be seen that in case of baleno there is less turbulences behind the car and turbulent zone is cleaner. As higher amount of down - force is results in better car stability and increases traction, thus from the analysis of velocity contours we conclude that aerodynamics of BALENO is better than that of CRETA.

The paper is in line with the thought process that the cars of future will respond to infrastructural constraints more proactively. Be it to meet road safety, flexibility & parking ease, or simply for responding to SHEQ [12].

Future Scope

With further advancement in computer capability of computers we can also analyze the 3-D car models within stipulated time. Also more multiple number/ types of different cars can be compared in similar manner.

References

1. Akhai, S. and Pandita, P., 2013. Design Optimization of Failure Mode for Improving Quality Control on the Engine Assembly Line. *International Journal of Emerging Technology and Advanced Engineering, 3*(12), pp. 201-204, ISSN 2250-2459.
2. Chowdhury, H., Moria, H., Ali, A., Khan, I., Alam, F. and Watkins, S., 2013. A study on aerodynamic drag of a semi-trailer truck. *Procedia Engineering*, *56*, pp.201-205.
3. Damjanović, D., Kozak, D., Živić, M., Ivandić, Ž. and Baškarić, T., 2011. CFD analysis of concept car in order to improve aerodynamics. *A JOVO jarmuve, A MAGYAR JÁRMÛIPAR TUDOMÁNYOS* LAPJA, 1(2), pp.63-70.
4. Thareja P, Sharma R. Cloud-Hosted Advanced Manufacturing and Assembly. J Adv Res Prod Ind Eng 2017; 4(1): 1-11. Digital Object Identifier (DOI): 10.24321/2456-429X.201701
5. Hetawal, S., Gophane, M., Ajay, B.K. and Mukkamala, Y., 2014. Aerodynamic Study of Formula SAE Car. *Procedia Engineering*, *97*, pp.1198-1207.
6. Kanda, R., Akhai, S. and Bansal, R., 2013. Analysis of MOST technique for elimination of ideal time by synchronization of different lines. *International Journal of Research in Advent Technology, 1*(4), pp. 151-158, E-ISSN: 2321–9637.
7. Kieffer, W., Moujaes, S. and Armbya, N., 2006. CFD study of section characteristics of Formula Mazda race car wings. *Mathematical and Computer Modelling*, *43*(11), pp.1275-1287.
8. Niu, J.Q., Zhou, D. and Liang, X.F., 2017. Experimental research on the aerodynamic characteristics of a high-speed train under different turbulence conditions. *Experimental Thermal and Fluid Science*, *80*, pp.117-125.
9. Thakur, S., Goyal,M. And Akhai,S., 2016. A computational and experimental investigation on thermal conductivity of fibre reinforced ceramic composite for high temperature applications J. Adv. Res. Prod. Ind. Eng. 2016; 3(3&4)
10. Thareja. Priyavrat; SHEQ to Shake the Car Development for Tomorrow-Part I. Trends in Mechanical Engineering & Technology. 2015; 5(1), 6–19 p.. Available at SSRN: https://ssrn.com/ abstract=2576839
11. Tsai, C.H., Fu, L.M., Tai, C.H., Huang, Y.L. and Leong, J.C., 2009. Computational aero-acoustic analysis of a passenger car with a rear spoiler. *Applied Mathematical Modelling*, *33*(9), pp.3661-3673.
12. Thareja. Priyavrat; SHEQ to Shake the Car Development for Tomorrow-Part II. Trends in Mechanical Engineering & Technology.2015. Vol 5(2).