© 2018 IJSRSET | Volume 4 | Issue 4| Print ISSN: 2395-1990 | Online ISSN : 2394-4099
Themed Section: Engineering and Technology

CFD Analysis on Aerodynamics of an Automobile Body

Deep Patel[1], Jitendra Jangid[1], Yash Devani[1], Saurabh Goswami[1], Milan Vachhani[2]

[1]UG Student, Mechanical Engineering Department, ITM Universe, Vadodara, Gujarat, India

[2]Assistant Professor, Mechanical Engineering Department, ITM Universe, Vadodara, Gujarat, India

ABSTRACT

Vehicle Aerodynamics plays a crucial role and parameters such as drag as well as lift affect the performance of vehicle running at high speed. The various forces acting on a vehicle, like drag force, lift force, weight, side forces, and thrust and all these forces affect the fuel consumption rate of the car. The production of drag force and down force in automotive body are determined by analysis of air flow around it. A generic model of a sedan car and a wing type spoiler was designed in CAD software (Solid works) and performed CFD (Computational fluid dynamics) analysis on the car with and without the spoiler in the ANSYS-CFX and determined the drag coefficient and lift coefficient acting on the generic model of the sedan car. On adding the spoiler, it helps to increase the down force, which ultimately results in the stability of the vehicle and reduced fuel consumption.

Keywords: CFD, Aerodynamics, Automobile body, drag coefficient, lift coefficient, ANSYS-CFX

I. INTRODUCTION

In today's world, the demand of high speed cars is very huge with enhanced stability being a great concern. Aerodynamics, a branch of dynamics, is the study of the properties of air in motion and its interactions with the solid bodies. Aerodynamics is a subfield of fluid dynamics and gas dynamics and a lot of laws and theories are shared between these two branches of dynamics. The production of lift and drag forces in automotive body are determined by analysis of air flow around it. The friction between the vehicle and the air in motion around it, is called the drag force. About 60% of the total drag force is produced at the rear end of the vehicle. The automobile body in motion experiences a lift force i.e. the perpendicular component of the force exerted on the vehicle body by the oncoming flow of air.

In this paper, the attempt to improve the fuel utilization by improving the aerodynamic structure of the car on adding generic model of wing type spoiler at the rear end, which helps to increase down force acting on it and ultimately increases the stability of the vehicle.

A generic model of sedan car with and without spoiler, is used to determine the optimum aerodynamic Structure. SOLID WORKS is used for designing the sedan car as well as the wing type spoiler and ANSYS-CFX is used for the CFD analysis. For finding the aerodynamic drag and lift coefficient at different velocities, the analysis was performed. To obtain the air flow structure around a passenger car with a rear spoiler, this study was carried out on the numerical model based computational fluid dynamics (CFD).

CFD (Computational fluid dynamics) a branch of fluid dynamics is a computational science of simulating fluid flow using computers.

Steps involved in CFD analysis:

1. Geometry Definition

2. Grid Generation
3. Flow Algorithm
4. Flow Simulation
5. Visualization

"CFD (Computational fluid dynamics) is a set of numerical methods applied to obtain approximate solution of problems of fluid dynamics and heat transfer." [1]

The below figure (Figure 1) shows the different disciplines contained within CFD:

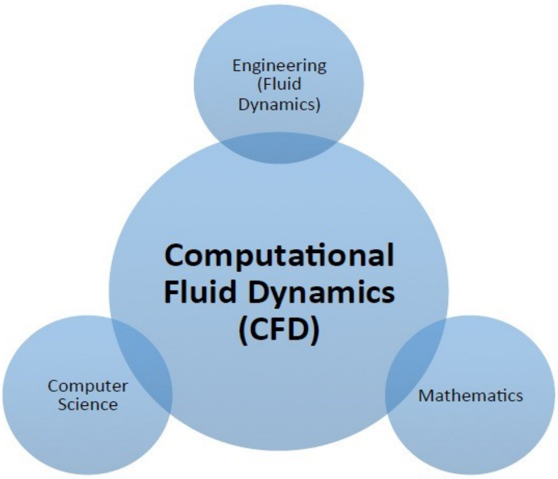

Figure 1. Different disciplines contained within CFD

CFD is not a science by itself but a way to apply methods of one discipline (numerical analysis) to another (Fluid dynamics). In retrospect, it is integrating not only the disciplines of fluid mechanics with mathematics but also with computer science as illustrated in Figure 1. The physical characteristics of the fluid motion can usually be described through fundamental mathematical equations, usually in partial differential form, which govern a process of interest and are often called governing equations in CFD.

II. METHODOLOGY

A. Designing of generic model of sedan car and the wing type spoiler

With the help of the CAD software (SOLIDWORKS), the generic model of a sedan car and the wing type spoiler was designed. By geometric modification, the spoiler was added on to the rear end of the vehicle. Following are the dimension of the generic model of sedan car:

Length (L) = 4.865 m
Width (W)= 1.916 m
Height (H)= 1.381 m

The frontal area of the car was calculated in SOLIDWORKS as 2.317 m^2 and it will be used later on to determine the lift coefficient and drag coefficient.

The below figure (Figure 2) shows the generic model of a sedan car without spoiler:

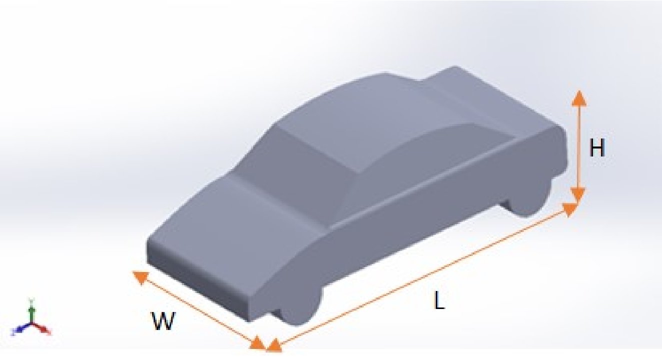

Figure 2. Generic model of a sedan car without spoiler

The below figure (Figure 3) shows the generic model of a wing type spoiler:

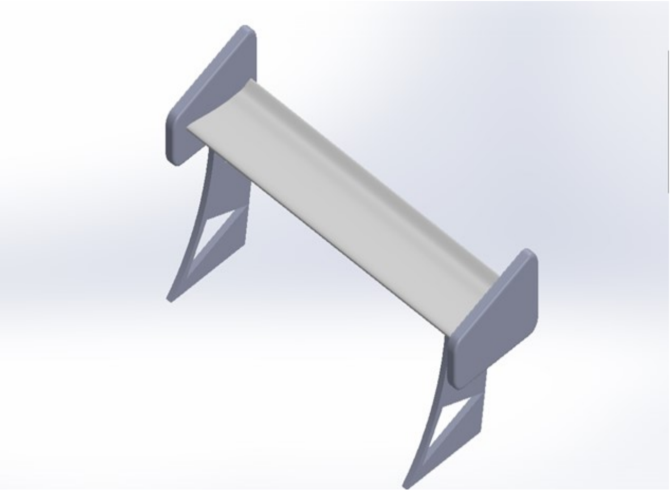

Figure 3. Generic model of a wing type spoiler

The below figure (Figure 4) shows the generic model of a sedan car with a wing type spoiler:

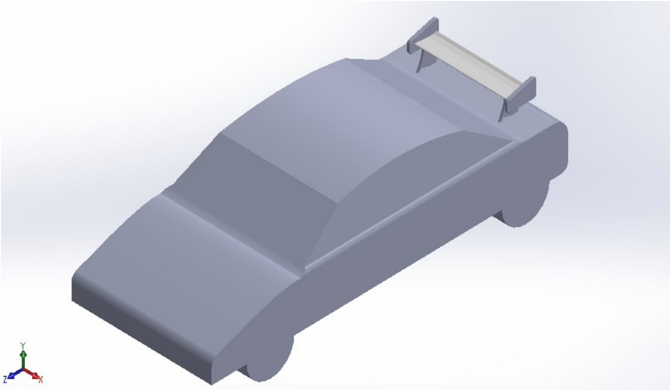

Figure 4. Generic model of a sedan car with wing type spoiler

The below figure (Figure 5) shows the distance at which the spoiler is being placed at the rear side of the car:

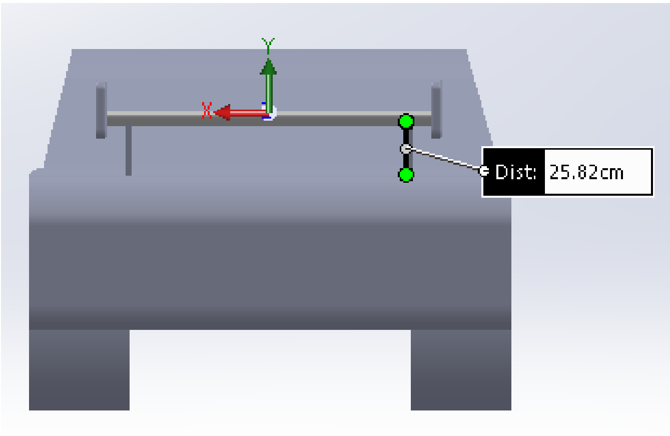

Figure 5. Distance of the generic model of a wing type spoiler at the rear side

The below figure (Figure 6) shows the vehicle orientation in the virtual wind tunnel:

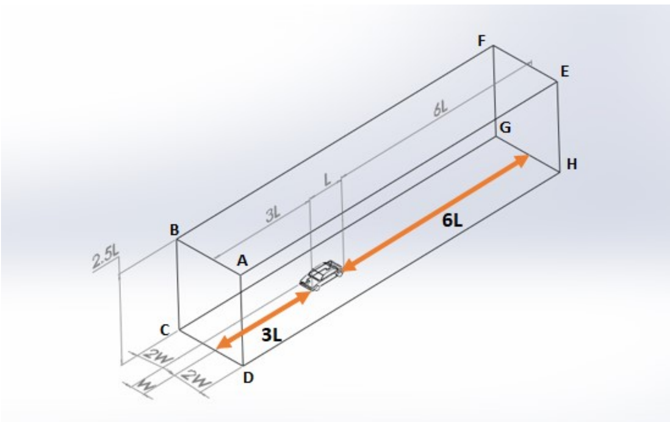

Figure 6. Vehicle orientation in the virtual wind tunnel

The virtual wind tunnel is created around the car in the ANSYS-CFX module to perform the analysis. The sedan car is placed in the virtual wind tunnel in such a way that the distance from the rear end of the car to the virtual wind tunnel is more as compared to the front end of the vehicle. This is done because, it is more important to capture the behaviour of the flow at the rear end of the vehicle where the "wake of vehicle" phenomenon occurs.

The length of the virtual wind tunnel is kept in accordance with the length of the car i.e. Length of the car = L, then the distance from starting of the virtual box to the front end of the car is 3L whereas from the rear end of the car to the end of the virtual box is 6L.

B. Boundary Conditions:

Table1. Boundary Conditions given to the virtual wind tunnel

Sr. No.	Plane	Zone name	Boundary type
1	ABCD	Inlet	Velocity inlet $V = \vec{V}$
2	EFGH	Outlet	Pressure outlet $P_{gauge} = 0$
3	ADHE, BCGF	Sides	Free stream $V = \vec{V}$
4	ABFE	Top	Free stream $V = \vec{V}$
5	CDHG	Road	Wall $V = 0$
6	-	Car	Wall $V = 0$

III. RESULTS AND DISCUSSION

A. Pressure Contour:

The below figure (Figure 7) shows the pressure contour over the vehicle. It is observed that the maximum pressure is exerted on the front end of the vehicle at bottom of the windshield and in front of hood because of sharp edge.

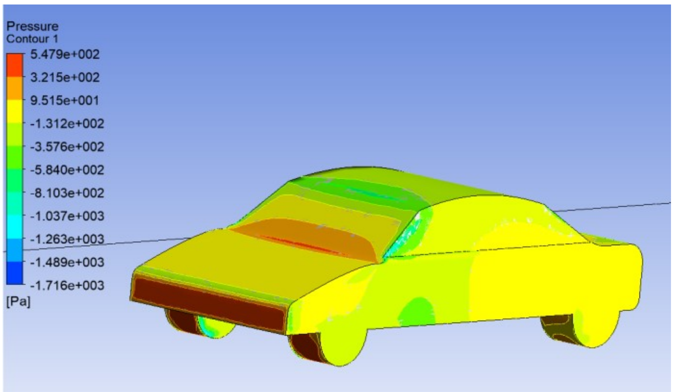

Figure 7. Pressure contour

B. Velocity Streamlines:

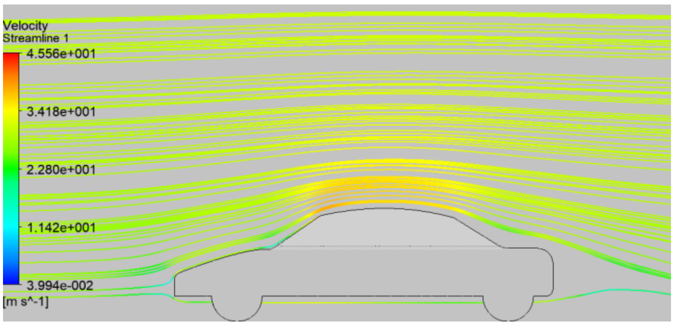

Figure 8. Velocity streamlines

C. Velocity streamlines for car without spoiler:

The below figures (Figure 9,10,11) shows the velocity streamlines on the car at 3 different speeds i.e. 20m/s, 25m/s and 30m/s.

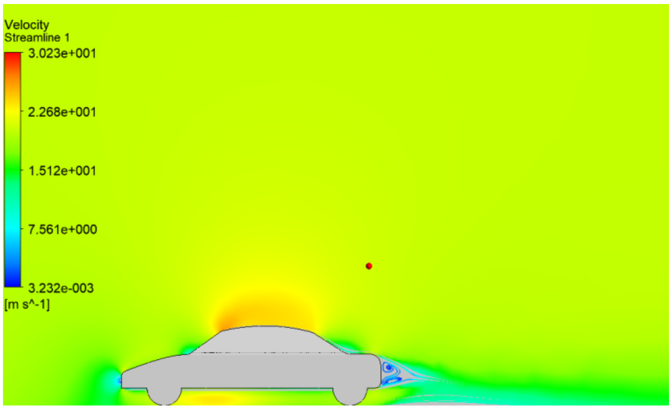

Figure 9. Velocity and magnitude [20 m/s]

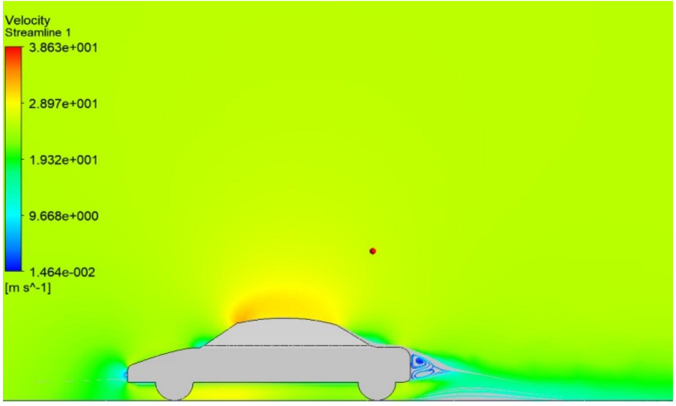

Figure 10. Velocity and magnitude [25 m/s]

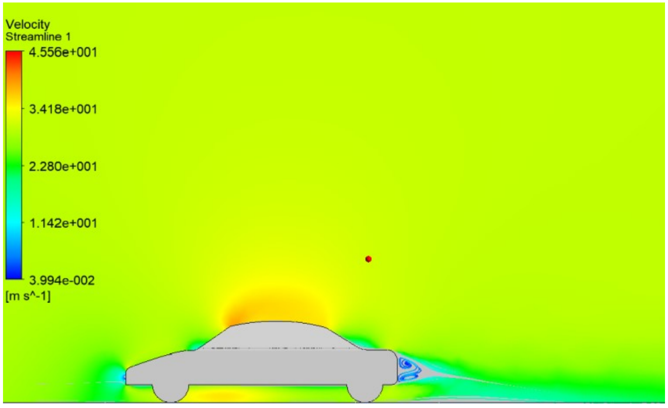

Figure 11. Velocity and magnitude [30 m/s]

D. Velocity streamlines for car with spoiler:

The below figures (Figure 12,13,14) shows the velocity streamlines on the car at 3 different speeds i.e. 20m/s, 25m/s and 30m/s.

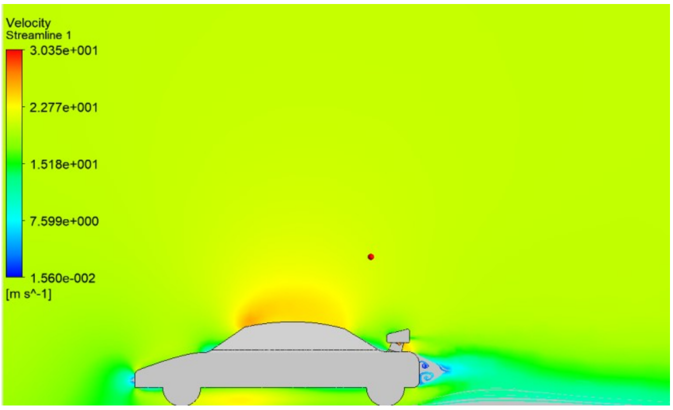

Figure 12. Velocity and magnitude [20 m/s]

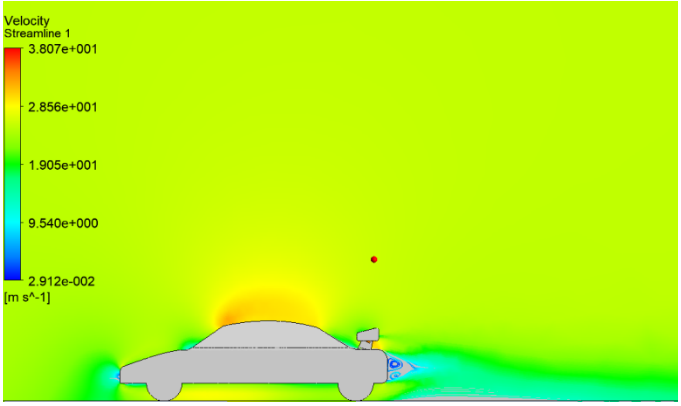

Figure 13. Velocity and magnitude [25 m/s]

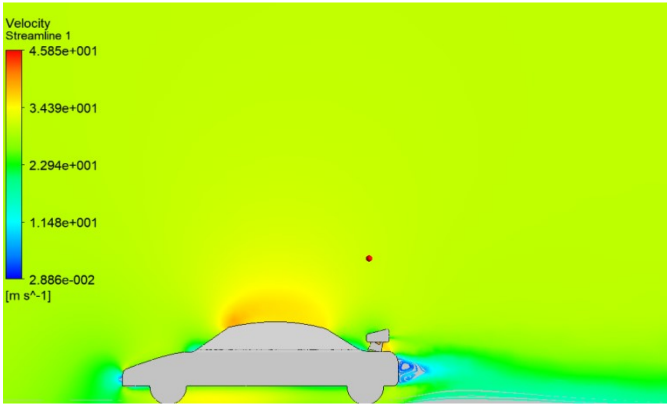

Figure 14. Velocity and magnitude [30 m/s]

E. Values of C_L and C_D for the Sedan car with and without spoiler:

Table 2

Sr. No.	Velocity (m/s)	Without spoiler		With spoiler	
		C_D	C_L	C_D	C_L
1	20	0.425	0.565	0.479	0.503
2	25	0.437	0.554	0.480	0.484
3	30	0.449	0.527	0.473	0.439

IV. CONCLUSION

On performing the CFD analysis in ANSYS-CFX on generic model of the automobile body (sedan car) with and without the wing type spoiler at 3 different speeds i.e. 20 m/s, 25 m/s and 30 m/s, the lift and drag coefficients acting on it were determined. On adding the spoiler, it helped to increase the downforce and decrease the lift force which ultimately helps to increase the stability of the vehicle.

V. REFERENCES

[1]. Investigation of Drag and Lift Forces over the Profile of Car with Rearspoiler Using CFD by V. Naveen Kumar, Dr. K. Lalit Narayan, L. N. V. Narasimha Rao, Y. Sri Ram http://ssjournals.com/index.php/ijasr/article/download/2510/1961

[2]. Aerodynamic analysis of a car model using Fluent-ANSYS 14.5 by Akshay Parab, Ammar Sakarwala, Bhushan Paste, Vaibhav Patil and Amol Mangrulkar http://www.ijrmee.org/download/aerodynamic-analysis-of-a-car-model-using-1429686382.pdf

[3]. Aerodynamic Design of F1 and Normal Cars and Their Effect on Performance by Shobhit Senger and S.D. Rahul Bhardwaj https://www.ripublication.com/iraer-spl/iraerv4n4spl_11.pdf

[4]. CFD analysis of aerodynamic drag reduction and improve fuel economy by J. Abinesh and J. Arun kumar http://www.ijmerr.com/uploadfile/2015/0409/20150409041757838.pdf

[5]. Automotive computational fluid dynamic simulation of a car using ANSYS by Praveen Padagannavar and Manohara Bheemanna http://www.iaeme.com/MasterAdmin/UploadFolder/IJMET_07_02_013/IJMET_07_02_013.pdf

[6]. Design and analysis of a car body using CFD software by Stjepan Galambos and Jovan Doric

[7]. Aerodynamic shape optimization based on free-form deformation.AIAA, 4630 Samareh, J. A.

[8]. "A Comparitive Assessment of two Experimental Methods for Aerodynamic performance Evaluation of a Car" Journal of Scientific & Industrial Research2008 by Manan Desai1, et all

[9]. Some innovative concepts for a car reduction: Parametric analysis of aerodynamic forces on simplified body by Mahmoud Khaled,Hischam.,Fabien. , and Hassan P.

[10]. Experimental and Computational Investigation of Ahmed Body for Ground Vehicle Aerodynamics. SAE Technical Paper 2001-01-2742. By Bayraktar, I., Landman, D., and Baysal, O.

[11]. An Aerodynamics Study of the Cockpit. School of Engineering.Cranfield University by Luca Iaccarino.

[12]. Simulation of the Ahmed Body 9th ERCOFACT/IAHR Workshop on Refined Turbulence Modelling by Braus, M., Lanfrit, M.

[13]. The effect of the vehicle spacing on the aerodynamics of a representative vehicle,Journal of Wind Engineering and Industrial Aerodynamics96 by Simon Watkins, Gioacchino Vino .

[14]. A review on CFD analysis of drag reduction of a generic sedan and hatchback by Shashikant, Desh Deepak Srivastava, Rishabh Shankar, Raj Kunwar Singh, Abhiral Sachan https://www.irjet.net/archives/V4/i5/IRJET-V4I5189.pdf