e-ISSN: 2582-5208
International Research Journal of Modernization in Engineering Technology and Science
(Peer-Reviewed, Open Access, Fully Refereed International Journal)
Volume:03/Issue:09/September-2021 Impact Factor- 6.752 www.irjmets.com

CFD ANALYSIS & AERODYNAMIC STUDY OF A SEDAN CAR

Danish Shaikh[*1]

[*1]B.E Student, Department Of Mechanical Engineering, M.H. Saboo Siddik College Of Engineering, Mumbai University, Mumbai, Maharashtra, India.

ABSTRACT

Aerodynamics is an important part of automobile design. A vehicle with better aerodynamics not only gives better output but also proves to be economic. Research and development is been carried out on a large scale to obtain lower coefficient of drag and lift to get better performance. This paper presents the air flow over a sedan car with the help of Computational Fluid Dynamics. The analysis aims at calculating the aerodynamic drag & pressure and improving the air flow over the sedan body at a realistic speed. The analysis was performed on Ansys Fluent and the designing of the 3D car model was done in Solidworks.

Keywords: CFD (Computational Fluid Analysis), Aerodynamics, Drag Force, Sedan Car.

I. INTRODUCTION

Automotive Aerodynamics is the study of aerodynamics of the vehicle. The main purpose of automotive aerodynamics is to reduce drag, prevent undesired lift force and instability at high speeds. An aerodynamically well designed body contributes in improving various factor for a vehicle such as reducing fuel consumption, improving of the driving characteristics by providing better handling and stability as well as comfort characteristics by reduction in the noise and better ventilation & cooling.

The 4 forces of Aerodynamics are: Drag, Lift, Thrust & Weight:

- Drag: Aerodynamic drag is the force opposing the vehicles direction of movement.
- Lift: Lift is force acting perpendicular to the motion of the vehicle.
- Thrust: Thrust is a force used in aircrafts it moves an aircraft in the direction of the motion.
- Weight is the force caused by gravity.

The automotive aerodynamics focuses on the drag and lift as the two major forces.

Drag: Drag is the aerodynamic force that opposes a vehicle's motion through the air. It is generated by the interaction of a solid body with air or fluid.

The coefficient of drag C_d can be calculated using:

$$C_D = \frac{Drag\ force}{(\rho V^2 A)/2}$$

Lift: Lift is the force that directly opposes the weight of a car. Lift occurs when a moving flow of gas is turned by a solid object. The flow is turned in one direction, and the lift is generated in the opposite direction, as per Newton's Third Law of action and reaction.

The coefficient of lift C_L can be calculated using:

$$C_L = \frac{Lift\ force}{(\rho V^2 A)/2}$$

where ρ is the density of air, V is the velocity of vehicle and A is the frontal area of sedan car.

Computational Fluid Dynamics: Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and data structures to analyze and solve problems that involve fluid flows.

Streamline: A steady flow of fluid without any turbulent fluctations in its velocity is called streamline flow. The streamline flow analysis on various bodies can be done by wind tunnel set-ups or computationally via CFD softwares.

II. METHODOLOGY

Computational method was used to perform analysis on the designed 3d model and analysis was performed in Ansys Fluent. The overview of the steps of Design & CFD methodology are as follows:

- Creating a 3D model.
- Giving the geometric conditions in Design Modeller.

e-ISSN: 2582-5208
International Research Journal of Modernization in Engineering Technology and Science
(Peer-Reviewed, Open Access, Fully Refereed International Journal)
Volume:03/Issue:09/September-2021 Impact Factor- 6.752 www.irjmets.com

- Meshing.
- Set up the physics and solver settings.
- Calculating the solutions.
- Obtaining the results.

III. MODELING AND ANALYSIS

3D Model of the sedan car body : The model was designed in Solidworks Software taking inspiration from the modern day sedan cars. After creating the model it was exported to Ansys for the further aerodynamic analysis.

For performing the aerodynamic analysis the geometry was opened in Ansys Design Modeller and an enclosure was created. The inlet, outlet and the boundary walls were nammed as per the flow conditions needed.

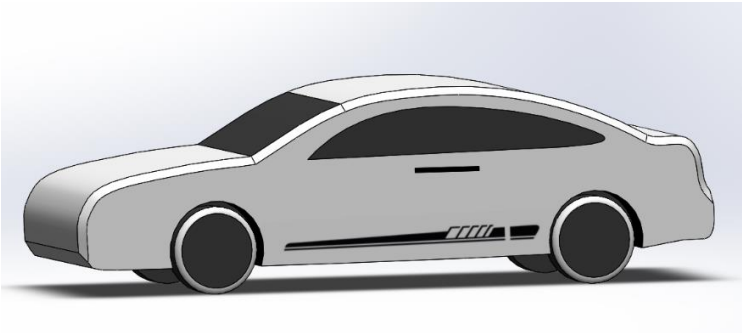

Figure 1: 3D Model of the Sedan Car

Meshing : A mesh is representation of larger geometric domain by smaller discrete cells. The higher the mesh quality the better & accurate results we obtain. For better meshing the mesh resolution was kept fine. The physics preference for mesh was set to CFD and the element order as linear.

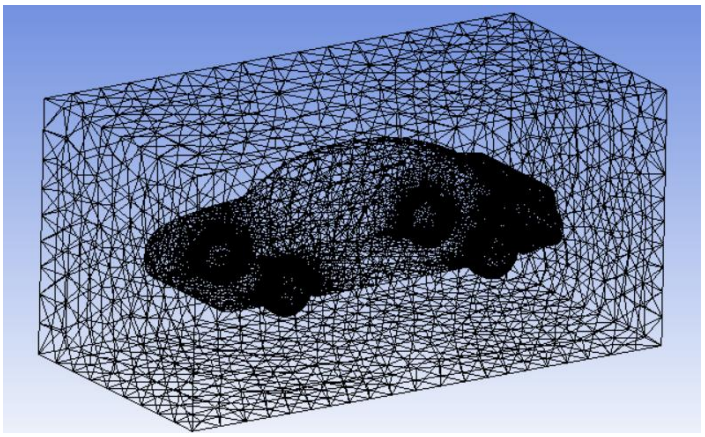

Figure 2: Wireframe view of the Mesh Model

Setting up the Physics & Solver Settings: This is the important part where all the operating conditions, boundary conditions and initial values are given to the model as per the analysis required. This step takes the longest time during the analysis and depending on the mesh quality and system configuration the amount of time varies largely. The higher the mesh quality the more time it will take to solve the problem as it solves the entire model for each and every element. In the solver setup for our analysis we used Pressure based Steady state analysis and the numerical model used was viscous laminar. The material used was air as the material through which the sedan car is passing is the air.

Boundary conditions: The boundary conditions are given to the model to obtain the required results. The initial boundary condition for our model are as follows:

- Initial Velocity : 80km/hr (22.22 m/s)
- Walls are kept stationary.
- Material medium : Air

Density: 1.225 kg/m^3 Viscocity: 1.7894e-05

e-ISSN: 2582-5208
International Research Journal of Modernization in Engineering Technology and Science
(Peer-Reviewed, Open Access, Fully Refereed International Journal)
Volume:03/Issue:09/September-2021 Impact Factor- 6.752 www.irjmets.com

Solution method

- Pressure Velocity Coupling: Scheme Coupled

Spatial discretization

- Gradient: Least Square Cell Based
- Momentum: Second order upwind

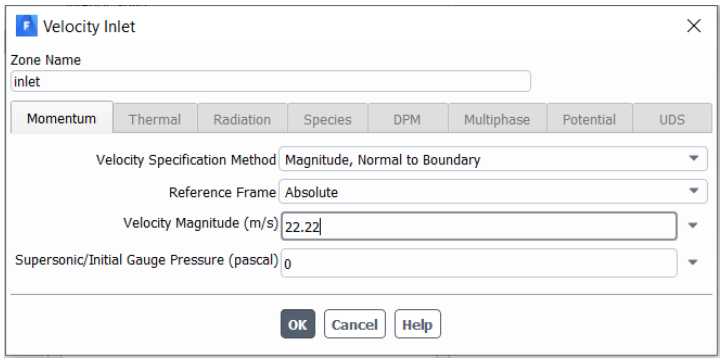

Figure 3: Inlet condition

After the boundary conditions we need to generate results for the drag report which can be obtained by clicking the Report definitions → New Report → Force → Drag

The axis for the drag report, the report output as Cd Drag coefficient and the graph plot was selected. Before the Final Calculation the Setup is Initialized using Hybrid Initialization.

For the Final calculation 100 iterations were solved as higher the iterations the more accurate value we obtain.

With the final calculation we complete the setup and the solving and obtain the results for our model.

IV. RESULTS AND DISCUSSION

We obtain the velocity magnitude vectors for our designed model from Graphics → Vectors → Display.

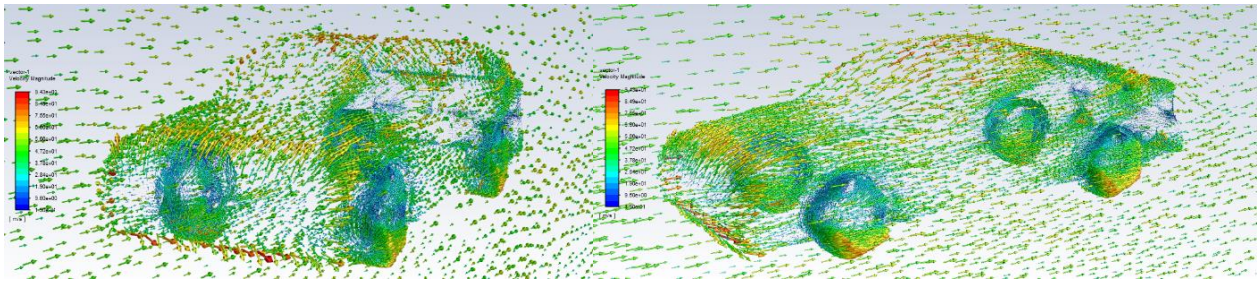

Figure 4: Velocity magnitude Vectors

The coefficient of drag was obtained as 0.76 after performing 100 iterations at wind speed of 80km/hr

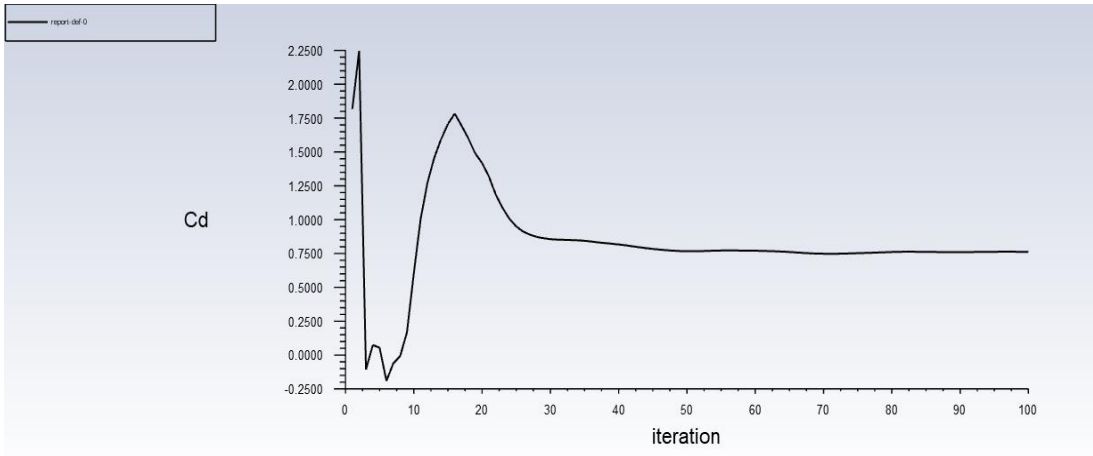

Figure 5: Graph plot of Coefficient of drag vs Number of Iterations.

e-ISSN: 2582-5208
International Research Journal of Modernization in Engineering Technology and Science
(Peer-Reviewed, Open Access, Fully Refereed International Journal)
Volume:03/Issue:09/September-2021 Impact Factor- 6.752 www.irjmets.com

A streamline flow over the sedan vehicle can be seen with no turbulent velocity fluctuations.

The Velocity streamline flow around the Sedan Body obtained shows that the velocity increases near the car surface.

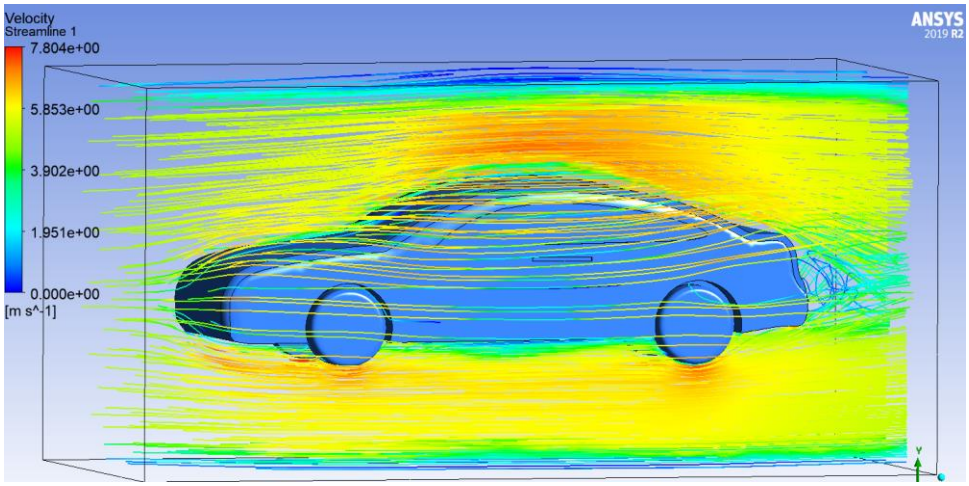

Figure 6: Streamline flow

The pressure distribution around the sedan car is given below at the front of the car the pressure can be seen as maximum.

Pressure at the front of the sedan vehicle was found to be 2.073e+03 Pa.

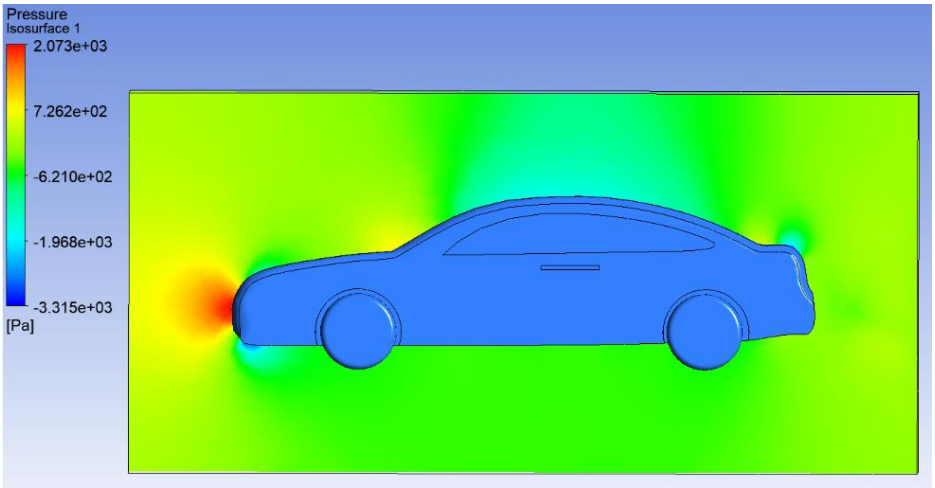

Figure 7: Pressure Distribution

Table 1: Values

Air Velocity	80km/hr
Drag Coefficient Cd	0.76
Front Pressure	2.073e+03Pa

V. CONCLUSION

The analysis in this research demonstrates the importance of CFD Analysis in the designing process. In this study we first started with the design of the 3D model with an aerodynamic perspective using Solidworks followed by discretization and then analyzing the streamlines and pressure distribution of this CAD model at realistic speed using Ansys. The results show a lower drag coefficient and high pressure on the frontal area. This lower aerodynamic drag of 0.76 leads to improvement in the performance and handling of the sedan car. Also the velocity streamline around the sedan car body showed that the velocity increased as it came near the

surface of the car and the high pressure on the frontal area of the car was a result of ram air pressure. With this we can say that, CFD Anlaysis & Aerodynamic Study in the design of any sedan vehicle plays a very major role.

VI. REFERENCES

[1] Sankar, Shanmugasundaram. (2018). CFD Analysis of Aerodynamics of Car. International Journal of Innovative Research in Science, Engineering and Technology. 7. 4689-4693.

[2] Ramya, P & Kumar, A & Moturi, Jaswanth & Ramanaiah, Nallu. (2015). Analysis of Flow over Passenger Cars using Computational Fluid Dynamics. International Journal of Engineering Trends and Technology. 29. 170-176. 10.14445/22315381/IJETT-V29P232.

[3] 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

[4] Galamboš, Stjepan & Dorić, Jovan. (2015). DESIGN AND ANALYSIS OF CAR BODY USING CFD SOFTWARE.