Ilkogretim Online - Elementary Education Online, 2021; Vol 20 (Issue 3): pp. 3206-3213
http://ilkogretim-online.org
doi: 10.17051/ilkonline.2021.03.332

Cfd Aerodynamic Analysis Of Two Different Cars

Sayed Mohammad Younus[1], Ishaan Bhasin[1], Maharishi Subhash[1], Rajesh P Verma[1,*], Dr B Deshpande[2]

[1]Department of Mechanical Engineering, Graphic Era Deemed to be University, Dehradun, 248002, Uttarakhand, India
[2] Professor, Department of Computer Science and Engineering, Graphic Era Hill University, Dehradun *rajesh_diva1@yahoo.in

Abstract: The drag coefficient (C_d) on different velocity of Mercedes Benz and Aventador is numerically examined. For the study of flow over 2D automobile using ANSYS Fluent, drag coefficient, pressure coefficient, static pressure, and velocity profile are used. The measurements are made at velocity ranging from 10m/s to 50m/s, and the effect of drag and pressure on the car body is observed. ANSYS Fluent is used to create a velocity profile on the top automobile body based on the provided velocities at 5 different places.

Keywords: Computational fluid dynamics, ANSYS Fluent, Drag Coefficient, velocity profile**.**

INTRODUCTION

Under normal operating conditions, the flow surrounding a road vehicle (car, bus, or truck) is primarily turbulent. Large-scale separation and recirculation zones, a complex wake flow, extended trailing vortices, and interaction of boundary layer flow on vehicle and ground are typical characteristics. The analysis was done to evaluate the fuel efficiency, vehicle stability and comfort of the passengers by Khan et al., [1]. The ANSYS Fluent software was used to calculate the air flow around the vehicle as well as the C_d values. For the evaluation of the turbulence, k-ε turbulent model was used as a standard model.

In his work; Damjanović et al., [2] employs the default rendering tool of Autodesk 3ds Max. The exterior design of the car is given priority, while the interior was not modelled. Furthermore, using the programme ANSYS Fluent, a 2D simulation of the airflow around the vehicle's side contour was performed in order to make adjustments to the geometry of the vehicle in order to optimise the design in terms of reducing air resistance and enhancing aerodynamics. The most emphasis was paid to adjusting the angle between the car's hood and front windscreen, as well as analysing the back of the car with and without the rear wing.

Velagapudi, N. K. et al. [3]; in his analysis lower aerodynamic drag force, Sedan cars with various sorts of spoilers are employed. The sedan car was designed in CATIA-2010 and is being analysed in ANSYS-2010 (fluent). The analysis was performed to determine drag and lift forces at various velocities, as well as spoilers. The flow structure surrounding a passenger automobile with a rear spoiler was obtained using an effective numerical model based on computational fluid dynamics (CFD).

Chaurasiya et al. [4] in his work use a commercial CFD solver to simulate C_d, coefficient of lift (C_L), flow separation, and vortex shedding over a vehicle and validate the simulation with an experimental result. The analysis was performed to determine C_d, C_L, and other flow properties with a flow velocity of 22.22 m/s on the Audi R8 automobile model. Solidworks2015 (CAD design), GAMBIT 2.4.6 (Meshing), ANSYS Fluent 17.0 (CFD Solver), and Tec plot 360 were utilised in this study (Post Processing).

In this our main focus is to compare the C_d at different velocity and to plot the velocity profile at different place over the car body using ANSYS Fluent. Further the paper is divided into sections; methodology gives a brief introduction of the turbulence model followed by result and conclusion.

METHODOLOGY

At flow inlets, velocity inlet boundary conditions are utilised to define the flow velocity as well as the essential scalar parameters of the flow. In ANSYS Fluent, these conditions are used to mimic a free stream condition at infinity, with free stream Mach No. and static conditions perfection. Wall boundary conditions are employed (fig. 1) to keep fluid and solid zones together. Based on the flow pressure (P) at the outlet, the pressure outlet boundary condition defines an outflow condition. This is typically utilised when a flow rate (or velocity) or higher pressure is assigned at the inlet. For the numerical modeling we are using ANSYS Fluent. Different turbulence models are available like Spalart–Allmaras , K-omega etc. [5].

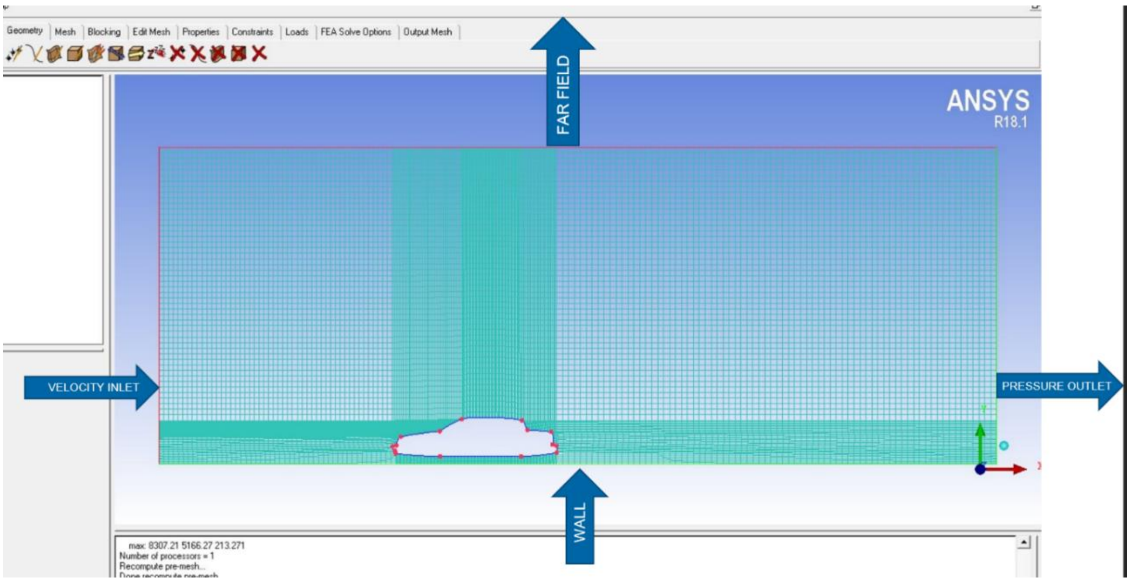
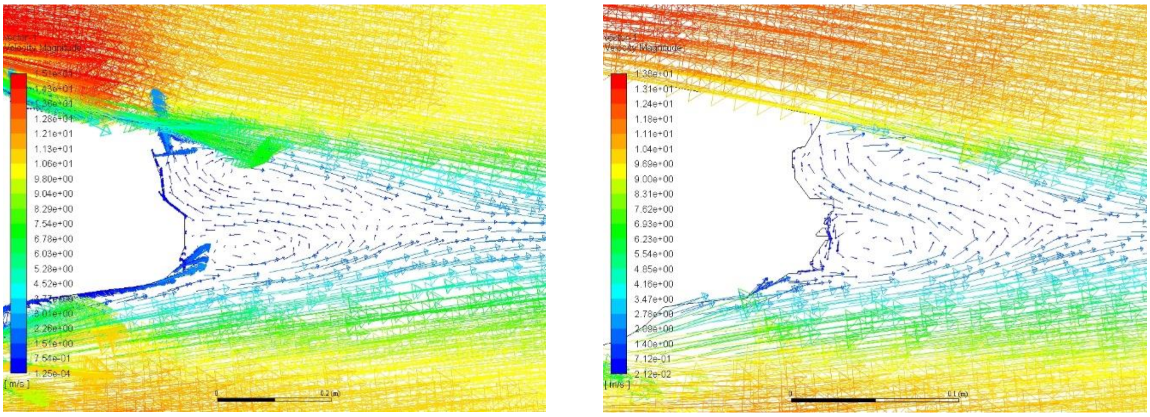
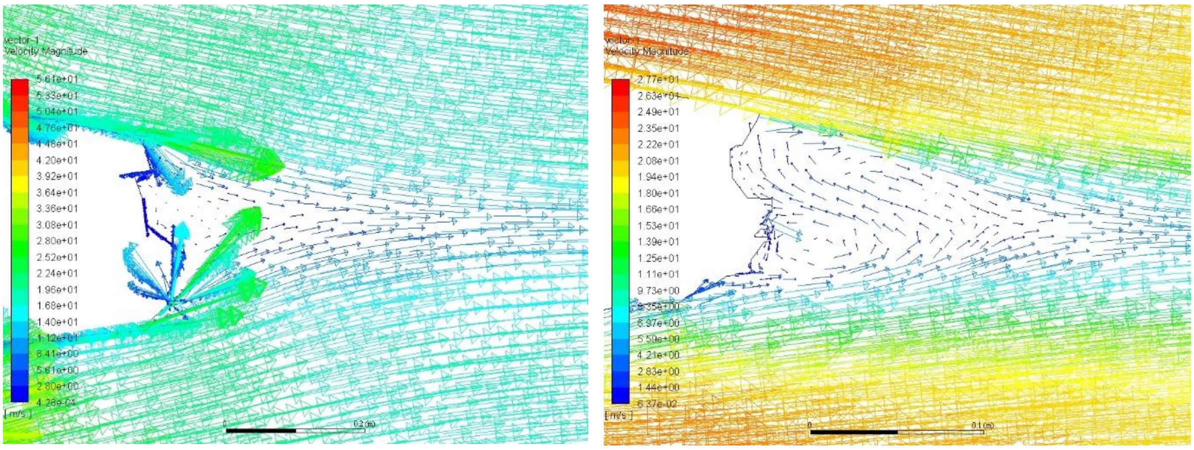


Figure 1. Boundary conditions employed for the problem.

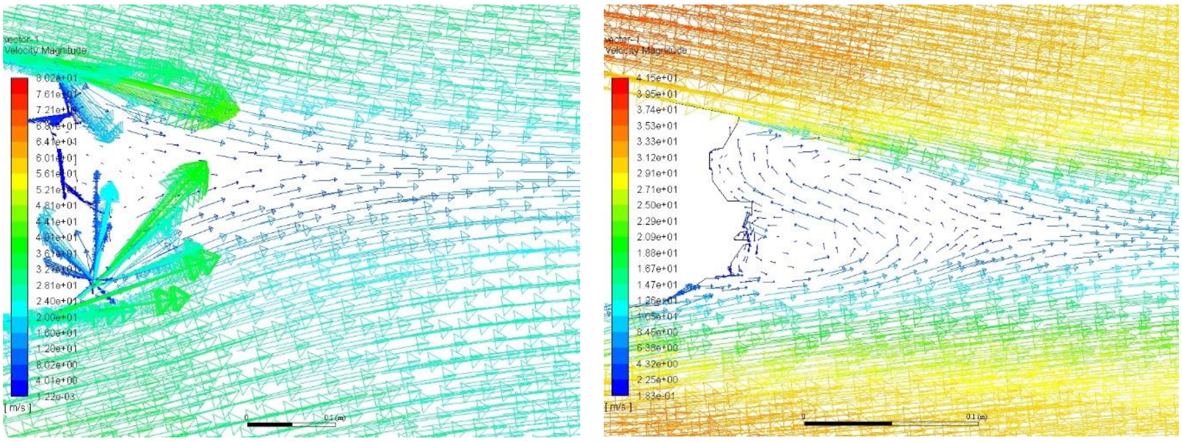
The standard car length is 1.2m, and the reference velocity ranges from 10 to 50m/s. We constructed a thin mesh near the body of the car since we are interested in determining the Cd on the surface body. While the Aventador's reference car length is 1.2m, the variation in velocity is the same as done in Mercedes. Velocity streamline shown in fig. 2 for the two cars at different velocity.



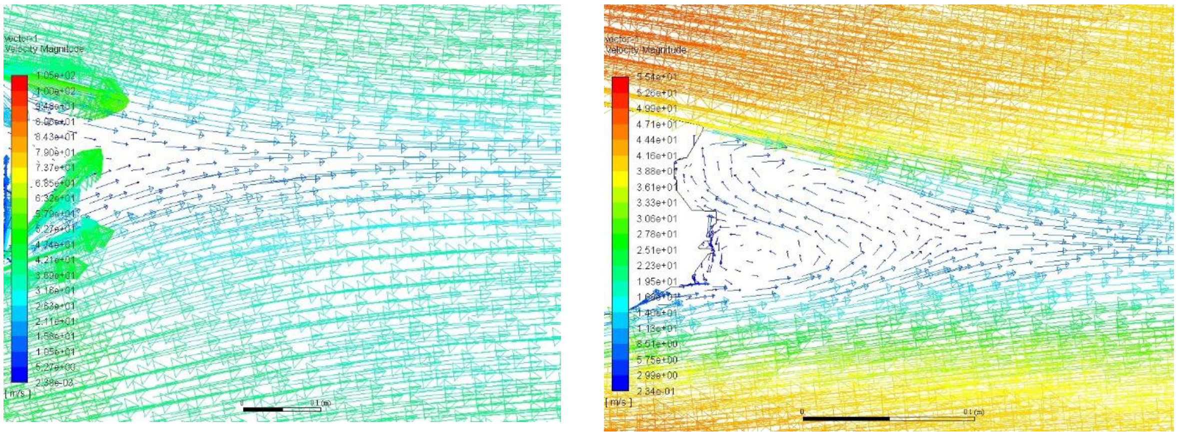
(a)



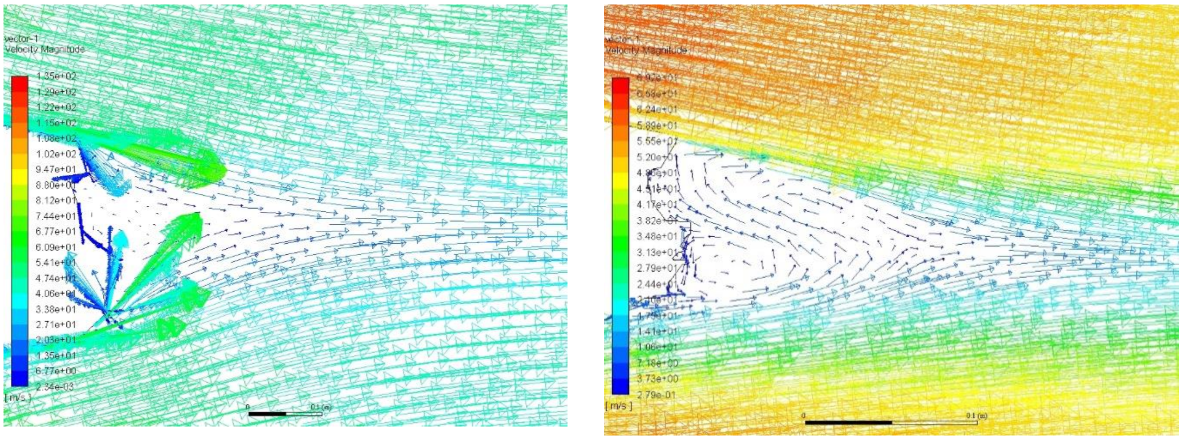
(b)



(c)



(d)



(e)

Figure 2. Streamline of the velocity for Mercedes (left) and Aventador (right) at (a) 10m/s, (b) 20m/s, (c) 30m/s (d) 40m/s, (e) 50m/s.

RESULTS AND DISCUSSION

We have observed that on increasing the velocity the values of the C_d increases for both the cars, as tabulated in the table 1. There is linear relationship between the C_d and the velocity for Aventador which is inferred from the fig. 3, while the same is not applicable for the Mercedes.

Table 1. Drag coefficients values at different velocities for the two cars

Cd	MERCEDES	AVENTADOR
10 m/s	2.86524	0.0381509
20 m/s	4.16487	0.084945
30 m/s	0.9537154	0.1500187

40 m/s	14.89144	0.2332939
50 m/s	-72.75336	0.2333678

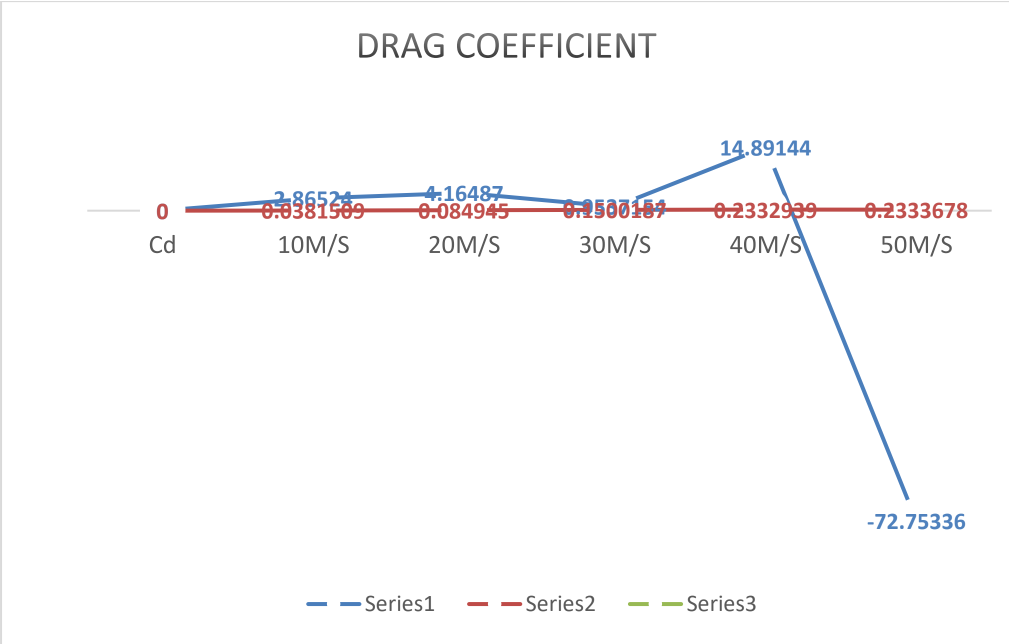
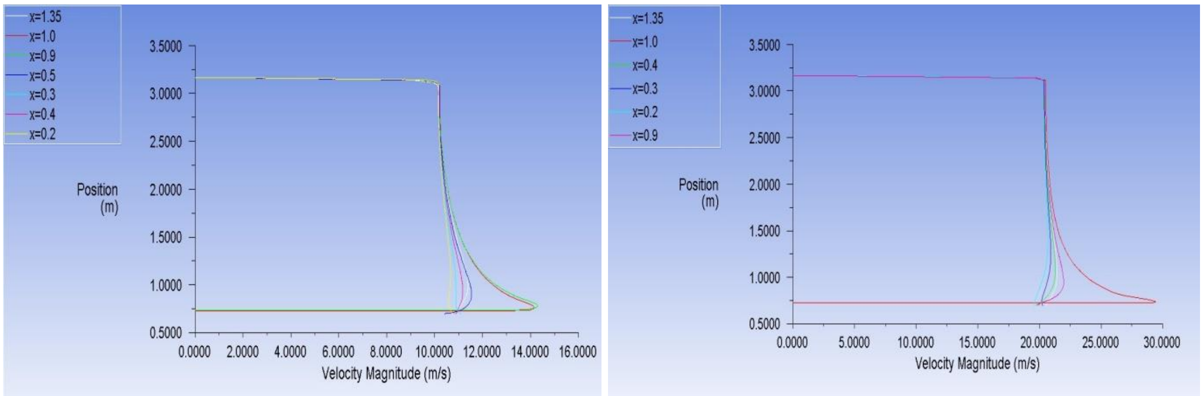
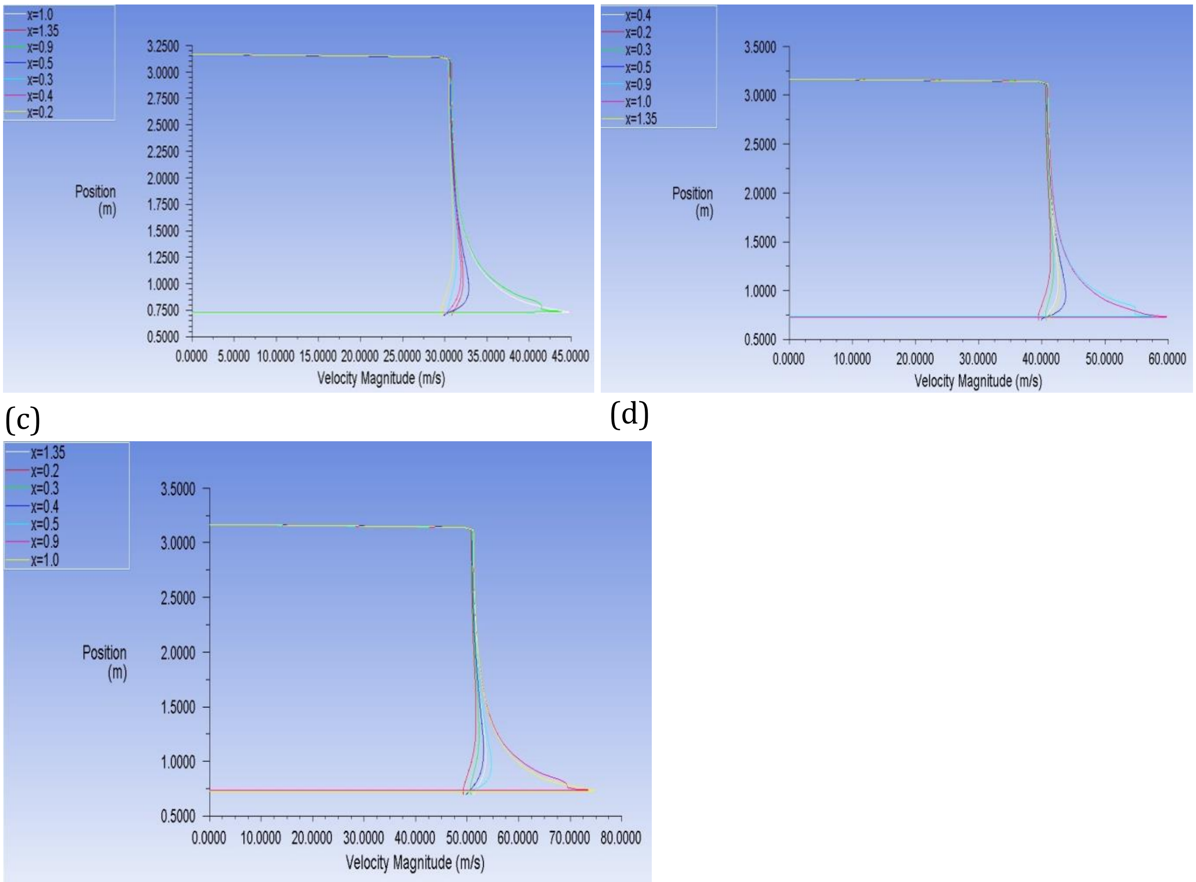


Figure 3 C_d vs velocity plot where series 1 and series 2 are for Mercedes and Aventador respectively.

The nature of viscosity demands a flow profile where velocity increases toward the centre of the tube under laminar flow conditions, as indicated. Figure 4 and 5 depicts the position versus velocity magnitude plot for Mercedes and Aventador at different velocity.



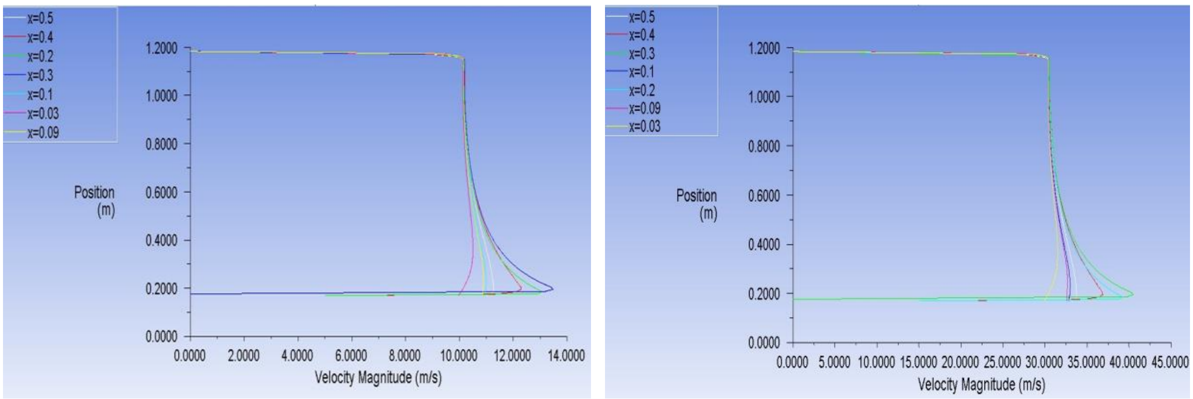
(a) (b)



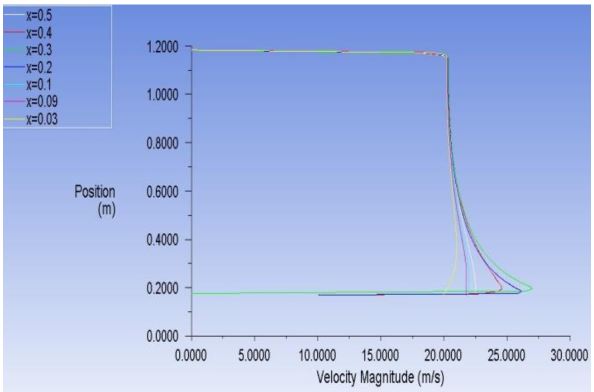
(c) (d)

(e)

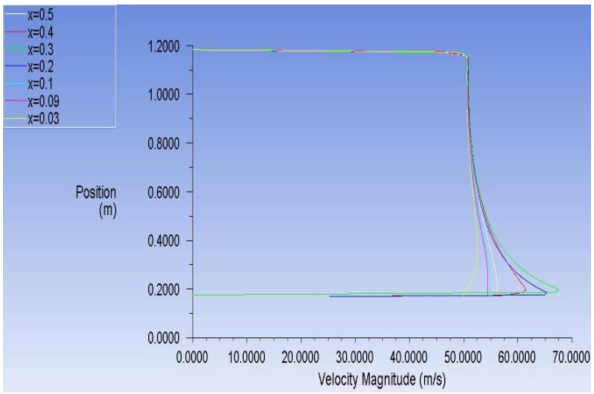
Figure 4 Mercedes Benz position versus velocity magnitude plot at: (a) 10 m/s, (b) 20m/s, (c) 30m/s, (d) 40m/s and (e) 50m/s



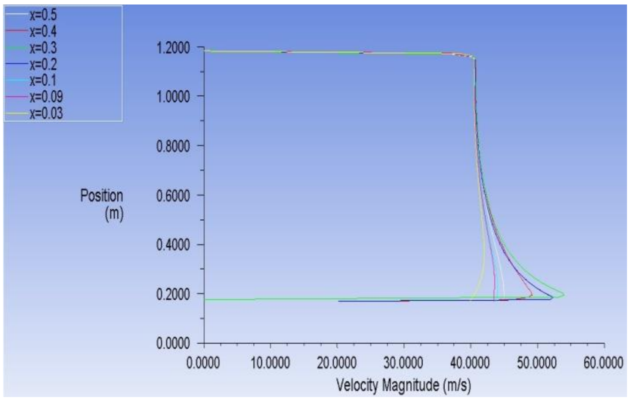
(a) (b)



(c)



(d)



(e)

Figure 5 Aventador position versus velocity magnitude plot at: (a) 10 m/s, (b) 20m/s, (c) 30m/s, (d) 40m/s and (e) 50m/s

CONCLUSION

From this study we get to know that how drag affects the automobile body and how to reduce drag to make the car more aerodynamically efficient. We took two cars, one sedan and one coupe, and discovered that the more the body of the car is streamlined, the better the performance and the more fuel efficient it is. We validated the CFD code by comparing its results to computational flow over 2D automobile-related benchmark experimental data, and we concluded that CFD is a good method for analysing computational flow over a 2d car. We can make the car more aerodynamic in the future by making its body more and more streamlined.

REFERENCES

1. Salahuddin Khan, R. and Umale, S. (2014) 'CFD Aerodynamic Analysis of Ahmed Body', International Journal of Engineering Trends and Technology, 18(7), pp. 301–308. doi: 10.14445/22315381/ijett-v18p262.
2. Damjanović, D. et al. (2011) 'CFD analysis of concept car in order to improve aerodynamics', International Scientific and Expert Conference TEAM 2010, 1(2), pp. 63–70.

3. Velagapudi, N. K. et al. (2015) ‘Investigation of Drag and Lift Forces over the Profile of Car with Rearspoiler using CFD’, International Journal of Advances in Scientific Research, 1(8), p. 331. doi: 10.7439/ijasr.v1i8.2510.
4. Chaurasiya, V. V, Kushwaha, D. B. and Raees, M. (2017) ‘Aerodynamic Analysis of Vehicle Using CFD’, International Journal of Recent Trends in Engineering and Research, 3(3), pp. 131–137. doi: 10.23883/ijrter.2017.3056.s0sem.
5. Versteeg, H. K. and Malalasekera, W. (2005) An Introduction to Parallel Computational Fluid Dynamics, IEEE Concurrency. doi: 10.1109/mcc.1998.736434.
6. Premakara, Hanoca. (2015). 3-D Numerical Analysis of a SUV Car Using Structured Grid.