Automotive Aerodynamic Drag and Lift Analysis using Computational Fluid Dynamics Software : A Review

Jowad Md Madha[1], Anika Nawar, Dr. Md Mizanur Rahman
Department of Mechatronics Engineering, Faculty of Science and Engineering
World University of Bangladesh
Dhaka, Bangladesh - 1230
jowad.madha@icloud.com[1]

Abstract

The reduction of drag force is a complex and intricate challenge for the automotive manufacturing industry, as it has a direct impact on the overall performance of vehicles. Reducing drag force on a car enhances its ability to efficiently disperse air separation, resulting in reduced impediment and increased maximum velocity. The air resistance, which acts as a hindrance to the vehicle's movement, is considered a crucial element for enhancing both car performance and safety. Minimising air resistance contributes to the enhancement of the vehicle's ability to retain superior traction and stability. Aerodynamic profiling is used in this study to illuminate the impact of the vehicle's shape and surface representation on its behaviour. Results, reduce the drag force, and enhance the performance of the car. The primary objective of this study is to investigate the aerodynamic characteristics of a car by using computational fluid dynamics (CFD) software. Additionally, the aim of this paper is to review and analyse relevant prior research in this field.

Keywords
Automobile, Drag, Aerodynamics, Computational Fluid Dynamics

1. Introduction
The aerodynamics of automobiles focus on the airflow patterns surrounding a vehicle and analyse these patterns to understand overall performance, mainly the effect of lift and drag forces. The primary objectives of the study of vehicle aerodynamics are to minimise the air resistance or drag force effect on the movement of the vehicle and enhance stability during high-speed operation. Aerodynamics is also used to minimise air flow turbulence for optimization of the cooling system and the reduction of wind noise. These endeavours ultimately contribute to a heightened level of comfort for the driver (Hucho & Sovran, 1993). Various categories of racing vehicles may also use difficult techniques to minimise downward force, therefore enhancing adhesion and, consequently, improving surrounding abilities. The recognition of the standard aerodynamic characteristics of a vehicle design is becoming more widely acknowledged. The continuous endeavour to enhance fuel efficiency necessitates a comprehensive examination of vehicle drag, while the importance placed on vehicle handling underscores the imperative of acquiring a profound comprehension of aerodynamics. It is important to highlight that aerodynamic drag constitutes more than 50% of the overall resistance to movement when the speed surpasses 32 m/s, and it becomes the predominant factor after 45 m/s (Sudin et al. 2014).

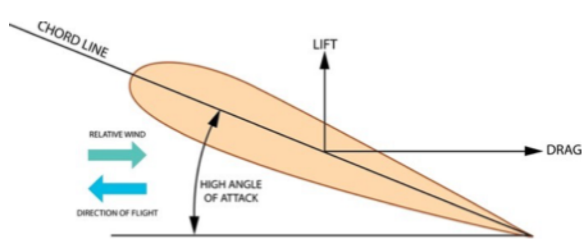


Figure 1. Drag & Lift impact on a Propeller

An undesirable shape of the car will experience more air interference, leading to increased loss of engine efficiency. Conversely, a reduction in power will lead to a decrease in the ability to move the vehicle, resulting in a decreased ability to carry loads and a lower velocity for the same amount of gasoline consumed. This is due to the reality that a vehicle is a comparatively small object surrounded by a significantly larger volume of air.

This research focuses on the analysis of drag and lift forces through the application of computational fluid dynamics (CFD) simulation techniques. In this paper on automotive aerodynamics, drag refers to the resistive force that opposes the forward movement of a vehicle through the surrounding air. The phenomenon occurs as a result of air resistance on the surface of the car and its effect on ambient airflow. However, the impact of lift force on the performance of a car in the context of automotive aerodynamics can be either advantageous or unfavourable. Positive lift is observed when the upward force generated by the airflow improves the stability and traction of a vehicle, especially during high-speed turns. On the other hand, a phenomenon known as negative lift, also referred to as downforce, appears when the downward force exerted by the airflow enhances the pulling power of the vehicle at the outermost portion of the road, particularly during rapid acceleration or deceleration. The examination of the characteristics of several variables, including wind velocity and orientation, road conditions, and vehicle velocity, on the aerodynamic characteristics of the vehicle can be conducted through the utilisation of computational fluid dynamics (CFD) simulation software. This software serves as an essential instrument in the process of designing and advancing automobiles. Fabricators have the ability to enhance the aerodynamic efficiency of a vehicle by utilising simulation software data to inform modifications to its shape, surface, and other relevant characteristics. This process can result in a reduction of drag, an increase in downforce, and an overall improvement in the vehicle's performance.

2. Literature Review : Review of Papers

The study conducted by Lai et al. (2011) employs empirical and numerical methodologies to evaluate and analyse the impact of the rear diffuser notchback angle on the composition of the wake and aerodynamic drag. This paper presents a comprehensive examination of the correlation between drag and rear diffuser angle, focusing on the interplay between these two factors. Additionally, a detailed analysis and discourse on the underlying flow mechanics are offered. The most recent study model puts forth and substantiates a theory regarding lower trailing vortices. The optimisations of the angle of the rear diffuser can contribute to the reduction of drag and enhancement of the aerodynamic performance of the vehicle, since it is a critical aspect that significantly affects the wake structure. The high span-wise clot gradient on the rear diffuser facet is caused by the efflux separation happening at a large rear diffuser angle. This is what causes two lower trailing vortices to form when the underflow is reattached. Hence, the optimization of the rear diffuser's position can contribute to the reduction of drag and improvement of wake characteristics.

The present study investigates the flow characteristics around a passenger car equipped with tail plates, utilising a numerical model based on the computational fluid dynamics (CFD) method, as described in the research conducted by Sharma (2013). The empirical work on the test vehicle and grid system was conducted using ANSYS 14.0 Fluid, a computational fluid dynamics (CFD) solver employed in this study. This study examines the aerodynamic data and complex flow structure once the numerical iterations have been completed. The present investigation employed a combination of testing and simulation techniques to determine the drag coefficient of a passenger automobile. This was achieved by first constructing a representative model of a passenger car using Solid Works 10 software. Subsequently, wind tunnel and boundary conditions were applied to the model using ANSYS Workbench 14.0 software. The present study examines and evaluates the aerodynamics of several tail plate designs to determine their impact on the drag coefficient of a passenger car. In this analysis, it has been seen that the coefficient of lift has experienced a reduction of 16.62%, while the coefficient of drag has undergone a decrease of 3.87%.

According to M (2013), the disruption of airflow in the area of the rear of a sedan is a significant factor contributing to the aerodynamic drag force experienced by the vehicle. In order to mitigate flow separation, the effectiveness of bump-shaped vortex generators is evaluated on the roof's trailing edge. Vortex generators are commonly employed on aeroplanes to mitigate flow separation, resulting in a dual effect of generating drag while simultaneously mitigating it by preventing flow separation transversely. The aggregate of positive and negative consequences is the determining factor in assessing the overall influence of vortex generators. The vortex generators located on the roof of a vehicle are easily noticeable due to their dimensions and configuration, which directly influence their aerodynamic impact. This research paper outlines the process of optimising several aerodynamic properties of a sedan vehicle through the utilisation of computational tools, including CFX, as well as empirical testing conducted using scaled models within a wind tunnel setting. This study presents findings from a computational simulation utilising Computational Fluid Dynamics (CFD) software and wind tunnel experiments. The results indicate that the drag coefficients of automobiles vary between 0.42 and 0.48, exhibiting a disparity of 12.5 percent.

The investigation focuses on examining the aerodynamic characteristics of three frequently seen rear designs, namely the fastback, notchback, and square-back, as stated by Wang et al. (2014). The goal of the research was to study the response of aerodynamic characteristics to modifications in the rear shape and to generate comprehensive experimental data that may serve as a reference for numerical simulations. Three models, each with yaw angles ranging from -150 to 150, were subjected to wind tunnel testing to determine their respective aerodynamic characteristics. The aerodynamic forces of drag and lift, as well as the distribution of surface pressure, were computed for a zero yaw angle utilising the realisable k-ε turbulence model. To enhance the precision and efficiency of calculations, a hybrid mesh technique involving tetrahedra, hexahedra, pentahedral, and prisms was employed to discrete the computing domain. The relationship between the rear shape and aerodynamic qualities is highly interconnected, such that alterations in the rear shape provide notable variations in aerodynamic properties. The majority of the aerodynamic drag was attributed to the negative pressure zone located in the rear.

This research done by Viveki and Chougule (2015) clarifies the impact of drag force on the performance of automobiles. The objectives of this study are to quantitatively assess the influence of roof curvature on drag and to gain a comprehensive understanding of the underlying systems responsible for drag. Studying the flow over passenger automobiles in a wind tunnel is a costly endeavour due to the expenses associated with the required framework and the substantial number of runs necessary to achieve effective drag reduction and optimization. The aforementioned expenditures can be mitigated through the utilisation of computational fluid dynamics (CFD) software, which enables the simultaneous execution of many iterations for the purpose of comparison and optimization. This work was motivated by the utilisation of a computational fluid dynamics (CFD) methodology to examine the aerodynamic characteristics of a MIRA reference car. The baseline model's roof has undergone modifications aimed at mitigating the aerodynamic drag experienced by automobiles. The optimization process for reducing drag using the Taguchi technique involves the consideration of three parameters: velocity, thickness to chord ratio, and location of thickness to chord ratio. These parameters are utilised in order to conduct simulations of air flow. Based on the results of the simulation, it was seen that an increase in roof curvature from the midpoint of the roof leads to a reduction in the wake zone and a corresponding decrease in the vehicle's drag coefficient by 2.58%.

3. Methods

To analyse the papers for a review, the simulation technique has been selected. The design was illustrated by using AutoCAD and Fusion 360. The simulation was done using Ansys Engineering Simulation Software, C15 Virtual Wind Tunnel and SD7032 Model Aerofoil. Figure 2 in represents the 2D and 3D CAD design of the SD7032 model aerofoil. The the drag and lift force upon the aerofoil will be tested following the conditions of table 1.

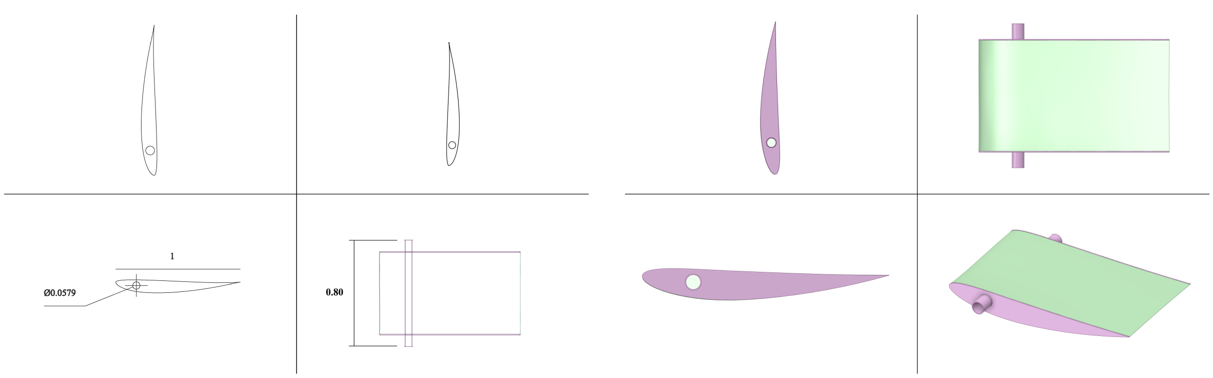


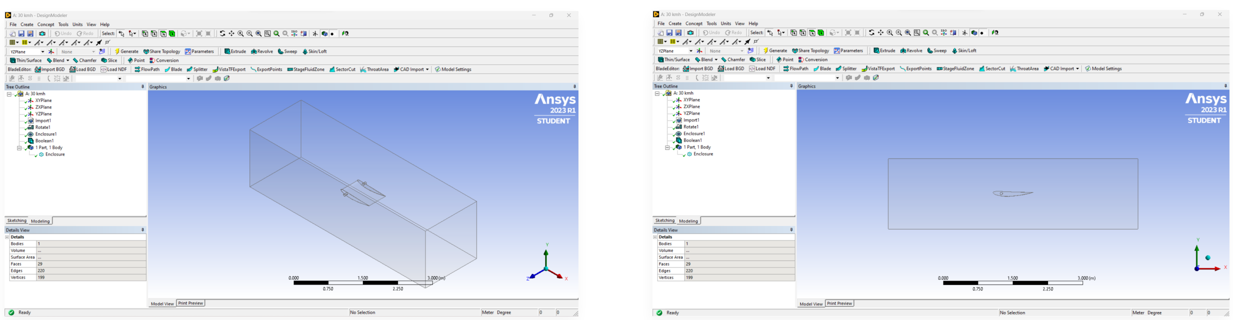
Figure 2. 2D and 3D Design of SD7032 Model Aerofoil.

Figure 3 represents the geometry setup in Ansys Engineering Simulation software. Figure 4 represents the meshing and Figure 5 shows the solution setup in the following software. The full setup has been done using the Fluent in Ansys.

Table 1. Simulation Setup Conditions

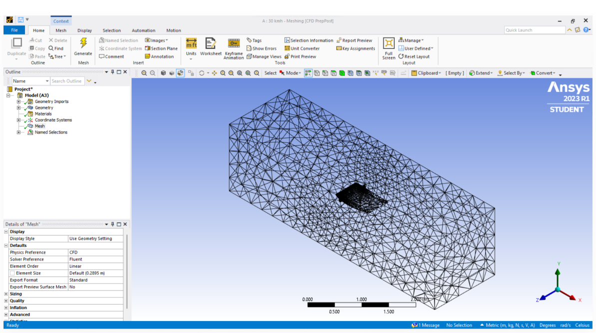
Geometry	Design Modeller	
Body Transformation	Angle of Attacks (AoA)	0°
		-5°
		-10°
Enclosure (C15 Wind Tunnel Dimension)	$x, -x$	2.250 m
	$y, -y$	0.700 m
	$z, -z$	0.460 m
Boolean Operation	Subtract Condition	
Named Portions	InletVelocity	Starting Wall of Fluid Flow
	OutletPressure	Back Wall
	FullWingEdge	SD 7032 Wing
Mesh	Sizing Operation	Edge
		Face
		Surface
	Inflation Operation	Enclosure
Setup	Double Precision	Serial
Model	k-omega (2 eqn)	SST
Material (Flow)	Air	Density = $1.225 kg\,m^{-3}$
		Viscosity = $1.7894e - 05 kg\,m^{-1}$
Cell Zone Condition	Fluid Flow	
Boundary Condition	InletVelocity	$30 km\,h^{-1}$
		$40 km\,h^{-1}$
		$50 km\,h^{-1}$
		$60 km\,h^{-1}$
		$70 km\,h^{-1}$
		$80 km\,h^{-1}$
		$90 km\,h^{-1}$
		$100 km\,h^{-1}$
		$110 km\,h^{-1}$

		$120 km\,h^{-1}$
Solution Method	Scheme	Coupled
	Turbulent Kinetic Energy	Second Order Upwind
	Specific Dissipation Rate	Second Order Upwind
Report Definition	Force Report (File & Plot)	Lift Force on Wing
		Drag Force on Wing
Solution Initialisation	Standard Initialisation	Computed from InletVelocity
Run Calculation	1500 Irritations	
Graphics & Animation	Contours	FullWingEdge
	Vectors	FullWingEdge



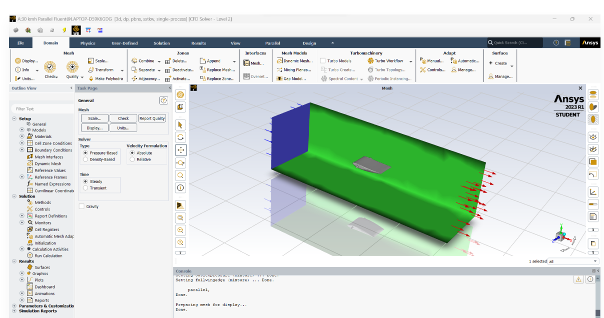
*Ansys | Engineering Simulation Software

Figure 3. Geometry Setup with 0° Angle of Attack



*Ansys | Engineering Simulation Software

Figure 4. Meshing with 0° Angle of Attack



*Ansys | Engineering Simulation Software

Figure 5. Solution Setup with 0° Angle of Attack

4. Data Collection

The tables of data collection are given below.

Table 2. Data for 0° Angle of Attack

Angle of Attack	Density of Fluid kgm^{-3}	Air Flow Speed kmh^{-1}	**Drag Force** F_{Drag} (N)	**Lift Force** F_{Lift} (N)
0°	1.225	30	0.44768648	-1.9723977
		40	0.82241367	-3.6595392
		50	1.2175055	-5.4795553
		60	1.7454471	-7.9312138
		70	2.3883174	-10.821938
		80	3.0946331	-14.240214
		90	3.9391721	-18.0173
		100	4.8206783	-21.922845
		110	5.7419137	-26.03835
		120	6.8421815	-31.80875

Table 3. Data for -5° Angle of Attack

Angle of Attack	Density of Fluid kgm^{-3}	Air Flow Speed kmh^{-1}	**Drag Force** F_{Drag} (N)	**Lift Force** F_{Lift} (N)
-5°	1.225	30	0.84938936	-4.2656923
		40	1.5042856	-7.5774823
		50	2.3632095	-11.80809
		60	3.3773118	-17.041968
		70	4.5589842	-23.202485
		80	6.0731916	-30.132829
		90	7.7400856	-38.089037
		100	9.5017109	-47.174491
		110	11.505901	-56.985056
		120	13.521855	-68.030244

Table 4. Data for -10° Angle of Attack

Angle of Attack	Density of Fluid kgm^{-3}	Air Flow Speed kmh^{-1}	**Drag Force** F_{Drag} (N)	**Lift Force** F_{Lift} (N)
-10°	1.225	30	1.4750587	-7.1890326
		40	2.6200968	-12.769841
		50	4.0880465	-19.951519
		60	5.893753	-28.766118
		70	8.0142861	-39.154253
		80	10.483422	-51.054276
		90	13.277513	-64.696539
		100	16.384969	-79.822828
		110	19.832215	-96.61419
		120	23.603061	-114.93201

5. Results and Discussion

For 0° angle of attack, the following figure 6, 7 and 8 shows the following results of drag and lift ratio.

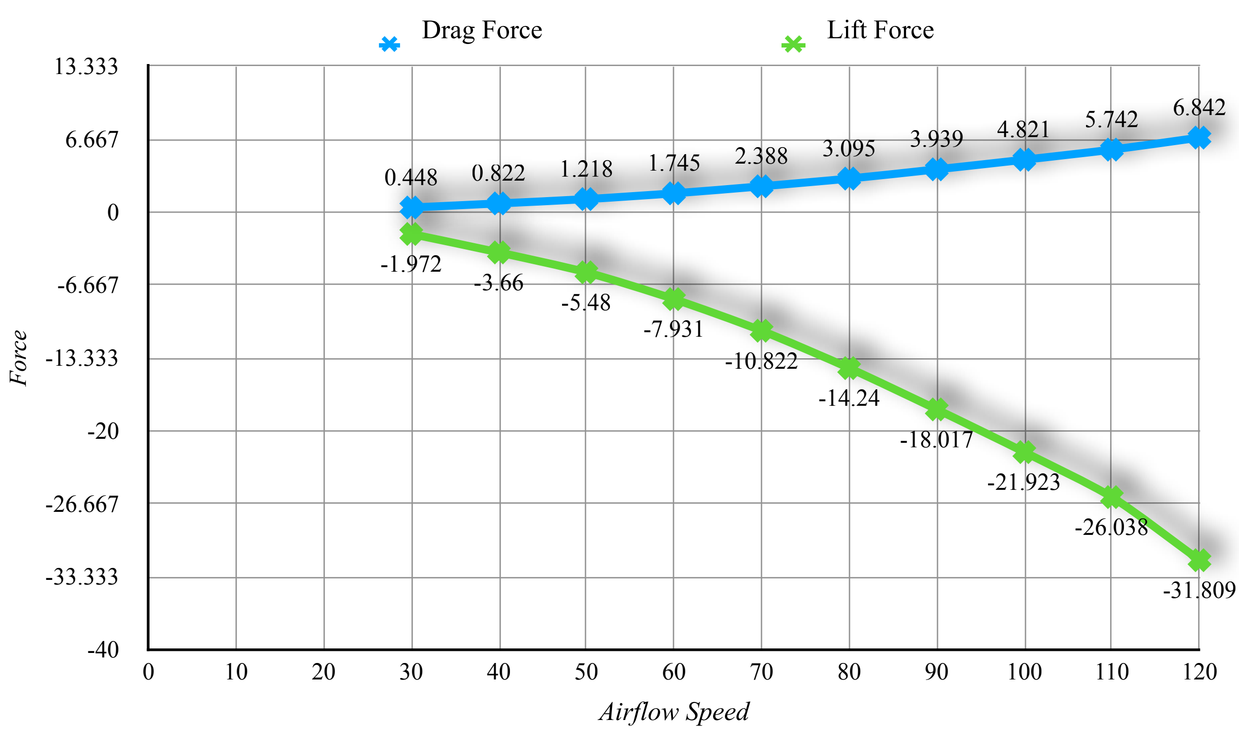
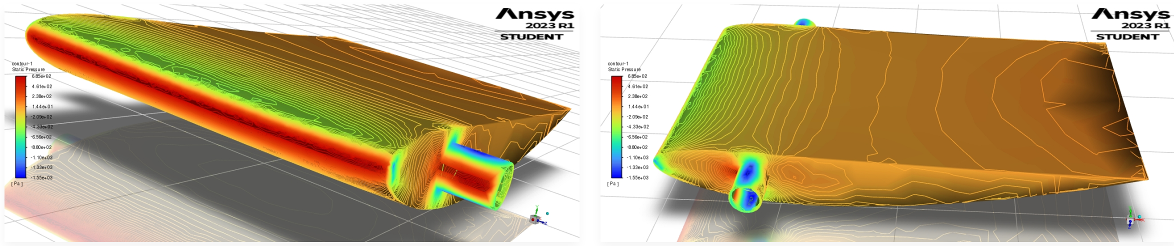
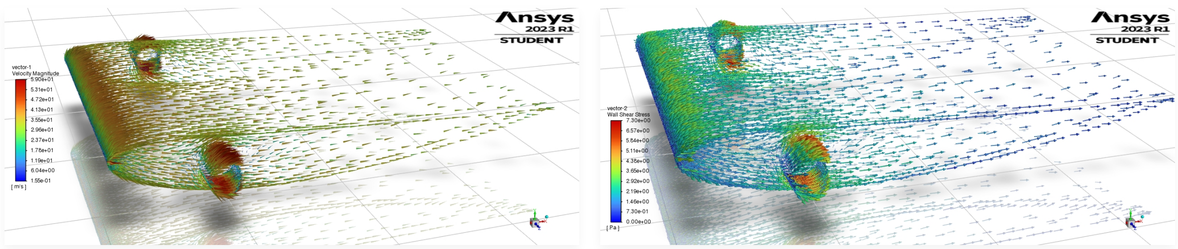


Figure 6. Acting Force with Increased Speed upon 0°



*Ansys | Engineering Simulation Software

Figure 7. Pressure Contour Result of the Simulation with a 0° Angle of Attack



*Ansys | Engineering Simulation Software

Figure 8. Velocity Result of the Simulation with a 0° Angle of Attack

For -5° angle of attack, the following figure 9, 10 and 11 shows the following results of drag and lift ratio.

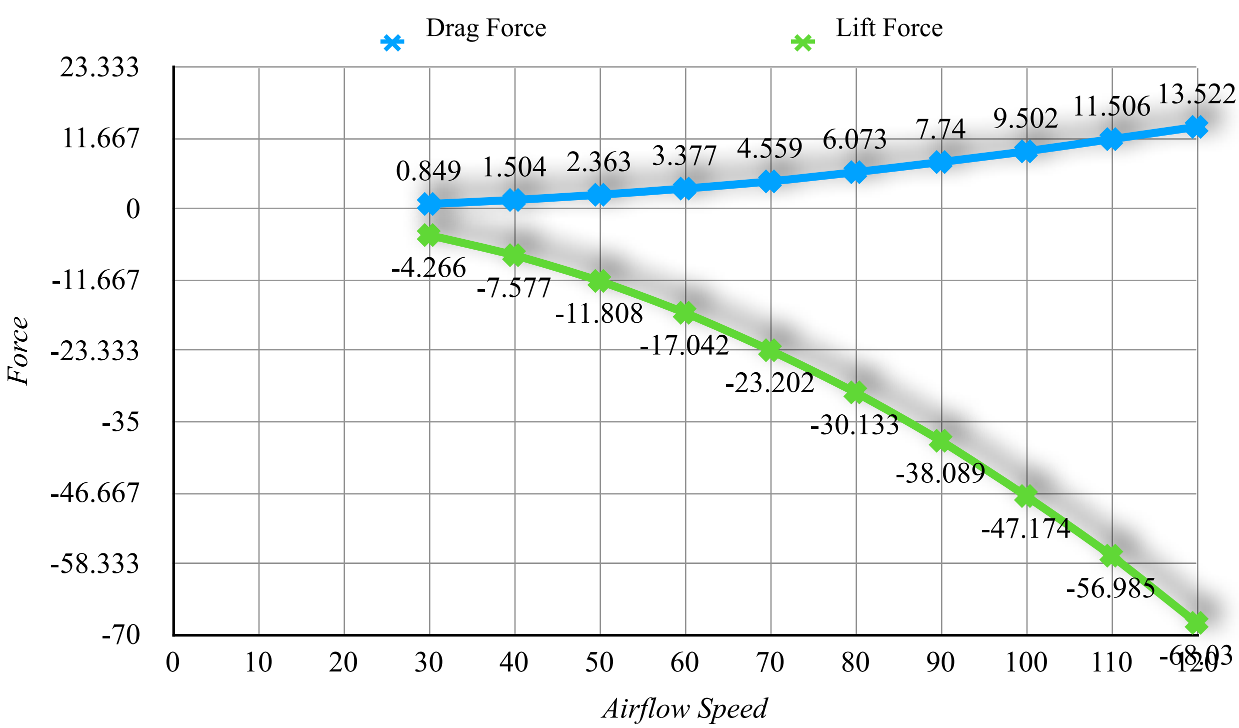
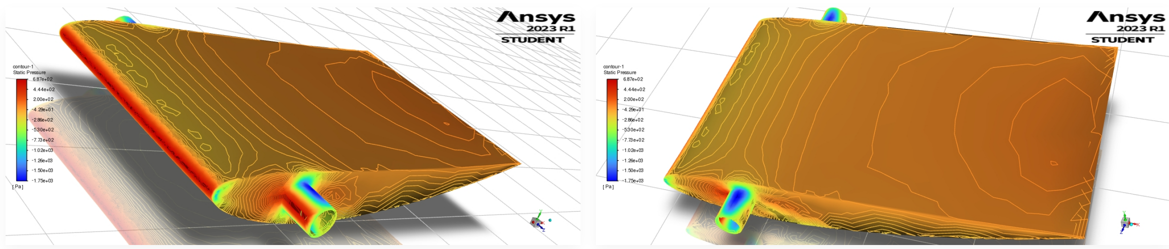
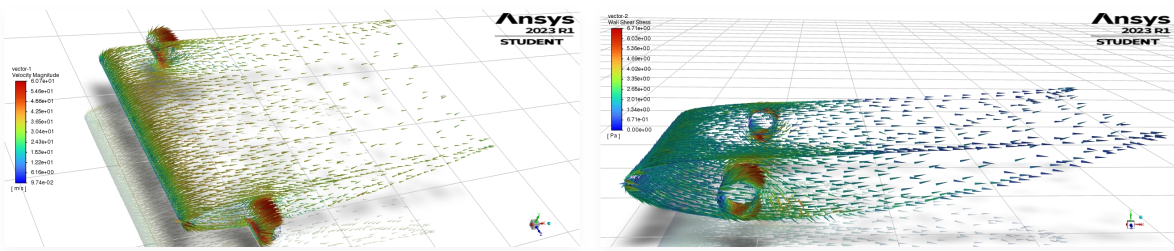


Figure 9. Acting Force with Increased Speed upon -5°



*Ansys | Engineering Simulation Software

Figure 10. Pressure Contour Result of the Simulation with a -5° Angle of Attack



*Ansys | Engineering Simulation Software

Figure 11. Velocity Result of the Simulation with a -5° Angle of Attack

For -10° angle of attack, the following figure 12, 13 and 14 shows the following results of drag and lift ratio.

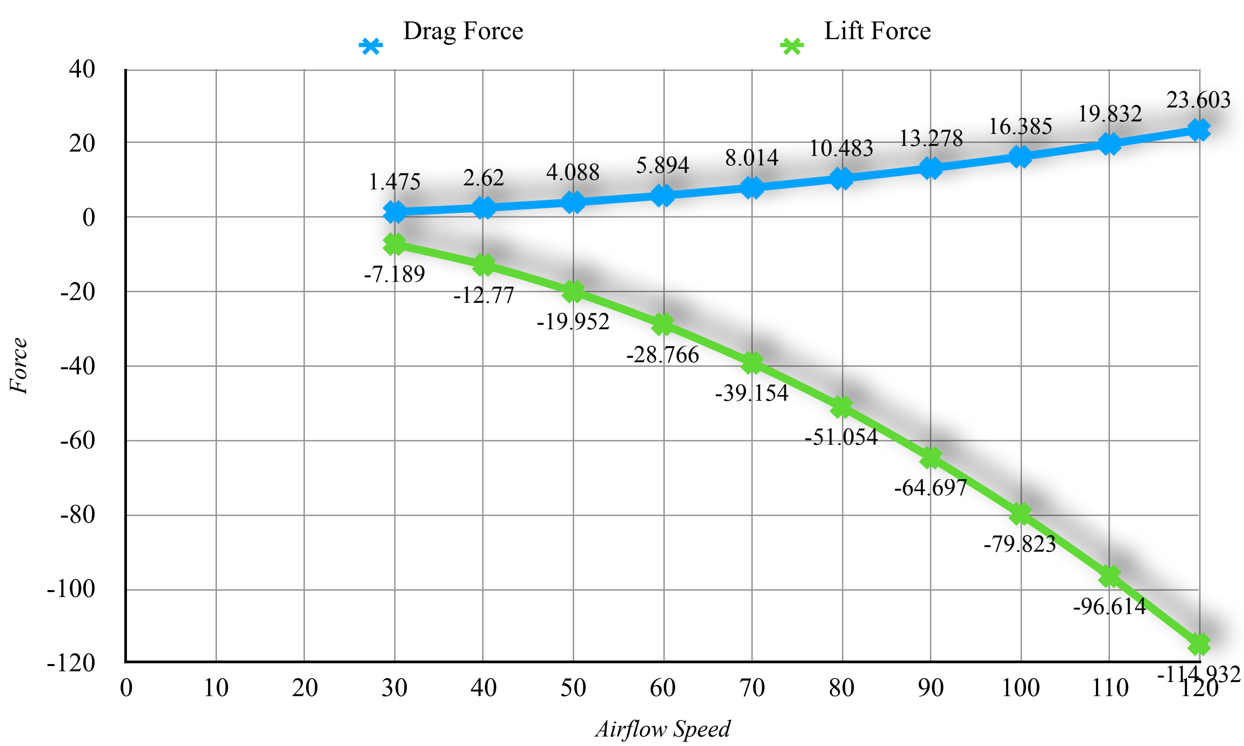
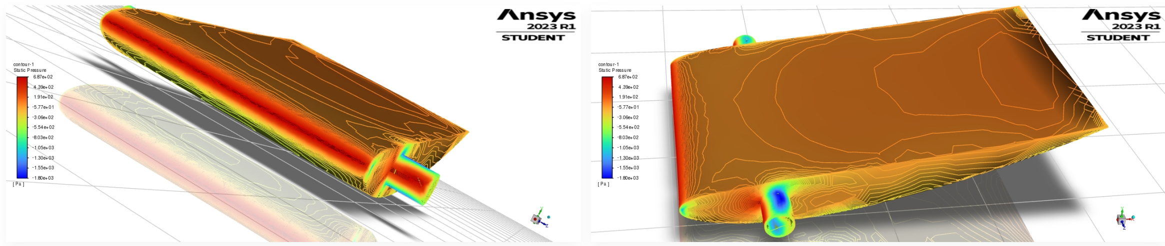
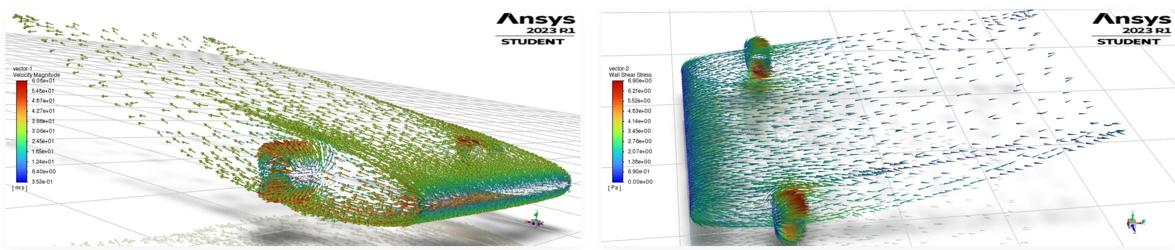


Figure 12. Acting Force with Increased Speed upon -10°



*Ansys | Engineering Simulation Software

Figure 13. Pressure Contour Result of the Simulation with a -10° Angle of Attack



*Ansys | Engineering Simulation Software

Figure 14. Velocity Result of the Simulation with a -10° Angle of Attack

These analysis shows a slight similar results according to the review papers have represented.

6. Conclusion

Based on the preceding discourse, it can be deduced that the utilisation of software-based computational fluid dynamics analysis serves as a supplementary instrument for quantifying aerodynamic forces. Computational Fluid Dynamics (CFD) is widely recognised as a rapid and precise method for quantifying aerodynamic forces. It is imperative to prioritise the exploration and implementation of diverse technological advancements aimed at mitigating aerodynamic forces, with the ultimate goal of enhancing the overall aerodynamic efficiency of automobiles. The fetched out data from the review papers and the analysis represents the accuracy of the researches. The addition of certain equipment to the car can effectively reduce the drag and lift force, thereby enhancing its overall performance.

References

Hucho, W., & Sovran, G. (1993). Aerodynamics of Road Vehicles. *Annual Review of Fluid Mechanics*, *25*(1), 485–537.

Sudin, M., Abdullah, M., Shamsuddin, S., Faiz, R., Ramli, & Tahir, M. (2014). I J E N S Review of Research on Vehicles Aerodynamic Drag Reduction Methods. *International Journal of Mechanical & Mechatronics Engineering IJMME-IJENS*, *14*(02), 35.

Lai, C., Kohama, Y., Obayashi, S., & Jeong, S. (2011). Experimental and Numerical Investigations on the Influence of Vehicle Rear Diffuser Angle on Aerodynamic Drag and Wake Structure. *International Journal of Automotive Engineering*, *2*(2), 47-53.

Sharma, R. B. (2013). CFD Simulation for Flow over Passenger Car Using Tail Plates for Aerodynamic Drag Reduction. *IOSR Journal of Mechanical and Civil Engineering*, *7*(5), 28–35.

M, P. K. (2013). Experimental Investigations on Optimisation of Aerodynmaic Charactericstics in a Hatchback Model Car 3 Using Vortex Generators. *Proceedings of the Eighth Asia-Pacific Conference on Wind Engineering*.

Wang, Y., Xin, Y., Gu, Zh., Wang, Sh., Deng, Y., & Yang, X. (2014). Numerical and Experimental Investigations on the Aerodynamic Characteristic of Three Typical Passenger Vehicles. *Journal of Applied Fluid Mechanics*, *7*(04).

Viveki, R., & Chougule, N. K. (2015). Computational Fluid Dynamics Optimization of Roof Curvature for Drag Reduction of Passenger Car. *International Engineering Research Journal (IERJ)*, *1*(5), 224–231.