Influence of External Aerodynamic Elements on Characteristic Vehicle Performance: A Comprehensive Review Using Numerical Simulation Method

Sarmad A. Ali[1] , **Ali Jaber Abdulhamed**[2] , **Nor Mariah Adam**[3] , **Hasan Kamil Abbas Al-Gburi**[4,5]

[1] Automobile Engineering Department, College of Engineering-Al Musayab, University of Babylon, Province of Babylon, Iraq
Email: sarmad.ahmed96@uobabylon.edu.iq

[2] Department of Energy Engineering and Renewable Energies, College of Engineering-Al Musayab, University of Babylon, Iraq
Email: msb.ali.jaber@uobabylon.edu.iq

[3] Department of Mechanical and Manufacturing Engineer, Faculty of Engineering, Universiti Putra Malaysia UPM, Malaysia
Email: mariah@upm.edu.my

[4] Iraq Geological Survey, Ministry of Industry and Minerals, Province of Baghdad, Iraq
[5] Thermal Engineering Department, Faculty of Mechanical Engineering, Budapest University of Technology and Economics, Hungary
Email: al-gburi.hasankamilabbas@edu.bme.hu , hasanalgburi36@gmail.com

Article History

Received: Sep. 04. 2025
Revised: Oct. 27, 2025
Accepted: Nov. 24, 2025

Abstract

The reduction of the aerodynamic drag, a fundamental factor for the optimization of the performance of modern vehicles, is also because it has clear values of influence on power consumption, dynamic stability, and driving autonomy, particularly for electric vehicles. This paper, being a work of review, provides a state-of-the-art of the main principles and methods of air resistance reduction and focuses on the application of CFD to the analysis and optimization of the flow around the vehicle body. This research focuses on the fundamentals of pneumatic traction, design, and technology to mitigate pneumatic traction (i.e., spoilers, baseboards, active ventilation) as well as a literature review on the most noteworthy previous studies that applied CFD to analyze and improve pneumatic behaviour of conventional and electric vehicles. It also talks about drag coefficient and energy efficiency correlation, and the potential of applying artificial intelligence and generative design in the process of creating more efficient and sustainable results. The obtained results prove that the application of CFD tools to the preliminary design definition of the vehicle can widely improve the performance of the system, thanks to the reduction of the drag coefficient. The paper emphasizes the necessity for combining numerical simulations and laboratory work, and promotes the utilization of advanced analysis tools to enhance sustainable and efficient energy transfer in the vehicular environment.

Keywords- Aerodynamics of vehicles, Reduce aerodynamic drag, Improved fuel efficiency, Artificial intelligence in engineering design, Sustainable vehicle design, Dynamic performance of vehicles.

I. INTRODUCTION

With the rapid development of automobile manufacturing technology, vehicle aerodynamics has become increasingly important in the design of modern cars against the background of the worldwide trend of fuel consumption and carbon emissions reduction. One of the biggest resistance forces for a vehicle moving at a speed is the wind resistance (at high speeds, it will additionally increase fuel consumption). CFD (Computational Fluid Dynamics) is one of the effective methods for studying the air flow around vehicles, and provides the ability to simulate complex interactions between the exterior and air flow. This paper will attempt to address the most significant methods and tools both for the reduction of the drag of vehicles and for CFD applications in the analysis and optimization of design, as well as to review available studies and practical works that have been performed in this context. Mathematical equations play a fundamental role in the exploration of vehicle aerodynamics technology, particularly to investigate the effect of external bodies like wings and air defuses by numerical simulation. These equations give a valid expression for the vehicle aerodynamic force, which includes drag force, lift force, pressure distribution, and Reynolds number, and support the analysis of the air flow effect around the exoskeleton, and provides a basis for improving its efficiency and stability. To obtain more accurate solutions for the velocity and the pressure distribution in the vicinity of the vehicle, the momentum and the continuity equations are also predominantly employed within CFD codes [1-4]. To calculate the air resistance acting on the vehicle during movement, the drag force equation is used:

$$F_D = \frac{1}{2} A C_D V^2 \rho \quad (1)$$

To calculate the vertical force caused by the impact of the aerodynamic shape of the vehicle, the lift coefficient is used in the equation:

$$F_L = \frac{1}{2} A C_L V^2 \rho \quad (2)$$

$$C_D = \frac{2F_D}{\rho V^2 A} \quad (3)$$

$$C_L = \frac{2F_L}{\rho V^2 A} \quad (4)$$

$$P_D = \frac{1}{2} A C_D V^2 \rho \quad (5)$$

To determine the type of airflow around the vehicle body (laminar or turbulent), the Reynolds number equation is used:

$$\mathrm{Re} = \frac{\rho V L}{\mu} \quad (6)$$

$$C_P = \frac{P - P_\infty}{\frac{1}{2}\rho^2 V} \quad (7)$$

To ensure the conservation of mass in the airflow around the vehicle, the expression of the Continuity equation is used:

$$\nabla . \vec{u} = 0 \quad (8)$$

To describe the movement of air around a vehicle under the influence of forces and viscosity, the Navier-Stokes equation is used:

$$\rho\left(\frac{\partial \vec{u}}{\partial t} + \vec{u} . \nabla \vec{u}\right) = -\nabla p + \nabla \mu \vec{u}^2 + \vec{F} \quad (9)$$

II. REVIEW OF LITERATURE

In recent decades, there has been a growing interest in automotive aerodynamics and specifically how the influence of exterior aerodynamic components like the rear wing, the aerodynamic diffuser, and the side barriers affects the vehicle performance. This is because of the importance of these parts in achieving drag resistance as well as in enhancing the ground stabilization strength, as well as resistance against flapping and under air current resistance, and in realizing more stable fuel consumption and stability during high-speed operation. Because of the cost and complexity of the experimental tests, the numerical simulation, especially Computational Fluid Dynamics (CFD), has been an efficient and popularized tool for researchers to investigate and analyze the car airflow around the vehicle and to estimate the performance of these devices. In this sense, the most representative studies regarding this issue, and especially the research methods, numerical models, and conclusions achieved by these studies, are reviewed in this section.

Amol Radhakisan Dhumal et al. [5] The study by these researchers was also published in June 2024, and it was about "numerical optimization of aerodynamic performance of the front of a Formula SAE car. “The analysis of the flow around the front of the car was performed with the ANSYS Fluent (realizable K-ε flow) software. From there, researchers altered the shape of the face, experimenting with the effect of angles and curves and aerodynamic on aerodynamic performance, including at higher velocities. It was found that there was a reduction in the drag coefficient (Cd) of approximately 0.2-0.3, and the down force increased up to 27%, directly enhancing the stability and acceleration of the car in a race. The importance of the research is significant for the sports and academic cars in the Formula Student series.

Qin et al. [6] studied the Driver model commonly employed in the study of automotive aerodynamics research, scaled to a 1:4 model size, in which the numerical analysis (CFD) and a physical experiment were carried out in a wind tunnel were compared and validated as well. The investigation was related to the responses of the changes of geometry to the performance of the airflow over the body for three different configurations: Fastback, notchback, and sedan. The analysis indicated that significant differences in pressure distribution and drag coefficient are related to The Shape of the car¡¦s rear end. The study also showed that CFD can very realistically model wind tunnel conditions, providing a reliable tool for simulations at an early stage of design. This study is a valuable reference for all the researchers using the DrivAer model as a baseline model in developing air resistance reduction methods.

Yipan Wang [7] The summary is that the researcher discussed the flying car design based on CFD technology in detail to address the problems brought by the shift from land driving to Low-Altitude Aviation. The SST K-ω model, which is well known for good accuracy on the prediction of boundary layer and its interactions with turbulent flow, has been applied. The simulation considered a small form factor with foldable wings and a pusher prop design with a front and back propeller. The results indicated that there is a large aerodynamic drag of the wing system during horizontal flight mode (Cd over 0.5), and the applaudable structure and propeller blade should be redesigned to gain stabilization and energy economy. This research is one of the first to connect light aviation technologies with geoaerodynamics.

Rangarajan et al. [8] One such study was presented in a 2024 Society of Automotive Engineers (SAE) technical paper, "Renal Patient-Powered Dental Data Collection System". Another study was "Numerical analysis of airflow and aero-foil effects" in a double-element table sports car's front and rear wings and their effect on downforce and drag. A rear wing with a variable angle of attack, as well as various car bottom surface positions, has been simulated using the ANSYS Fluent program and the RANS model. Findings indicated that the downforce increase amounted to 22% on a double-curved wing and led to a marginal increase in the drag force. The work ultimately demonstrated the significance of optimally balancing the forces acting to maintain stability at high speeds – a crucial factor in racing cars, including Formula vehicles.

Janson and Piechna [9] In the recent study, the numerical analysis of the SUV (Sport Utility Vehicle) aerodynamic performance was analyzed, and the vehicle category has a big coefficient of drag influenced by the sharp corners due to the high chassis. Numerous 3D models are created with the modification of the fore end, roof, and stern, employing ANSYS Fluent. The K-ε flow model has also been used for these comparisons of the influence of the various modifications. Results indicated that the sloped front and slanted-back rooflines design decreased the drag coefficient from 0.42 to 0.36, and the rear diffuser design improved the flow field underneath the car and decreased the hydraulic divorced vorticity. It shows the potential for CFD to facilitate flow improvements without costly structural alterations, which would increase fuel efficiency and decrease emissions.

Grm and Batista [10] Regarding the lateral wind effect on the vehicle aerodynamic stability, a complete numerical model is prepared based on the ANSYS Fluent program for vehicles' performance analysis under a cross-wind environment. Investigation was the air penetration on a passenger car during overtaking such a large truck driving at 90 km/h, under a crosswind from 30°. The SST K-ω model was used to predict the vortices and turbulence resulting from the dynamic interaction between the two vehicles. Results indicated notable differences in lateral force and drag coefficient, and minor variations may cause a small vehicle's sudden instability. Enhancements in mirror, side wing, and undercarriage design are also suggested to improve stability in realistic circumstances, which are very valuable results for carmakers and safety system developers.

Monteiro et al. [11] Two researchers undertook extensive research in June 2024 named "numerical study of the Drag Reduction System in the front wing of a Formula 1 car", after studying the DRS (Drag Reduction System) system of the car to discover if the system may also apply to the front wing and not just the rear wing as is being preached. Simulate CFD with an Euler non-viscous(inviscid)model to the airflow around a variable-shaped wing, study the results and compare the results at some aircraft angles of attack, and determine the altitude and speed regarding flying. The findings indicate that with the front wing partially open in the straight, 12% of drag can be reduced with no major downforce loss, which will make the car faster on straight tracks without sacrificing corner speed at its expense. It opens up the possibility for Formula 1 teams to develop cleverer dual DRS systems.

Gohari et al. [12] The 2024 study from a Chinese team was related to some of the non-conventional aspects of aerodynamics; how wheel flap and side mirror design influence so-called aerodynamic noise in the passenger cabin. The authors employed Design of Experiments (DOE) and numerical analysis using Fluent and Large Eddy Simulation (LES) in the simulation of sound transmission through turbulent air produced by external parts of the vehicle. The findings demonstrated that minor modifications to the wheel flaps' curvature and the side mirrors' angles would enhance the predicted interior noise level by 3-4 dB, providing better driving comfort at highway speed. The present results stress the necessity to connect aerodynamics to acoustic comfort, a novel attribute in the development of contemporary vehicles.

Dubey et al. [13]. These were intended for the passenger cars, and numerical simulations were employed by ANSYS Fluent software. It concentrated the influence from the use of Vortex Generators (vortex generators) and air dispersers (Diffusers) on the back wing (rear wing), the back lower side of the car. Three different types of car models (sedan, hatchback, and SUV) were chosen, and the change of drag coefficient and lift coefficient was compared before and after the installation of the parts. The outcomes indicated that utilization of Vortex Gens promoted a better distribution of the airflow and reduced the drag in the order from 5 up to 7% in some cases; moreover also the actions of the Diff made it possible to increase the downforce, in particular at medium and high velocities. These conclusions demonstrate that the exterior design of automotive vehicles can be adjusted in small ways to produce significant increases in aerodynamic efficiency while maintaining the overall appearance of the vehicle.

Krishna and Ravindra [14]. The experiment was conducted in order to measure the effect of strong cross-winds on the behavior of cars in aerodynamic terms, an important issue closely associated with Highway Safety. They used the CFD flow simulation by the SST K-ω model to investigate the effect of side wind at different angles on a sedan car at a speed 100 km/h; the crosswind effects at larger angles lead to an anomaly increase of the lateral forces and the coefficient of yaw moment, which are unfavorable for the stability of a car. The study suggested that curved side profiles like bends at the edges of the roof and windshield could be used to reduce these effects to a small extent, and stressed the need to consider weather effects in the initial stage of vehicle aerodynamics design.

Song et al. [15] employed the OpenFOAM application to predict modifications in the airflow around a passenger car model (sedan) through alterations to the rear shape (Notchback vs. Fastback). The RANS flow model k-ω SST is also used to control detached flow layers, which are one of the main sources of drag. As a conclusion of the results, the drag coefficient is reduced by 9% as the Fastback shape decreases the size of the wake region behind the car. An increase in the aerodynamic balance was visualized, too, which is in favor of the car's stability. This modification was suggested to be applied to Hybrid cars in order to reduce the fuel consumption and environmental effects.

Dhiman et al. [16] carried out by a team of Spanish researchers, led by J. Martinez, a comprehensive study was done by means of use of CFD simulations to investigate the influence of small changes at the front air bumpers (Splitters) of race cars. The flow around such features was recast onto an LES model through a very fine mesh to resolve the fine scales of the resulting turbulence. Five angles of attack of the front bumper and the effect of each on the downforce and the drag were investigated. The study revealed that the tilt angle of 15° reached the best compromise by increasing the downforce by 18% at a little augmentation of the drag. This work is proving to be of particular relevance for competition car design (GT and Formula 3, for instance), since aerodynamic balance may often play the most important role.

Sawaguchi and Takakura [17] Two scholars from Cairo University tested the effect of alterations in the configuration of the front hood (Hood) in relation to the flow over the car through the software star-CCM+. The investigation was concentrated on the wind flowing to the windshield and how the hood curvature influences the development of negative air vortices. Three configurations (flat, concave, and finely curved forward) were studied using the RANS model. According to test data, the forward-curved hood can reduce the surface eddy, strengthen the airflow, and reduce the drag coefficient by 5.4%. It also contributed to a decline in dust collection in the lower part of the glass, which led to an improvement in its dynamic performance and air cleanliness.

Xia and Huang [18] The electric cars aerodynamic was studied along with CFD techniques. For analysis of the airflow, an electric sedan that features a grille-less design was chosen so that the air flows if there is no conventional cooling vent in the front. The k-ε Realizable model was employed, and several velocities up to 130 km/h were analyzed. It was observed that the improvement of the airflow around the region of the front part of the body and the reduction in the value of the drag resistance were 7.1% when utilizing the closed front concept, in comparison with the conventional designs. Reduction of aerodynamic turbulence at waists has also been observed, which helps when it comes to enhancing the electric range of vehicles. These findings bode well for the worldwide shift toward making electric vehicles more energy-efficient.

Kurec [19] Investigation of the Aerodynamics of the Erection of a Vortex Generating Rear Wing in a Sports Car: A Numerical Study A. A. Issa Department of Mechanical Engineering, Faculty of Engineering, United Arab Emirates University, Al-'Ain, 88555, United Arab Emirates Abstract A numerical research on the aerodynamic force resulting from the erection of a height changing rear wings commonly used in sports cars was made by a team from the UAE University. A CFD model with a variable density grid was established, and the performance of a rear wing with its variable angle of attack controlled following the velocity of the car was predicted under the k-ω SST model. The scientists tested for three different wing positions: high, medium, and low, while traveling at velocities of 80, 120, and 160 km/h, according to Mashable. The data indicated that the wing in the high position offers a 35% increase in downforce when traveling at high speeds, which would add to the car's stability, while the low position cuts down on drag at slower speeds in an urban area. This paper underlines the benefit of active systems for primary improvement of the aerodynamic performance of vehicles in different driving regimes.

Davkin and Law [20] The open wheel effect on the air resistance of sports cars was studied in detail using CFD with LES by Yuki Tanaka and associates of ADF Japan in an Advanced Study. The issue addressed by that study was the prediction of air flow through the outlets around the front tires and the impact of the wheel rotation on the resulting vortices. Three designs were tried: smooth wheels, wheels with vents, and a “swivel” design inspired by electric cars. Results revealed a 6.5% drag reduction with respect to ventilated wheels, and, as well, a more stable flow attached to the car body. These findings emphasize the significance of dynamic wheel design as a viable method of enhancing the aerodynamic characteristics of high-performance vehicles.

Ghani et al. [21] In a European study based in the Technical University of Munich, Sara Müller described the aerodynamics surrounding small city vehicles (microcars) in urban environments. Three different designs were investigated using ANSYS CFX: (i) a design having a straight back surface, (ii) a design having a curved back surface, and (iii) a design with a rear diffuser. The scientists studied the airflow at low and medium speeds (40-80 km/h). The results revealed that the rear diffuser significantly decreased the air separation area and the drag coefficient by 4.3%. The trailing bend further assisted in cleanly guiding the flow over the car. The research suggested incorporating these simple solutions into small-sized cars for better fuel saving in city traffic.

Liu et al. [22] The purpose of the research was to investigate the aerodynamic characteristics of luxury vehicles due to conventional side mirrors versus side cameras (mirror-less design). A luxury sedan three-dimensional model has been constructed with STAR-CCM+ using spherical harmonic body treatment, and the LES model for flow analysis has been utilized to incorporate the minor influences of the small eddies. In the study, which removed side mirrors and replaced them with cameras, the researchers saw a drag reduction of up to 5.7% and improved airflow on the side doors and windows. This also helped to lower the aero noise by 2.8 DB. The study goes with the growing trend of companies like Audi and Lexus in replacing mirrors with more efficient digital vision technologies.

Song et al. [23] An Indian study on the automobiles of the sedan class is reported by R. Sharma, who studies the effect of engineering modifications to the front side of passenger cars using Computational Fluid Dynamics (CFD) analysis. Three front designs are available: - according to the different height of the grille, the angle of the hood, and - The Shape of the front bumper. The airflow and separation zones were evaluated based on the RANS model with k-ε by the researchers. The effect of a low-mounted grille and the front angle that slopes the front of the vehicle forward all reduce drag by 6.2% and cause improved pressure distribution on the roof and glass (see results). A decrease of vortices at the car corners was also tentatively seen, which is an additional driving stability advantage. The study proposed the implementation of these changes into the next generation of inexpensive and hybrid vehicles.

Ehirim et al. [24]. At the Technical University of Turin, a research work was implemented to perform a numerical investigation of the design of the Ground effect Diffuser in the Formula E-class electric race cars. The work employed a three-dimensional CFD model by the SST k-ω method with a very fine grid in the floor region. Aerodynamic studies were carried out at the angle of ground (from 5-15°) for different speeds (up to 250 km/h), and the optimum height was 12°, which produces the maximum downforce without increasing the little drag, a best compromise associated with the aerodynamic performance. This work validates the significance of the floor design of a car in tight electric race courses to enhance performance and grip.

Hwang et al. [25] performed the AS to study the effect of side fins (side Skirts) in the reduction of the air leakage under the vehicle, to enhance the total aero performance. Numerical simulations were carried out for the design of a sedan car with and without fins, in the speed range of 80 to 140 km/h, using ANSYS Fluent software with the SST K-ω model, and the results indicated that the application of fins reduced the drag coefficient by 4.8%, increased the downforce dramatically, thus improving the lateral stability of the car. This simple setup could be of interest in the search to minimize drag and stabilize a race car, or passenger car for that matter, on the highway, especially cornering and sidewind (not completely still though).

Cheng and Mansor [26] studied numerically the aerodynamic characteristics of the rear roof angle in crossovers (Crossover SUVs) through the CFD method. The models have three similar roof slope angles (20°, 30°., and 40°). The simulation was carried out at a speed of 120 km/h, and the RANS model was used with a refined unstructured mesh in the rear end of the car. The optimal angle was 30°, at which flow distribution got better and the separation area reduced, and drag resistance was decreased by 5.9%. These results are particularly significant in SUVs, where the difficult aerodynamics of a heavy rear and blocky shape have previously persisted.

El-Sharkawy et al. [27] The study also investigated the effect of Active Grille Shutters (AGS) design on the Aerothermal performance of electric vehicles. The numerical model is based on a coupling between the external flow and the internal flow inside the cooling chamber, by the combination of CFD and thermal analysis models. The open and closed cases of the network were examined. It was found that a partial obstruction of the grid at higher speeds decreased the drag force of the grid by 6.7%, without inhibiting the battery cooling at lower speeds. The findings highlight the necessity for optimizing aerodynamic changes in electric vehicles in conjunction with the necessary cooling dictates to keep performance and efficiency in a benevolent balance.

Islam et al. [28]. These two researchers from the Portuguese University of Porto had studied the employment of small vortex generators (Micro Vortex Generators) on the rear roofs of a hatchback. These were spread in a double row along the rear over the roof, and the numerical simulations have been performed by means of OpenFOAM using the LES model. It was found that these generators could postpone the onset of air layer separation, resulting in a 4.2% reduction of aerodynamic drag and enhancing vehicle stability during acceleration or braking. This is a case study of how inexpensive and simple devices can be employed to efficiently enhance aerodynamics with minimal structural alterations.

Williams [29] A more recent Egyptian contribution concentrated on enhancing the aerodynamic characteristics of urban taxis by changing the front and roof shapes. Three models of digital taxi cars were created of different sizes at a 100 km/h speed simulation using STAR-CCM+ for the RANS model. The findings indicated that the streamlined roof shape combined with a decrease in the angle of the windscreen reduces the drag coefficient by an average of 8% when compared to the traditional configuration. The study also aimed at integrating the above-mentioned improvements with a hybrid powertrain to increase the operating efficiency, especially in big cities where traffic is dense. This study is an intermediate step applicable to the real world towards efficiency optimization of the city bus fleet, without any change in the car's basic design.

Yuan and Wang [30] carried out a sophisticated numerical study to examine the impact of the non-flat bottom (Non-flat Underbody) of vehicles on the aerodynamics. The process simulation ANSYS Fluent software was employed to simulate a sedan underbody with the bottom body side, including technical cavities and mechanical holes. The SST K-ω model was adopted by the team, coupled with a high-density mesh in the turbulence regions beneath the vehicle. The study found that these door sills create floor vortices at the car’s base, which increase drag by 6.3 per cent, compared to if the car had a fully covered flat bottom. The report concluded that series cruise ships could save energy and travel more efficiently if their hull lines were as smooth as possible, and that the adoption of simplistic undersides as a low-cost improvement would benefit the shrinking cruise vessel budgets.

McKay and Gopalarathnam [31] The American project at Stanford University concentrated on lateral wing bends in sports cars and developed a three-dimensional model of a coupe with adjustable side wings. They applied the LES advanced CFD technology to model the flow around the wings at different angles of attack and sideslip. Results showed that compared to the straight wing, this 20° wing curved inside had 22% downforce gain and no increase in Drag. The present study provides further support for the new developments in interactive suites in which the garment can be fitted for road conditions and speed to improve performance in racing and high-speed civilian applications.

Hu et al. [32] Iranian researchers reveal research on the design analysis of rear air diffuser (Rear Diffuser) for a Family car, extracted from a sports car. Three configurations, 7°, 10°, and 14° degrees were simulated in the Star-CCM+ software. The 10° angle proved to be the best compromise between DR and increased downforce, with DR being reduced by 5.1%. The work has also indicated that, by fitting dispersants in common cars, it is possible to increase the vehicle aerodynamic efficiency without a substantial increase in the mass of the chassis, therefore confirming the possibility to integrate sports solutions in ordinary design in order to save fuel and to ensure vehicle stability.

Pisanti [33] performed research on the aerodynamics of the Solar Roof on electric vehicles. Two models are modeled, a standard smooth roof and a solar roof with differently sized raised modules. The RANS model was simulated on ANSYS Fluent, and it was analyzed the influence of the surface ripple on air flow in the speed range from 60 to 100 km/h was analyzed. It was noted that the coefficient of drag is increased up to 4,5% in a solar roof designed without a dynamic study. It is also suggested that if solar energy is to be employed, then a close to optimum module design should be considered to prevent decreased efficiencies due to air drag.

Wąsik and Skarka [34]. In the Saudi reference (King Fahd University of Petroleum and Minerals), the effect of using air vents behind the front wheels on SUV aerodynamics was evaluated. He employed the RANS model and dispersion to a three-dimensional model of an SUV. The model was analyzed using nozzles, while tested through nozzles of three different configurations. Results were that the inclined horizontal design to the rear reduced the stagnation of air behind the wheels and decreased the turbulence without them, and thus the drag coefficient was lowered by 4.2%, with an improvement in the air flow over the side doors. These researchers conclude that such vents may deliver a balance between the aerodynamic characteristics and aesthetics of large family cars.

Numerical analysis on automotive aerodynamics has experienced significant growth in the past few years as a result of both computing resources and the development of CFD computations. Mayes such as the influence from the front and rear wings, air dispersers, side skirts, side mirrors, roof designs, and changes in the rear. The effect of environmental factors, including crosswinds, ground settling, and urban obstructions, on the vehicle stability was also examined. Analyses included different classes of vehicles such as racing and Formula cars, city cars, and electric vehicles. The results demonstrated that enhancing the geometric shapes of external body parts of vehicles has a substantial effect on the reduction of the drag coefficient and the increment of the dynamic stability, as well as the increased level of fuel or electric power consumption performance of vehicles, as shown in Table (I). In some works, future applications such as flying or electric cars without front grilles have been considered; others have devoted particular attention to the interference of car parts with the surrounding environment of the flow using models, such as LES or RANS.

Table I. SUMMARY OF PREVIOUS STUDIES FOR AERODYNAMIC VEHICLES.

No.	Author (year)	Vehicle/component type	Numerical technique	Notable results
1	Dhumal *et al.* (2024)	Formula SAE car	CFD	Improved front shape reduced drag by 18%
2	Qin *et al.* (2024)	DrivAer models	CFD + Wind tunnel	High verification between simulation and actual testing
3	Wang (2024)	Flying car	CFD	Fin adjustment improves lift and reduces drag
4	Rangarajan *et al.* (2025)	Rear wing	CFD-RANS	The wing generates a downforce without a significant effect on Drag
5	Janson and Piechna (2015)	SUV	CFD	Rear splitter reduced air resistance by 12%
6	Grm and Batista (2017)	Passenger car	CFD + Crosswind	Side winds negatively affect stability by a large percentage
7	Monteiro *et al.* (2023)	F1 front wing (DRS)	CFD	Improved DRS reduces drag and maintains balance
8	Gohari *et al.* (2022)	Vehicle + side mirrors	CFD + Acoustic	30% noise reduction while maintaining dynamism
9	Dubey *et al.* (2013)	Various vehicles	CFD	Distractors are better than vortex at reducing drag
10	Krishna and Ravindra (2022)	Sedan	CFD + Crosswind	Wheel positioning reduced lateral deflection by 22%
11	Song *et al.* (2012)	Sedan	OpenFOAM	Flat butt reduces drag and improves balance
12	Dhiman *et al.* (2019)	Racing car	CFD	The 15° spacer angle gives the best balance between drag and down thrust
13	Sawaguchi and Takakura (2017)	Passenger car	CFD	Arc hood design minimizes forward turbulence
14	Xia and Huang (2024)	EV without front grille	CFD	The absence of a grille reduces drag by 8%
15	Kurec (2022)	Sports car	CFD	The height of the wing affects the downforce
16	Davkin and Law (2021)	Sports car	LES	Convertible wheels increase drag but improve cooling
17	Ghani *et al.* (2023)	Urban microcar	CFD	Improved design reduced drag by 15%
18	Liu *et al.* (2025)	Mirrorless car	CFD + Noise	Improved dynamics and reduced noise by 25%
19	Song *et al.* (2012)	Sedan	CFD	The introduction adjustment improved

				the flow by 20%
20	Ehirim *et al.* (2019)	Formula E	CFD	The ground dispersant improved the downforce by 27%
21	Hwang *et al.* (2016)	Sedan	CFD	Side skirts reduce drag and increase stability
22	Cheng and Mansor (2017)	Crossover SUV	CFD	Roof slope of 10° reduced drag
23	El-Sharkawy *et al.* (2011)	Electric vehicle	CFD + Thermal	Active shutters balance dynamics and cooling
24	Islam *et al.* (2013)	Hatchback	CFD	Vortex generators reduced drag by 13%
25	Williams (1985)	Urban taxi	CFD	Ceiling optimization reduced drag and increased advertising space
26	Yuan and Wang (2017)	Sedan	CFD	Non-flat bottom structure improves airflow
27	McKay and Gopalarathnam (2002)	Sports coupe	CFD	The curvature of the wings reduces drag and increases the stability of the car.
28	Hu *et al.* (2011)	Family sedan	CFD	The ideal rear diffuser angle was 12°
29	Pisanti (2015)	EV with solar roof	CFD	Solar roof increased lift but reduced drag
30	Wąsik and Skarka (2016)	SUV	CFD	Side air vents behind the wheels reduced drag by 10%

III. BASIC CONCEPTS IN VEHICLE AERODYNAMICS

Aerodynamics is an important aspect of modern vehicles and directly influences the vehicle's operation and fuel economy. Drag is the force of air resistance (a form of friction). It is the apparent force in the opposite direction of the relative motion of an object moving through a fluid. There are two parts to this draw [35].

- Pressure drags: Caused by an unequal distribution of pressure on the leading and trailing surfaces (face and back) of an object due to the air flow around it and the separation of the flow.
- Skin friction drag (Surface friction drag): It results from the friction between the air and the surface of the vehicle and is proportional to the roughness of the surface, length of the vehicle, and speed of the air.

How to decrease aerodynamic drag is more and more significant when we take the global tendency toward eco-friendly transportation technology. Devices to decrease the drag led to decreased fuel consumption and CO_2 emissions, as well as better acceleration and stability while driving on highways. The car has a better flowing line across the body, and many fewer separations and air vortices will appear. For this, the correct pressure distribution, flow velocity, and Boundary Layer transition from laminar to turbulent are required. But studies indicate that simply improving the aerodynamics could lower the drag resistance instituted by the airflow on some 10% to 50% depending on the car's drag area and the technology applied, a proportion that would translate directly in terms of performance and energy conservation [35].

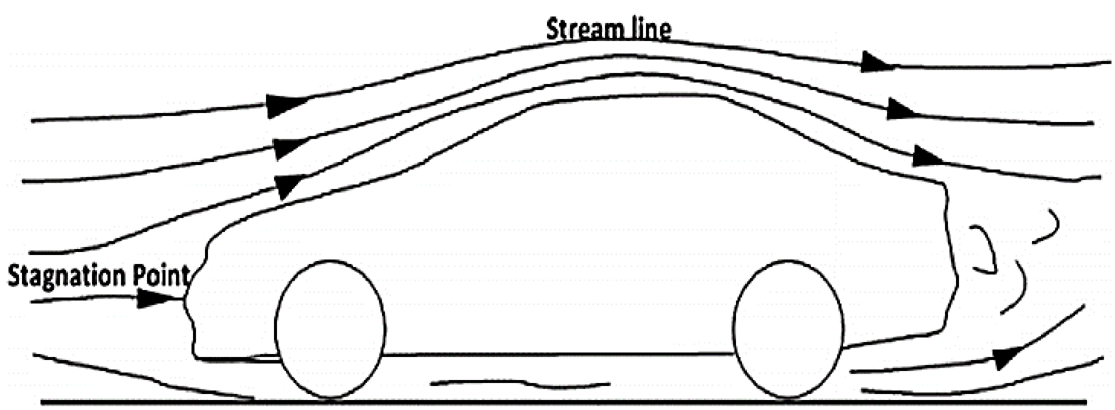

Figure 1, Streamline around vehicle [35].

IV. THEORETICAL FUNDAMENTALS OF COMPUTATIONAL FLUID DYNAMICS ANALYSIS (CFD)

Thanks to its ability to model the behavior of air around complex geometries without expensive physical experiments, computational fluid dynamics (CFD) has become a must-have in vehicle aerodynamics research. CFD methods are a set of techniques that are used to solve the equations that govern the movement of fluid in a computational method, which are the Navier-Stokes equations, whose equations are regarding the conservation of mass, momentum, and energy of the fluid media [36-38].

A. Governing Equations

- For mass conservation, use the continuity equation.
- Acceleration, viscosity, and pressure forces are all included in the momentum equations.
- When heat analysis is relevant, use the energy equation.

These equations are solved using various techniques such as the finite element method (FEM) or finite volume (FVM), which are often used in programs such as ANSYS Fluent, OpenFOAM, Star-CCM+, and others.

B. Models of Turbulence

Since most air flows around vehicles are turbulent, mathematical models are used to approximate the effects of turbulence, the most notable of which are:

- K-ε model (Standard k-epsilon): commonly used for its stability and speed of dissolution.
- Model SST k-ω (Shear Stress Transport): more accurate near walls and air separations.
- LES model (Large Eddy Simulation): offers high accuracy but requires great computing power.
- RANS (Reynolds Averaged Navier–Stokes) model: used in general studies with reduced calculation time.

C. Mesh/Grid of Numerical Simulation

The quality of the numerical grid, which partitions the engineering field, is a good CFD simulation is based on the quality of the numerical grid.

- A network with a regular or irregular structure (Str/Unstr)
- 3D elements (Tetrahedral, Hexahedral, Prism layers).

Adding more points to the sensitive areas (forelimbs, mirrors, detachments) greatly increases outcomes.

D. Applying Boundary Conditions

To ensure the correctness of the simulation, it is determined:

- Velocity or inward flow
- Pressure or outflow
- Hard surfaces of the vehicle (No-slip condition)
- Symmetry conditions or invisible walls

These settings are used to simulate various scenarios such as high speeds, the effect of crosswinds, or the presence of dynamic additions to the structure.

V. Effect of the External Shape of a Vehicle on Aerodynamic Drag

The car's external vehicle geometry is one of the most significant influencing parameters for the value of the air drag resistance, because it determines the way the air meets the car (how the air marries the car) and the locations where separation and the regions with creation of the torques occur. Aerodynamic shapes minimize the low-pressure areas behind the vehicle, which in turn ease the drag of the vacuum that pulls the vehicle from the increased pressure that's pushing. The most influential design elements [39]:

- The front end (Nose): rounded and sleek front edges make the front air looser.
- Roof and Rear Slope – sloped gradually to reduce the flow separation, especially for the fastback cars.
- Tail Nozzle tails: Sharp and/or long nozzles tend to minimize eddies and loss of pressure.
- Exposed wheels and side mirrors: They generate a lot of aerodynamic turbulence and can either be covered or adjusted to less flagrant positions to reduce drag.

So, it was not surprising to learn that changing the shape of the car can have a 30% drag Cd coefficient. It's a number that's a big factor in the EROEI. For instance, changing from a boxy to a semi-streamline design decreases the Cd by a factor of 0.4 to a Cd <0.25.

VI. Design techniques to reduce aerodynamic drag

In modern cars, the decrease of the air drag is achieved by a variety of engineering designs that enable as much as possible smooth air flow around the car body, which in turn will minimize the zones of separation and vortex intensity. The most common of these is the spoiler and diffuser. The rear spoiler redirects airflow up straight into a diffuser, which decreases airflow and pressure under the vehicle, down and out the back of the vehicle, which reduces the air vortices down the side of the car. As for the floor diffuser, it is fitted at the rear of the car and serves to speed up air under the car, explaining that it generates less negative pressure at the rear and enhances the dynamic performance. Bottom panels are the other practical answer to decreasing air resistance, as they help with some shielding of baked mechanicals underneath the car, which minimizes poor air flow under the car. Additionally, a more aerodynamic shape for the side mirrors or their replacement with cameras, along with streamlined wheel covers, can further reduce the drag created by these sticking-out parts. One of the most important advances on this basis is the use of active aerodynamics ventilation systems that can automatically change the shape of parts of a vehicle, or the shape of ventilation openings in a vehicle, according to the vehicle's speed and driving conditions. It is therefore possible with such systems to obtain an optimal compromise between air resistance and cooling or stability, providing a vehicle having variable and multifunction performance. The combination of these technologies and their coordinated use leads to an increase in total aerodynamic performance and to a reduction in fuel or energy consumption, thus making the vehicle more efficient and environmentally friendly [39].

VII. Relationship Between Aerodynamic Drag and Fuel Efficiency

Fuel efficiency Aerodynamic drag is a key component in the overall drag effect (drag) of the car; it is the reason the car uses more fuel at speeds greater than 60 km/h (38 mph). When the vehicle drives over 80 km/h, air resistance is the main resistance to energy consumption, up for more than 50% of total resistance forces. In this way, improving the aerodynamic performance through a smaller drag coefficient (Cd) not only helps to reduce the energy demand for the vehicle movement (what is directly translated into fuel economy in vehicles that still run with conventional fuels, or, in the case of electric vehicles, it is translated into operations for longer distances between charges). We know that 1% to 2% of better fuel efficiency can be achieved in such a case of lowering the drag coefficient by 0.01, which is a valuable effect for the long term. "For electric vehicles, the effect is even greater because lowering drag improves fuel economy and increases the driving range for a given battery size." This is why nowadays electric car manufacturers design sleek exteriors with the Cd figure falling below 0.24. Furthermore, the drag reduction not only brings technical but also environmental and economic advantages. It works to reduce fuel consumption emissions, meet global environmental standards, and lower the carbon footprint of the traffic domain. It also leads to economic savings for fleets and operators where fuel consumption is high, and thus makes technologies associated with the improvement of aerodynamic performance a key decision when designing modern vehicles [40].

VIII. Challenges and Future Prospects for Improving Aerodynamic Drag Using CFD

Computational Fluid Dynamics (CFD) capabilities have developed rapidly in recent times for the evaluation of aerodynamic drag of vehicles; however, there are technical and scientific challenges confronting researchers and designers in this area. One of the main issues of concern is to find a trade-off between the accuracy of the results and the computational time spent (or effort), which is especially difficult when employing more sophisticated perturbation models like LES and DNS, which have a high computing resource and long processing time demand. In addition, the realization of an accurate and complex numerical grid system consisting of small or moving parts (such as wheels, mirrors, etc.) is a sensitive issue that also has a direct influence on the stability and accuracy of numerical solutions. Validation of CFD prediction and comparing it with the practical experiment in a wind tunnel or on the road is still the preliminary step to guarantee the reliability of the prediction, in which the numerical modeling and testing should be integrated. Moreover, the simulation tools should be further developed considering nonlinear, multiphase flows, or complex thermal effects handling. Notwithstanding these challenges, the passage of time bodes well because the developments are oriented towards the integration of CFD with artificial intelligence (AI) and machine learning (ML) algorithms, which would accelerate the design process and enhance the quality of results. In addition, the utilization of cloud computing approaches and parallel systems also allows computational time to be significantly cut down. Furthermore, the bridge between CFD and generative design is anticipated to help unlock novel forms that are not achievable using traditional techniques, and more aerodynamically efficient ones at that. Hence, the minimization of aerodynamic drag via CFD is now more than ever a fundamental tool and a nucleus for the development of modern vehicles and a cornerstone towards a green and energy-conscious design in the transportation industry.

IX. Effect of Angle of Rear Diffuser on Air Flow and Improvement of Downforce

The rear diffuser is the key to the aerodynamics of a car, and in passenger vehicles, it uses airflow to further reduce air turbulence and drag by creating static air that is channeled out through the diffuser, thereby improving the grip levels of the car. When the diffuser is configured at substantial angles (for example, 20°), the flow stream becomes semi-discrete beneath the vehicle and results in the formation of stable side stream vortices, reducing the pressure under the vehicle, which increases the downforce on the ground and enhances the vehicle's traction. This rolling configuration leads to a partial separation of the flow rather than a full separation, which improves aerodynamic efficiency and lowers fuel consumption. On the other hand, excessive angles (more than 30 degrees) cause the flow to separate behind the vehicle prematurely, hence more drag force and poor aerodynamic values. So, to optimize aero foil and aerodynamic stability at high speed, the correct diffuser angle is an important feature [41].

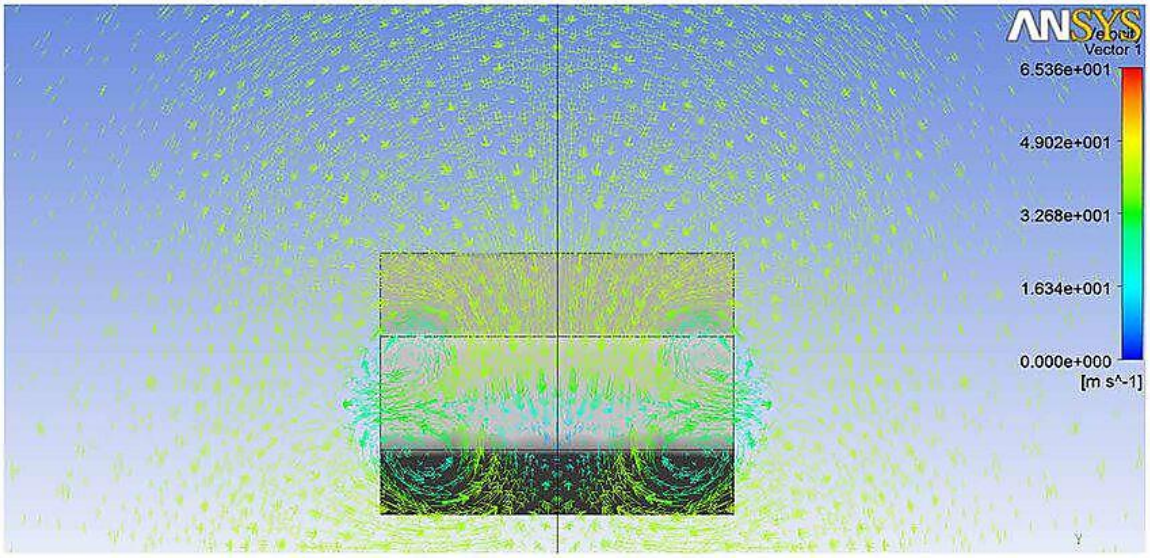

Figure 2, Directions of Velocity lines and vortices form below the diffuser at an angle of 20° at the rear of the vehicle [41]

X. Efficient aerodynamic systems: recent developments and numerical vision

Recently, over the past period, the inclusion of efficient aerodynamic solutions in modern vehicle design has experienced tremendous development, which provides benefits in that the solutions adapt to changing driving conditions and dynamically increase the aerodynamic performance. They differ from the normal stationary such devices by their automatic regulation according to the speed of travel, the cooling necessary, or the state of the road, the resistance of the air being minimized, and the aerodynamic stability reinforced. Noteworthy features in said technologies include active front air shutters that automatically close at lower speeds to minimize air drag and open at higher speeds to assist in airflow around the vehicle body, as shown in Figure (3). The numerical simulation results conducted using CFD Technologies showed that the employment of these shutters in the closed position decreases the drag coefficient by over 6%, resulting in an improvement in fuel efficiency and reducing aerodynamic pollutants [42].

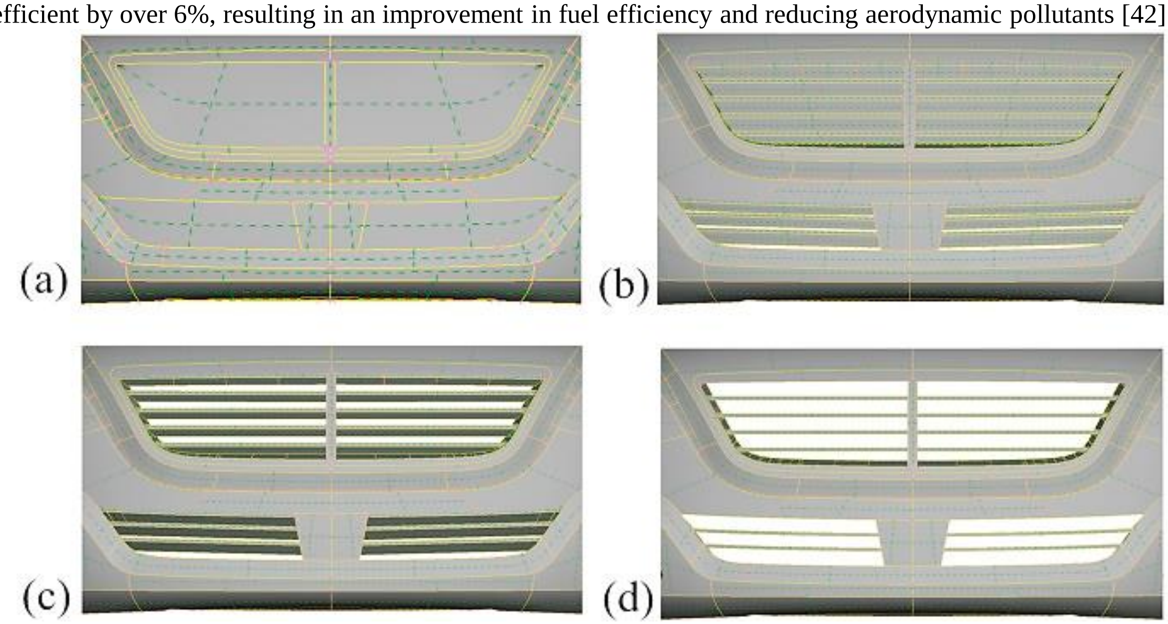

Figure 3, A model of an active front air shutter at different openings: (a) 0°aperture ،(b) 18°aperture ،(c) 54°aperture ،and (d) 90°aperture [42].

XI. Conclusions

This investigation illustrates that air drag reduction and its influence on vehicle economy are important. CFD has an essential function of design enhancement and drag decrease. The ongoing development of numerical simulation and artificial intelligence makes more economical, sustainable designs within reach, despite the current difficulties. The most important conclusions could be summarized:

- Drag reduction has been a problem of concern in the design and evaluation of vehicles, specifically in aircraft, due to its strong effect on fuel efficiency and performance, with an increase in significance as the operating speed increases.
- Different design techniques (like spoilers, baseboards, and active ventilation) can provide an increase in the mass flow inside the vehicle, and also decrease separation zones; this has the effect of reducing the drag coefficient and improving its VHDS (Vehicle Handling Dynamical Stability).

- Computational Fluid Dynamics (CFD) is a powerful means of predicting the effects of design changes on aerodynamic drag, capable of reducing the time and expense associated with the traditional wind-tunnel experiment.
- Practical works show that design optimizations based on CFD simulations can decrease the drag coefficient of a vehicle by around 30%, which has a direct advantage on fuel consumption or electric vehicle range.
- However, the accuracy of numerical models and the computation time are still major concerns. Artificial intelligence and machine learning based technologies hold great promise for accelerating and enhancing the simulations and design.
- Such a synergy between numerical simulations and physical tests is mandatory to validate the results and verify their accuracy and correctness, towards the design of more efficient and environmentally friendly vehicles.
- Using a simulation based on an effective aerodynamic system, especially active front air shutters, one can demonstrate that the use of that kind of technology directly affects signalize on the flow of the vehicle, at the result reducing its drag coefficient, in some cases, more than 6% that could be more reactive of the fuel efficiency of the vehicle and its aerodynamic stability. These results indicate that the introduction of clever controllable systems in the aerodynamic design of cars of the present time is a good choice and a promising way to improve the overall performance of cars, especially at high speeds.

List of Abbreviations

Cd	Drag Coefficient
CFD	Computational Fluid Dynamics
EV	Electric Vehicle
FEM	Finite Element Method
FVM	Finite Volume Method
LES	Large Eddy Simulation
RANS	Reynolds Averaged Navier–Stokes
SST	Shear Stress Transport

List of Symbols

Symbol	Quantity	Unit	Description
$\nabla\cdot\vec{V}$	Divergence of the velocity field	1/s	Used in the continuity equation (mass conservation)
∇P	Pressure gradient	Pa/m	Appears in the Navier–Stokes equations
A	Reference area	m^2	Frontal area of the vehicle
C_D	Drag coefficient	Dimensionless	Represents the ratio of the drag force to dynamic pressure and area
D_F	Drag force	N (Newton)	Resistance force along the flow direction
g	Gravitational acceleration	m/s^2	Typically, 9.81
C_L	Lift coefficient	Dimensionless	Ratio of lift force to dynamic pressure and area
F_L	Lift force	N (Newton)	Force perpendicular to the flow direction
P	Static pressure	Pa (Pascal)	Fluid pressure exerted in static conditions
Re	Reynolds number	Dimensionless	Ratio of inertial to viscous forces
V	Flow velocity	m/s	Speed of air relative to the vehicle
μ	Dynamic viscosity	Pa· s	Measures fluid's resistance to shear or flow
ρ	Air density	kg/m^3	Mass per unit volume of air
τ	Shear stress	Pa	Force per unit area from fluid viscosity

References

[1] M. A. Öztemel, F. Aktaş, and N. Yücel, "Aerodynamic analysis of car rear spoiler with computational fluid dynamics for different angles and profiles," *Int. J. Energy Stud.*, vol. 8, no. 4, pp. 649–665, 2023. [Online]. Available: https://doi.org/10.58559/ijes.1362690

[2] T. D. Duc, D. P. Xuan, and V. N. Duy, "Aerodynamics Simulation of Prototype Car Based on CFD Technology," 2023. [Online]. Available: [source not specified—insert if available]

[3] J. Li, F. Liu, and L. Wang, "Design and aerodynamic characteristics analysis of an electric racecar body based on CFD," *World Electr. Veh. J.*, vol. 15, no. 5, p. 192, 2024. [Online]. Available: https://doi.org/10.3390/wevj15050192

[4] X. Liang, X. Gao, T. Gu, and X. Jia, "Aerodynamic Drag Coefficient Analysis of Heavy-Duty Vehicle Platoons: A Hybrid Approach Integrating Wind Tunnel Experiments and CFD Simulations," *Energies*, vol. 17, no. 24, p. 6383, 2024. [Online]. Available: https://doi.org/10.3390/en17246383

[5] A. Dhumal, N. Ambhore, P. Tamkhade, A. Marne, and N. Mujawar, "Numerical Optimization for Aerodynamic Performance of Nose Cone of FSAE Vehicle through CFD," *CFD Lett.*, vol. 16, pp. 161–171, 2024. doi: 10.37934/cfdl.15.11.161171

[6] P. Qin, A. Ricci, and B. Blocken, "CFD simulation of aerodynamic forces on the DrivAer car model: Impact of computational parameters," *J. Wind Eng. Ind. Aerodyn.*, vol. 248, p. 105711, 2024. doi: 10.1016/j.jweia.2024.105711

[7] Y. Wang, "Aerodynamic Optimization and CFD Simulation of Flying Cars," *J. Adv. Eng. Technol.*, vol. 1, no. 2, pp. 1–14, 2024. doi: 10.62177/jaet.v1i2.47

[8] K. Rangarajan, B. Pushpananthan, L. Anumolu, K. Selvakumar, and S. S. Jayakumar, *Numerical Study of Aerodynamic Flow and the Effects of Spoiler Wings on a Formula Student Race Car*, SAE Technical Paper 2024-01-5247, 2025. doi: 10.4271/2024-01-5247

[9] T. Janson and J. Piechna, "Numerical analysis of aerodynamic characteristics of a high-speed car with movable bodywork elements," *Arch. Mech. Eng.*, vol. 62, no. 4, 2015. doi: 10.1515/meceng-2015-0026

[10] A. Grm and M. Batista, "Vehicle aerodynamic stability analysis under high crosswinds," *Stroj. Vestn.-J. Mech. Eng.*, vol. 63, no. 3, pp. 191–200, 2017.

[11] C. Monteiro, M. Brito, and D. F. da C. Vieira, *Numerical Iterative Analysis of Drag Reduction System for a Racing Car Rear Wing*, SAE Technical Paper 2023-01-5079, 2023. doi: 10.4271/2023-01-5079

[12] M. Gohari, R. Norozi, and A. H. Aghdam, "Evaluation and optimization of the aerodynamic noise reduction of vehicle side view mirrors: experimental and numerical study," *HighTech Innov. J.*, vol. 3, no. 1, pp. 73–84, 2022.

[13] [13] A. Dubey, S. Chheniya, and A. Jadhav, "Effect of Vortex generators on Aerodynamics of a Car: CFD Analysis," *Int. J. Innov. Eng. Technol.*, vol. 2, no. 1, pp. 137–144, 2013.

[14] P. M. V. Krishna and B. Ravindra, "Effect of Crosswind on Vehicle Dynamics," in *Conf. Fluid Mech. Fluid Power*, Singapore: Springer, 2022, pp. 593–604.

[15] K.-S. Song et al., "Aerodynamic design optimization of rear body shapes of a sedan for drag reduction," *Int. J. Automot. Technol.*, vol. 13, no. 6, pp. 905–914, 2012.

[16] V. Dhiman, T. Joshi, and G. Singh, "Effect of Front Slant Angle on Aerodynamics of a Car," in *Adv. Eng. Des.: Sel. Proc. FLAME 2018*, Singapore: Springer, 2019, pp. 641–653.

[17] T. Sawaguchi and Y. Takakura, "Aerodynamic effects by cooling flows within engine room of a car model," *IOP Conf. Ser.: Mater. Sci. Eng.*, vol. 249, no. 1, p. 012012, 2017.

[18] Z. Xia and M. Huang, "Optimizing the Aerodynamic Efficiency of Electric Vehicles via Streamlined Design: A Computational Fluid Dynamics Approach," *Int. J. Heat Technol.*, vol. 42, no. 3, 2024.

[19] K. Kurec, "Numerical study of the sports car aerodynamic enhancements," *Energies*, vol. 15, no. 18, p. 6668, 2022. doi: 10.3390/en15186668

[20] L. Davkin and C. Law, "Aerodynamic Effect of Wheel Fairings on the Wake of a Formula One Car," in *MATEC Web Conf.*, vol. 347, p. 00008, 2021.

[21] I. A. Ghani, R. Hassan, I. Amin, A. Suhan, K. Husni, D. H. Didane, and B. Manshoor, "Computational fluid dynamics (CFD) study on the aerodynamic of truck," *J. Des. Sustain. Environ.*, vol. 5, no. 2, pp. 23–27, 2023.

[22] X. Liu, Z. Gao, C. Ren, X. Yang, and P. Lu, "A study on the aerodynamic noise of automotive rearview mirrors relative to different side window openings," *J. Phys.: Conf. Ser.*, vol. 3021, no. 1, p. 012047, 2025.

[23] K.-S. Song, S.-O. Kang, S.-O. Jun, H.-I. Park, J.-D. Kee, K.-H. Kim, and D.-H. Lee, "Aerodynamic design optimization of rear body shapes of a sedan for drag reduction," *Int. J. Automot. Technol.*, vol. 13, no. 6, pp. 905–914, 2012.

[24] O. H. Ehirim, K. Knowles, and A. J. Saddington, "A review of ground-effect diffuser aerodynamics," *J. Fluids Eng.*, vol. 141, no. 2, p. 020801, 2019. doi: 10.1115/1.4040501

[25] B. G. Hwang, S. Lee, E. J. Lee, J. J. Kim, M. Kim, D. You, and S. J. Lee, "Reduction of drag in heavy vehicles with two different types of advanced side skirts," *J. Wind Eng. Ind. Aerodyn.*, vol. 155, pp. 36–46, 2016. doi: 10.1016/j.jweia.2016.04.009

[26] S.-Y. Cheng and S. Mansor, "Influence of rear-roof spoiler on the aerodynamic performance of hatchback vehicle," in *MATEC Web Conf.*, vol. 90, p. 01027, 2017. doi: 10.1051/matecconf/20179001027

[27] A. E. El-Sharkawy, J. C. Kamrad, T. H. Lounsberry, G. L. Baker, and S. S. Rahman, "Evaluation of impact of active grille shutter on vehicle thermal management," *SAE Int. J. Mater. Manuf.*, vol. 4, no. 1, pp. 1244–1254, 2011.

[28] M. R. Islam, M. A. Hossain, M. Mashud, and M. T. I. Gias, "Drag reduction of a car by using vortex generator," *Int. J. Sci. Eng. Res.*, vol. 4, no. 7, pp. 1298–1302, 2013.
[29] J. Williams, "An automotive front-end design approach for improved aerodynamics and cooling," *SAE Trans.*, pp. 662–685, 1985.
[30] Z. Yuan and Y. Wang, "Effect of underbody structure on aerodynamic drag and optimization," *J. Meas. Eng.*, vol. 5, no. 3, pp. 194–204, 2017.
[31] N. J. McKay and A. Gopalarathnam, "The effects of wing aerodynamics on race vehicle performance," *SAE Trans.*, pp. 2254–2263, 2002.
[32] X. Hu, R. Zhang, J. Ye, X. Yan, and Z. Zhao, "Influence of different diffuser angle on Sedan's aerodynamic characteristics," *Phys. Procedia*, vol. 22, pp. 239–245, 2011. doi: 10.1016/j.phpro.2011.11.038
[33] C. Pisanti, "Design and energetic evaluation of a mobile photovoltaic roof for cars," *Energy Procedia*, vol. 81, pp. 182–192, 2015. doi: 10.1016/j.egypro.2015.12.073
[34] M. Wąsik and W. Skarka, "Aerodynamic features optimization of front wheels surroundings for energy efficient car," in *Transdisciplinary Engineering: Crossing Boundaries*, IOS Press, pp. 483–492, 2016.
[35] G. Sivaraj, K. M. Parammasivam, M. S. Prasath, P. Vadivelu, and D. Lakshmanan, "Flow analysis of rear end body shape of the vehicle for better aerodynamic performance," *Materials Today: Proceedings*, vol. 47, pp. 2175–2181, 2021. [Online]. Available: https://doi.org/10.1016/j.matpr.2021.05.521
[36] S. A. Ali, E. S. Barrak, N. J. Alrikaby, and M. R. Hameed, "Numerical Study of Thermal-Hydraulic Performance of Forced Convection Heat Transfer in Dimple Surface Pipe with Different Shapes using Commercial CFD Code," *Heat Transfer*, vol. 125, no. 2, pp. 1–15, 2025. [Online]. Available: https://doi.org/10.37934/arfmts.125.2.115
[37] S. A. Ali and M. R. Hameed, "Computational Fluid Dynamics Analysis for Heat Transfer Enhancement in Single Horizontal Pipe Heat Exchanger Full Filled Using Different Types of Porous Material," *Journal of Research and Applications in Mechanical Engineering*, vol. 13, no. 2, 2025.
[38] A. Al-Akam, H. K. Kadhim, S. A. Ali, and A. M. Al Juboori, "Numerical Analysis for the Airflow Behaviour around Vortex Generators Used for Air-Cooling Technologies Considering Rotation," *Computational Fluid Dynamics Letters*, vol. 17, no. 9, pp. 127–144, 2025. [Online]. Available: https://doi.org/10.37934/cfdl.17.9.127144
[39] A. N. M. M. Islam and M. Z. Abedin, "Review on aerodynamic drag reduction of vehicles," *International Journal of Engineering Materials and Manufacture*, vol. 4, no. 1, pp. 1–14, 2019.
[40] C. Fernandez, S. Shofiah, and A. A. Nampira, "The Effect of Aerodynamic Design on Fuel Efficiency in Commercial Vehicles," *Journal of Moeslim Research Technik*, vol. 2, no. 2, pp. 87–95, 2025. [Online]. Available: https://doi.org/10.70177/technik.v2i2.1936
[41] D. J. S. Aulakh, "Effect of underbody diffuser on the aerodynamic drag of vehicles in convoy," *Cogent Engineering*, vol. 3, no. 1, p. 1230310, 2016, doi: 10.1080/23311916.2016.1230310.
[42] J. Li, Y. Deng, Y. Wang, C. Su, and X. Liu, "CFD-based research on control strategy of the opening of active grille shutter on automobile," *Case Studies in Thermal Engineering*, vol. 12, pp. 390–395, 2018, doi: 10.1016/j.csite.2018.05.009.