Advancing Aerodynamic Optimization: Reducing Drag and Fuel Consumption in Modern Vehicles

Yichen Hu [1], Bochuan Jiao [2, *]

[1] Shanghai International High School of BANZ Shanghai, China

[2] Farragut Admiral Academy Tianjin Fayao Education Tianjin, China

* Corresponding Author Email: 15510838159@163.com, eason_hu2008@163.com

Abstract. In the developed world, almost every household owns a car. Despite the increasing prevalence of electric vehicles, gasoline-powered vehicles continue to contribute significantly to emissions. Air and fluid resistance are the primary factors contributing to poor fuel economy; however, these forces are inherent to vehicle operation and cannot be eliminated. Several prominent methods for reducing the drag have been proposed and compared. It was found that pneumatic control systems could reduce the drag by 20% to 30%. For the GT model, which is both inherently optimized and a high-performance racing car, the maximum observed drag reduction was 16.53% when tail and diffuser modifications were applied. The physical modeling method is a fundamental approach for obtaining aerodynamic data. To show that the relationship between drag force and fuel consumption of cars can be revealed by performing computational fluid dynamics (CFD) to provide aerodynamic design. However, considering ride comfort, the effectiveness of these modifications is considerably impacted owing to their complex configurations. Based on existing data, it is imperative to further develop this body of knowledge to meet the rising demand and stringent standards in the future, particularly as AI-driven intelligent driving technologies rapidly advance.

Keywords: automotive aerodynamics; reference models; experimental fluid dynamics; computational fluid dynamics (CFD).

1. Introduction

1.1. Physical model research methods

The automotive aerodynamics model is moderately abstracted and simplified in shape, and the geometric features of the same or similar to the real car are retained to ensure that the fluid physical features of the model can be mapped to the real car under the premise of similarity. In this way, through the study of the fluid physics of the model, the aerodynamic mechanism and laws of the model can be clarified, and the corresponding theory can be established to understand the aerodynamic nature of the real vehicle. These understandings and theories can then be used to provide support for the innovative design of real vehicles. It is important to note that in the automotive industry, this model is called the Standard Model or Reference Model of Automotive Aerodynamics.

1.2. Computational Fluid Dynamics method

Among the various factors that influence a vehicle’s overall efficiency, aerodynamic drag is a critical consideration. Computational Fluid Dynamics (CFD) is an innovative method that calculates fluid flow around objects, including vehicles, by solving complex mathematical equations governing fluid behavior to analyze drag forces using ANSYS Fluent software and many other packages. CFD offers an invaluable virtual testing ground for calculating the vehicle drag coefficients. CFD predicts aerodynamic forces by simulating airflow over vehicle surfaces, enabling researchers to precisely quantify drag. CFD is one of the most commonly used techniques for analyzing vehicle drag owing to its accuracy and cost efficiency.

2. Review

2.1. Method 1

The authors, Mukut emphasized the importance of reducing aerodynamic drag in vehicles in response to fuel consumption and environmental concerns. The authors mentioned the reduction in aerodynamic drag in vehicles and its impact on fuel consumption and environmental pollution. With concerns about environmental pollution, the automotive industry is still working to produce vehicles with good aerodynamics to lower emissions and improve fuel economy.

1) Principle: Different flow control techniques were used in the simulated wind tunnel for comparison.

2) Observation: The authors made two main observations and conducted related experiments.

First, the effect of aerodynamic resistance: The aerodynamic resistance of a vehicle is proportional to the square of its speed. When a vehicle is traveling at high speed, approximately 70% of the resistance comes from the resistance of the overall vehicle [1]. On the highway, more than half of the fuel consumption is due to aerodynamic drag.

Second, flow control technology: The authors discussed the effect of active and passive flow control technology on reducing aerodynamic drag, and both control methods have their advantages and disadvantages. Passive flow control methods affect the flow field by adding extra shapes to the vehicle but sometimes add extra drag and are therefore relatively less controllable. Active flow control systems are activated only when needed and are relatively simple to install but require additional energy consumption.

3)Result of measurements: Details of experimental issues.

First, the passive flow control method: The authors mentioned additional modifications to the shape of the car body to affect the flow field and thus achieve drag reduction. For example, a "bottom diffuser" can reduce the air pressure by increasing the flow rate of air under the vehicle, thereby improving the stability of the vehicle. Specific data showed that after the transformation of the bottom diffuser, the aerodynamic resistance of the car is reduced by approximately 10%, and the station wagon is reduced by 2-3% [1].

The second type is an active flow control system. The authors mentioned that the aerodynamic drag reduction of active flow control systems can reach 20% using technologies such as pulse injection [1]. These data are visualized through charts and graphs, which enhance the persuasiveness of the argument.

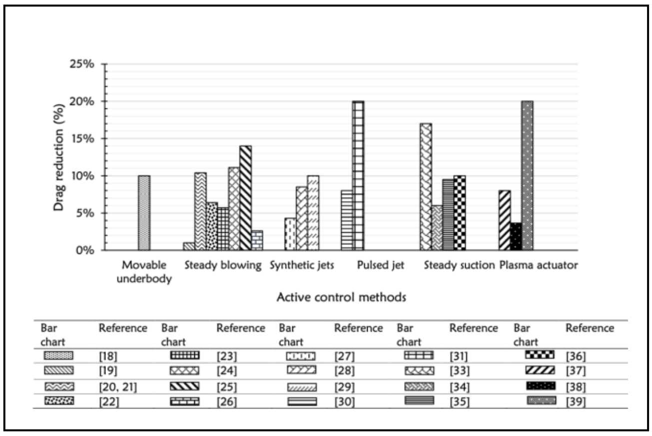


Fig. 1 Active control methods [1].

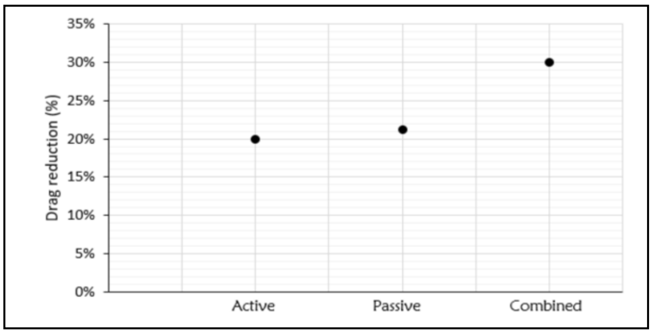


Fig. 2 Control methods [1].

In terms of the combined flow control technique, the authors showed that the combined flow control technique using the jet nozzle and porous layer can achieve up to 30% reduction in aerodynamic drag [1]. These data highlight the advantages of combined technology, demonstrating its potential for drag reduction.

The results show that the resistance reduction effects of active, passive, and combined control systems can reach 20%, 21.2%, and 30%, respectively [1].

4) Summary: Body modification, as a passive technique to enhance aerodynamic performance by changing the shape and bottom geometry of a vehicle, can effectively reduce fuel consumption and is by far the most effective and widely used method. Whether from the point of view of the lack of more efficient energy and the current perfection of the body design, it is the best choice for current designers. A shape optimization study using an artificial neural network (ANN) shows that the aerodynamic drag coefficient of the optimized car is reduced by 5.639%, and the aerodynamic drag is reduced by 13.23% after the six parameters of the simplified car are optimized [1]. These optimization results provide empirical support for the importance of the shape design. These examples and data are of particular significance as they not only provide specific data on the quantification of drag reduction effects but also demonstrate the effectiveness and applicability of different flow control methods. These empirical results make the discussion more authoritative and reliable and provide a reference for future research and application. The actual effects of aerodynamic design on fuel consumption and environmental impact were also more clearly expressed.

2.2. Method 2

Another study explored the best combination of pneumatic devices to contribute to the best performance of racing cars under emission restrictions. The four authors, Devang S. Nath, Prashant Chandra Pujari, Amit Jain and Vikas Rastogi, mainly explored how the application of different pneumatic devices in racing cars to reduce air resistance can improve vehicle performance and fuel economy. The authors pointed out that with the gradual depletion of natural resources and increasingly stringent environmental policies, automakers are under pressure to produce efficient, low-emission vehicles. Therefore, the improvement of aerodynamic performance is considered an important way to improve the speed and efficiency of the car.

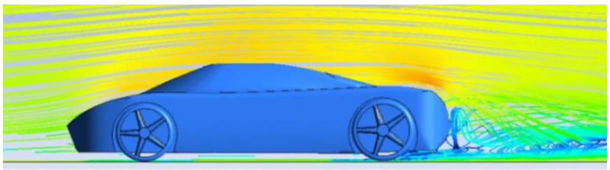
1) Principle and results: The pneumatic devices mentioned by the authors include rear wing, spoiler, diffuser, and fins. The impact of the application of these devices on the performance of a car is mainly reflected in the following aspects.

First, reduce drag: By optimizing the aerodynamic shape and flow of the vehicle, reduce airflow separation, and thus reduce the drag coefficient (C_D). For example, when GT models were equipped with a spoiler and diffuser, the authors observed a maximum drag reduction of 16.53% [2].

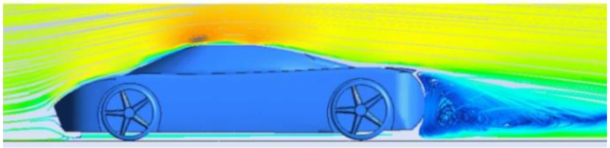
Second, improve stability: Certain devices, such as fins, can improve stability at high speeds, helping the vehicle maintain better handling at high speeds. Increased downforce: The rear wing is designed to increase drag, but its main function is to provide downforce (negative lift), which improves the vehicle's grip and cornering speed at high speeds.

Finally, flow control: by reducing the vortex area in the tail, the turbulence is reduced, thereby further reducing the drag.

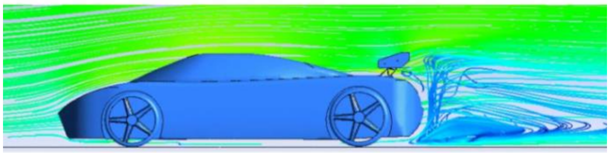
The airflow changes caused by each accessory are shown in the figure below:



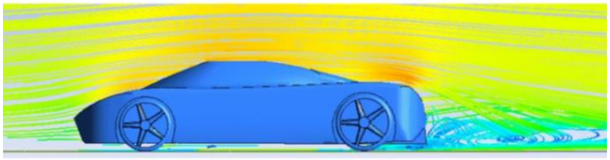
(a) GT baseline



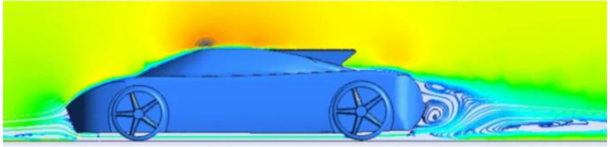
(b) GT with spoiler



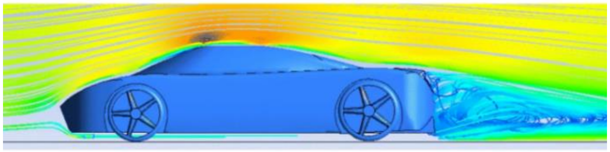
(c) GT with wing



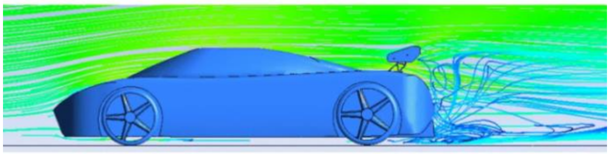
(d) GT with diffuser



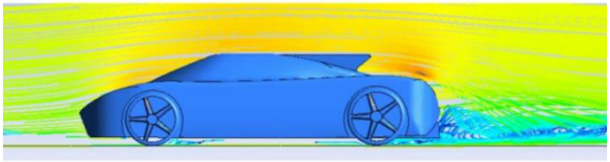
(e) GT with fins



(f) GT with spoiler and diffuser



(g) GT with wing and diffuser



(h) GT with fins and diffuser

Fig. 3 Different effects [2].

2) Summary: Among all combinations of accessories, the pneumatic combination that most effectively reduces the drag of the car is the rear wing and diffuser. As mentioned earlier, the GT model was observed to have a maximum drag reduction rate of 16.53% when a tail and diffuser were used [2]. The physics behind the combination of these pneumatic devices includes the following points.

The first is flow attachment: To reduce drag, the air flow should be attached to the surface of the vehicle for as long as possible. The streamlined body design reduces the flow separation, thereby reducing turbulence and drag.

The second is the tail wing: the main function of the tail wing is to provide a downforce (negative lift), thereby increasing the traction of the vehicle at high speed and preventing the vehicle from losing control at high speed. Although the rear wing increases drag, the downforce it provides is crucial for improving the stability and handling of the vehicle.

Finally, the diffuser reduces the pressure under the car by accelerating the airflow under the car, thus reducing the drag. According to Bernoulli's principle, a fast-flowing gas creates a lower pressure, which helps to reduce the low-pressure zone behind the vehicle, which in turn reduces drag. In summary, the combination of the tail and diffuser can not only effectively reduce drag but also improve the stability and handling performance of the vehicle.

2.3. Method 3

The authors J. S. LIU and S. J. XU and Q. Y. WANG focused on Ahmed's physical model in automotive aerodynamics research. Combined with the Ahmed model, the active and passive control of both sides of the tail was studied. Since the 80s of the 20th centuries, more than a dozen models with different characteristics have appeared in the world, such as Ahmed, SAE, Rover, Davis, DOCTON, FordBlock, GM, ASMO, RMIIT, Chrysler, MIRA, DrivAer model, etc. Among these, the most famous and popular are the Ahmed, MIRA, and DrivAer models (Fig. 4).

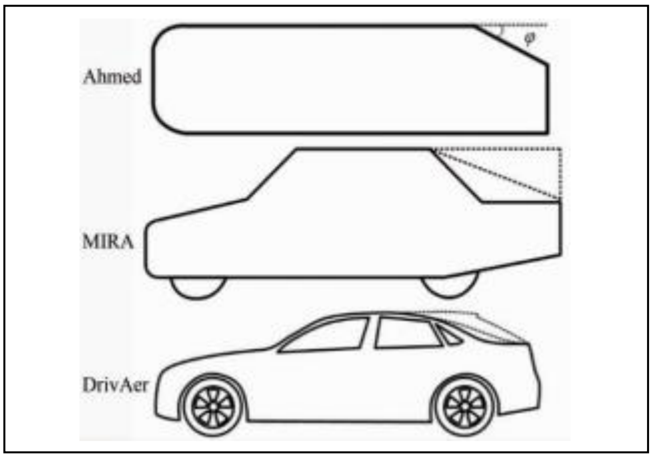
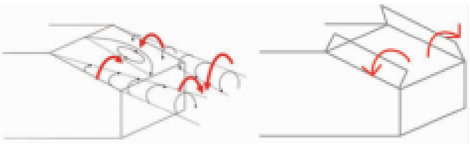


Fig. 4 Ahmed, MIRA and DrivAer models [3].

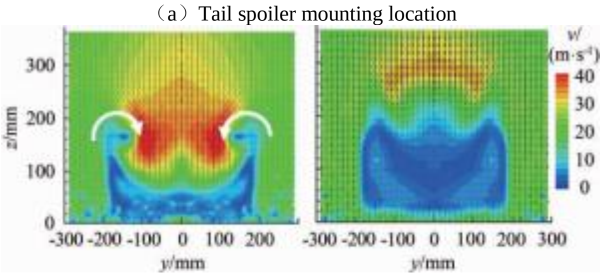
1) Principle: As the airflow winds around the model, the flow separates at different parts of the model owing to changes in the geometry of the model, inertia, and viscosity of the fluid. The separated flow generally contains flow structures with different shear strengths and scales, forming a flow containing vortices or waves as typical characteristic structures and vortex streets. The geometric and kinematic characteristics of the vortex reflect the distribution and change in pressure on the model, and thus, the change in aerodynamic force.

Applications: Based on the Ahmed model, active and passive control applications are on both sides of the tail.

2) Passive control: based on a full understanding of the flow characteristics of the surface of the Ahmed model, added spoilers on both sides of the tail to passively control the main flow vortex (see Fig. 5). When the angle between the spoiler and back is close to 70°, the spoiler affects the formation of the flow vortex on both sides to a certain extent, weakens the strength of the vortex, and provides space for the free growth of the dorsal separation bubble. The flow in the back separation zone has a strong interaction with the weakened flow vortex, which further reduces the strength of the vortex-containing structure, increasing the pressure at the tail of the model, and reducing the resistance of the model (see Fig. 6).



（a）Tail spoiler mounting location



（b）Velocity contour without spoiler

（c）Velocity contour with spoiler

Fig. 5 The flap controls the flow separation over the rear slant [3].

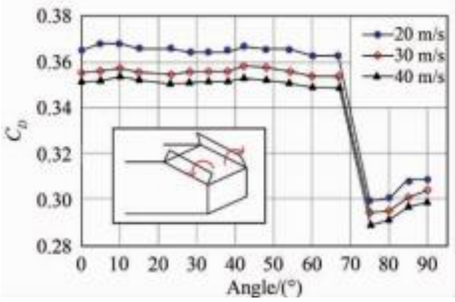


Fig. 6 Evolution of the drag of the bluff body as a function of the angle of the flap relative to the slant surface [3].

Active control: Using the Ahmed model, the wake structure is obtained by using a jet to the tail and actively controlling the tail dissociation, where φ=25°. After the active jet control is adopted at the tail (blue is the jet port), the scale and intensity of the flow vortex on both sides of the model are suppressed, which expands the range of the back-separation zone and promotes the pressure recovery of the inclined surface and bottom surface of the tail. The experimental results show that a drag reduction efficiency of 29% can be obtained by selecting the appropriate jet position and jet intensity (see Fig. 7).

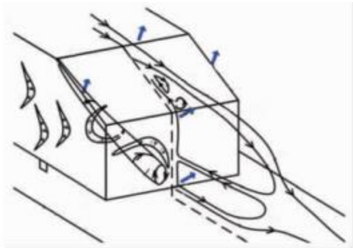


Fig. 7 Conceptual model of the flow structure under the combined actuation [3].

Summary: One advantage of the Ahmed method is that it can be used in an actual car. Another advantage is that it can reduce the cost of experiments and obtain accurate experimental data [3].

First, the results of the flow field study of physical models, such as Ahmed, can be deduced for real vehicles. Although the flow around a real car is more complex, its main flow field structure is similar to that shown in the model, and like the model, the different back inclination angles of the real car have an important impact on the vortex system of the wake and the pressure distribution of the tail of the model [7]. Second, the cost of model experiments is low, accurate experimental data can be obtained, and the rich research data of existing automotive aerodynamic physical models can be directly referenced. Through the flow field energy analysis power spectrum method, DMD, POD, and other methods, the structural composition and evolution characteristics of different energy levels in the flow field can be understood.

However, this method still has shortcomings. First, the drag coefficient of the Ahmed model is interrupted when the back-inclination angle is 30°, and the current research only has the results of the change in the drag coefficient and the qualitative flow field, and there is no specific mechanism explanation. Even in experimental studies, the drag coefficient of the back inclination angle is not clear when it is perturbed at approximately 30. Second, in the current study of the Ahmed model, the motion characteristics of sewage dripping were obtained mainly based on kinematic analysis, and the motion after contact with the car body and the influence on the flow of the boundary layer were not sufficiently clear. Third, from the perspective of research related to automobile acoustics, the model experiment is not like the real car experiment [3]. However, if an approximate similarity relationship can be established, the results of the model experiment can be used to approximate the prediction of the actual vehicle results, which will save resources [6].

2.4. Method 4

Method 4 involves the Computational Fluid Dynamics method in vehicle design.

1) Principle: Ava Martoma's research on CFD [4] relies on a virtual 2D or 3D simulation model-based approach that employs complex mathematical models and high-performance computing to numerically solve the Navier–Stokes equations governing fluid flow around a vehicle. By partitioning the vehicle surface into discrete elements, CFD simulations offer detailed insights into the distribution of the drag forces, pressure gradients, and flow patterns. The adaptability of this method to various conditions, geometries, and environmental factors makes it a versatile and efficient choice for modern vehicle aerodynamic analysis [9-10].

Fluid dynamics studies how fluids (such as air) move and interact with forces in various situations. For an incompressible fluid, the governing equations are the Navier–Stokes equation and the continuity equation. The Navier–Stokes equations dictate the flow of motion according to Newton's Second Law. They are as follows:

$$\rho\frac{\mathrm{D}u}{Dt}=\rho(\frac{\partial u}{\partial t}+(u\cdot\Delta)u)=-\nabla p+\nabla\cdot\mu[\nabla u+(\nabla u)^{\mathrm{T}}-\frac{2}{3}(\nabla\cdot u)I]+\rho g \tag{1}$$

$$\frac{\partial\rho}{\partial t}+\nabla\cdot(\rho u)=0 \tag{2}$$

where ρ is the fluid density and u are the fluid velocity vector. Eq. (1) the Navier-Stokes equation (for moment evolution) and Eq. (2) is the continuity equation (for mass conservation). These equations form the basis of the computational fluid dynamics performed using ANSYS Fluent software in this study.

2) Summary: For this study, the authors used ANSYS Fluent to analyze the drag force on unique car models by simulating intricate factors such as air viscosity, pressure distribution, and turbulence.

In this study, the authors considered the 2D side view of each car. The models were based on photographs taken from the side view of each car. By using ANSYS Space Claim, the curves of the cars were placed in the center lengthwise of the box. To simplify the fluid analysis, we removed the car wheels and set the car as if the wheels did not exist [4].

Using ANSYS Fluent software at varying velocities, the authors simulated 2D models of each car and compared the average drag forces and coefficients of the car models at different velocities. Then, they can see which car is the most aerodynamic.

3) Applications: Ava Martoma's (Ava Martoma et al., 2023) study [4] used the CFD method to implement an aerodynamic package for a sports vehicle. For the design of two active aerodynamic devices, the front splitter and rear wing are highlighted. Their methodology can be applied to the design of aerodynamic packages for sports vehicles [11].

In Davide De Cupis's (Davide De Cupis et al., 2023) study [5], the CFD method was used to compare the impact of car design on drag using 2D models of Toyota Corolla, Hyundai Elantra, and Kia Soul. In their simulations, the results show that the similarly shaped, sleek, and smooth Hyundai Elantra and Toyota Corolla have similar drag coefficients and drag force. For the considered velocity range, there are differences in the drag coefficient and drag force between these three cars. The results demonstrate that cars that may look similar, such as Toyota Corolla and Hyundai Elantra, are designed with fuel efficiency and aerodynamic, rather than aesthetic, considerations in mind.

4) Summary: Incorporating CFD into drag coefficient calculations reduces the need for expensive physical prototypes and accelerates the design optimization process. Engineers can explore innovative design modifications virtually and pursue ideas based on simulation feedback.

Regarding limitations, computational fluid dynamics relies on certain approximations that cause small margins of error between the predicted and actual results. CFD is ideal for comparative studies such as this one, in which we examine the relative differences between two designs because the error margins cancel out.

3. Discussion

In vehicle aerodynamics, the development and implementation of diverse body designs and auxiliary devices have significantly enhanced the overall aerodynamic performance of vehicles. This not only boosts vehicle speed but also reduces fuel consumption and emissions. Numerous studies have utilized wind tunnel testing and computational modeling to investigate the impact of aerodynamic devices, modifications, and body streamlining on drag reduction, vehicle handling, and fuel efficiency. It has been found that pneumatic control systems can reduce resistance by 20% to 30% [1]. For the GT model, which is both a highly optimized racing car and standalone design, the maximum drag reduction achieved was 16.53% when the tail and diffuser were applied [2]. However, when ride comfort is considered, these results are significantly influenced by the complex interplay between various components. In fact, modern family car designs integrate, and balance drag reduction with ride comfort, ensuring that there is no excessive bias towards one aspect, thereby avoiding potential design flaws. In summary, appropriate optimization leads to varying degrees of performance enhancement, benefiting not only racing and environmental protection, but also energy efficiency and ride experience. Moreover, the theoretical framework integrating pneumatics and comfort in vehicle design needs to mature rapidly to meet the increasing demand and standards, as well as to adapt to the rapid development of AI-driven intelligent driving.

The physical model of automobile aerodynamics is the basic method to study the changes in aerodynamics that affect the operation of a car, and the active and passive airflow changes can be obtained on the basic model, which can obtain different data and provide help for the improvement of a real vehicle [8].

The Computational Fluid Dynamics (CFD) method using ANSYS Fluent software is a fast and simple methodology that can be applied to the design of vehicles. This method allows the reduction of the lead time for the evaluation of the feasibility of a proposed aerodynamic solution and the reduction of the computational power required to perform the complete analysis of the vehicle [10].

At present, aerodynamic research has achieved fruitful results and accumulated a lot of important data. However, there is still a lack of clear mainline, and it is necessary to condense existing research to form a systematic theory. In general, future automotive aerodynamics research must develop a mature knowledge system as soon as possible in at least three aspects.

- To develop a car styling theory based on aerodynamic guidance.
- In the development process, experiments and calculations are reduced, resulting in a significant waste of resources.

- Also need to face the development of future automobiles and use models to reserve aerodynamic data. Accumulating learning and training data for the future realization of unmanned driving using artificial intelligence.

4. Conclusion

This article is a review article on aerodynamics research, with four articles selected. The first two articles discussed the role of active and passive flow control methods, both of which have a positive effect on the driving speed and energy conservation of automobiles. The third article describes the process of using the Ahmeh model to obtain flow control data, whereas the fourth article starts with the model used to obtain the data and explains how to obtain flow control data through CFD models. The fourth article has a progressive relationship. The first two articles start with different flow control technologies to understand their effects on reducing resistance and energy consumption, whereas the latter two articles demonstrate the process of obtaining flow control data. The research content of the four articles has strong relevance, focusing on the theme of aerodynamics research and covering multiple methods for controlling aerodynamics with scalability. Of course, there are still many shortcomings to this review. First, this article is still in the introductory stage of aerodynamics research, and the literature used is also relatively basic in this field. Second, this review has not yet included more directions in aerodynamics research, such as the application of artificial intelligence. Therefore, there should be more specific fields that require in-depth research in the future, and this study should also be used as the basis for subsequent research. In the future, more in-depth and specific fields should be selected for further research.

Authors Contribution

All the authors contributed equally and their names were listed in alphabetical order.

References

[1] A. N. M. Mominul Islam Mukut and Mohammad Zoynal, “Review on Aerodynamic Drag Reduction of Vehicles Abedin”, https://doi.org/10.26776/ijemm.04.01.2019.01

[2] Devang S. Nath, Prashant Chandra Pujari, Amit Jain and Vikas Rastogi, “Drag reduction by application of aerodynamic devices in a race car”, https://doi.org/10.1186/s42774-020-00054-7

[3] J. S. LIU, S. J. XU, Q. Y. WANG, et al. “Review of automotive aerodynamics research based on physical models” [J]. Journal of Experiments in Fluid Mechanics, 2020, 34(1): 38-48.

[4] Ava Martoma and Zbigniew Kabala “Drag on Passenger Cars: Computational Fluid Dynamics Analysis of 2D Body Design Using ANSYS Fluent”, (2023): https://sciprofiles.com/pfofile/698728.

[5] Davide De Cupis, Henrique de carvalho Pinheiro, Alessandro Ferraris, Andrea Airale, “Active Aerodynamics Design Methodology for Vehicle Dynamics Enhancement”, In book: Advances in ltalian Mechanism Science (2023) (pp.777-785) DOl: 10.1007/ 978-3-030-55807-9 86

[6] V. DOLCI and R. ARINA, “Proper orthogonal decomposition as surrogate model for aerodynamic optimization” [J]. International Journal of Aerospace Engineering, 2016, 2016:1-16. https://doi.org/10.1155/2016/8092824Citations: 28

[7] C. ZHANGY, J. T. ZHANG, K. G .WU, et al. “Aerodynamic characteristics of MIRA fastback model in experiment and CFD” [J]. International Journal of Automotive Technology, 2019,20(4):723-737

[8] B. F. ZHANG and Y. ZHOU, et al. “Unsteady flow structures around a high-drag Ahmed body” [J]. Journal of Fluid Mechanics, 2015, 777(1): 291-326.

[9] M. Carello, A. Bertipaglia, A. Messana, A. Airale, et al. “Modeling and optimization of the consumption of a three-wheeled vehicle” SAE Technical Paper 2019-01-0164 (2019). https:// doi.org/10.4271/2019-01-0164

[10] A. Ferraris, F. Micca, A. Messana, A.G. Airale, M. Carello. “Feasibility study of an innovative urban electric-hybrid microcar.” [J] Automot. Technol. 20(2), 237–246 (2019). https://doi.org/10.1007/s12239-019-0023-x

[11] S. Dey, R. Saha, “CFD study on aerodynamic effects of NACA 2412 airfoil as rear wing on a sports car.” In: ICMIEE, 18–25 December 2018