Fluid Dynamics in Automotive Design: Developments, Applications and Future Prospects

Yu Liu
School of Engineering University of Edinburgh Edinburgh, United Kingdom
y.liu-521@sms.ed.ac.uk

Abstract. This paper explores the significant role of fluid dynamics in automotive design, with a particular focus on its evolution and practical applications over the years. Using the Tesla Model S as a prime example, the paper highlights how advancements in fluid dynamic optimization have greatly contributed to improving energy efficiency, which is vital in electric vehicle design. Over time, automotive engineers have refined aerodynamic designs to reduce drag, enhance stability, and improve overall performance. Looking to the future, the paper discusses the emerging synergy between Artificial Intelligence (AI) and Computational Fluid Dynamics (CFD). AI is poised to revolutionize CFD simulations by enabling adaptive mesh refinement, optimizing parameters, and providing more accurate turbulence models. This integration is expected to accelerate the design process by facilitating quicker iterations and enhancing the precision of simulations. Additionally, AI can help address environmental uncertainties such as wind patterns and temperature variations, thereby boosting the robustness and reliability of automotive designs. In conclusion, the combination of AI and CFD represents a transformative step forward in automotive engineering, promising more efficient and innovative vehicle designs.

Keywords: fluid dynamics, automotive design, CFD simulation.

1. Introduction

After entering the 21st century, the automobile industry has undergone significant changes, and new energy vehicles on the road have gradually replaced the original diesel vehicles. This is the reason why the design focus of the automotive industry has gradually shifted from the traditional mechanical performance improvement and durability guarantee to the direction of more attention on energy conservation, emission reduction, and environmental protection. In today's increasingly popular new energy vehicles and intelligent driving, the car's designation must balance performance optimization and energy efficiency. In this context, engineers have been paying more and more attention to applying fluid dynamics in the automotive field. As a subject to study the law of fluid movement and its interaction with objects, fluid dynamics provides an important theoretical basis and technical means for optimizing vehicle design and improving vehicle performance. The use of fluid dynamics can not only improve vehicle fuel economy, reduce carbon emissions, and improve driving performance but also bring a more comfortable driving experience to the owner.

2. Historical developments

From the perspective of historical development, since the birth of the automobile, its design concept has undergone tremendous changes with the development of The Times. In the late 19th and early 20th centuries, automotive design focused more on mechanical structures' reliability and power performance. At that time, the car was a transportation tool, and the basic transportation needs of people were taken as the original intention of the design. More attention was paid to how to improve the efficiency of the engine and the robustness of the body. Moreover, car design can only be improved according to people's feedback because of the need for more early science and technology.

By the middle of the 20th century, with the continuous pursuit of car speed, major automobile manufacturers will design faster and better performance models as the primary goal. Air resistance is gradually found to be one of the key factors affecting vehicle fuel economy and driving performance. As the speed of the vehicle increases, the influence of air resistance on energy consumption and power performance becomes more and more significant. Designers also thought of reducing air resistance by simulating the flow field around the car, and wind tunnel experiments, a significant historical development, became the mainstream optimization tool at the time, testing the airflow distribution and pressure changes around the vehicle through physical models. However, this experimental method is costly and inefficient and can only play a role in later design validation.

In recent years, the emergence of computational fluid dynamics (CFD) has improved the situation. CFD technology accurately simulates fluid flow with the help of powerful computers. The advantage of CFD compared to wind tunnel experiments is that multiple scenarios can be tested quickly at the beginning of the design process, thus reducing development costs and shortening the design cycle. CFD technology has gradually become an indispensable tool in modern automotive design.

In the future, the further combination of AI technology and CFD technology can reduce the shortcomings of traditional CFD. Machine learning can calculate the effect on air resistance by analyzing data simultaneously, such as changing multiple parameters simultaneously, thereby speeding up the optimization process, improving design efficiency, and making optimization more efficient and precise. This will also be an important direction for future car design.

3. CFD Technology

Computational Fluid Dynamics (CFD) is a technology that combines with computational method and fluid mechanics. It is based on Navier-Stokes equations, which calculate and predict the flow of liquids and gases using computers using the governing equations of mass, momentum, and energy conservation [1]. Compared with the traditional wind tunnel method, it requires many solid model manufacturing and testing, which is high cost and low efficiency. CFD technology allows engineers to quickly test on simulated data in computers, which can significantly shorten development cycles and reduce design costs.

The general step of CFD technology is first to establish a three-dimensional model of the car in CAD software, including the body, chassis, wheels, and various parts, and then use a grid of different thicknesses to divide various areas of the car. The fine mesh is used for complex areas such as the front and rear of the car, and the coarse mesh is used for the middle of the body and the area where the airflow is relatively stable. Engineers then simulate real driving situations by setting the speed of the vehicle in different environments. After that, CFD software such as Ansys and Fluent can calculate the flow characteristics of air around the body through iterative solution. Finally, the flow separation point, high pressure area and turbulent area can be found and optimized through the visualization results such as pressure distribution diagram, velocity field diagram and eddy current diagram.

In addition, CFD technology is widely used in designing traditional fuel vehicles and plays a crucial role in designing new vehicles, such as electric ones. Compared with fuel vehicles, electric vehicles rely more significantly on aerodynamics in the design because electric bicycles usually have an energy recovery mechanism, and air resistance is the primary source of energy loss when the car is driving at high speed, a large amount of energy loss will also significantly affect the driving range of electric vehicles. Therefore, reducing air resistance through hydrodynamic optimization directly affects electric vehicles' endurance and energy efficiency performance.

4. Applications in tesla model S

As one of the most representative brands in the field of new energy vehicles, Tesla fully embodies the profound combination of fluid mechanics and CFD technology in its model design. Among them, the Tesla Model S is known as the model of new energy vehicles, the object of learning for significant new energy vehicle brands, and one of the representative works of the successful application of CFD technology in automobile design. As shown in Fig. 1 and 2, the Model S design team used the theory of fluid mechanics throughout the vehicle development process. It analyzed every detail of the body with the application of CFD, from the overall shape to the local optimization of small components, reflecting the precise fluid dynamic design.

Fig. 1 CFD simulation of Tesla Model S [2].

First, the front design of the Tesla Model S abandons the complex decorative lines standard in traditional fuel vehicles and instead adopts an integrated smooth arc structure. This design has no sharp Angle or abrupt design, significantly reducing the air separation phenomenon in the front area. Through CFD simulations, the design team found that the circular front shape can direct airflow around the body surface, avoiding turbulence and eddy flows caused by irregular shapes. In addition, the circular arc structure also effectively reduces the source of air resistance, which can optimize the airflow distribution and reduce the front side pressure so that the flow field in the front area is more uniform. This simplified design helps reduce energy consumption and gives the Model S a more modern, simple, and aesthetically pleasing look, perfectly balancing function and beauty.

The overall body shape of the Model S presents a continuous and smooth curve because the optimal curvature of the structure is calculated using CFD technology, which makes the design smooth and reduces the excess resistance and airflow caused by airflow on the body surface.

In addition, the Tesla Model S has a low body height of about 1445mm [3] to reduce air resistance, which complements the streamlined structure. CFD simulation analysis also indicates that lower body height can effectively reduce airflow disturbance and separation, especially in the transition area between the roof and the side of the body.

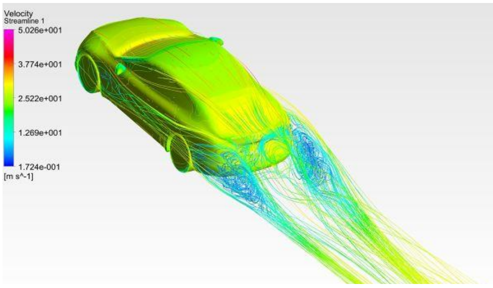

Fig. 2 CFD simulation of Tesla Model S [2].

The tail of Model S adopts a slightly upturned design, like a spoiler, and can effectively reduce the pressure resistance caused by the tail vortex. This design makes the air resistance coefficient of Model S is about 0.24 [4]. It is much lower than the 0.3~0.35 of most traditional cars [5]. This is because when the car is driving at high speed, a low-pressure area will be formed at the rear part which results in an air pressure difference. By changing the path of the air flow, the spoiler helps the air flow smoothly through the rear of the car, reduces the area of the low-pressure region, and reduces the pressure resistance. In addition, when the vehicle is running, the airflow along the roof reaches the tail, which is easy to separate due to the sudden change of the shape of the tail, forming turbulence. The designers use CFD simulation of spoilers to adjust the Angle of the tail air flow and guide the air closer to the body surface, thereby reducing turbulence resistance. Additionally, the spoiler structure can also increase the rear's downforce by changing the airflow direction, making the rear wheel grip stronger, and improving the stability of high-speed driving.

The connection between the window and the body of the Model S is designed with a high-precision seal strip, which makes the window and door surface smoother. The side lines of the body are simple and smooth, reducing obtrusive design elements. This can also reduce the separation of the side airflow to a certain extent. In addition, the hidden door handle design can also reduce local air resistance because the hidden door handle flushes with the body surface design, which can eliminate this excess drag sources. Reducing any slight air resistance for a range-focused electric vehicle like the Model S will help improve the overall range. By simulating the local air flow of the traditional car with CFD, it is easy to find that the protruding door handle design in the traditional car will significantly lead to the separation of the airflow, which is why the Model S uses the hidden door handle. After the door handle is retracted, the airflow on the side of the car can flow smoothly along the door's surface, avoiding air separation. The hidden door handle reduces turbulence and wind noise; the reduction of turbulence directly reduces the wind noise generated while driving, makes the car quieter, and improves ride comfort.

What is more, the design teams also used CFD to analyses the airflow characteristics around the area where the tire contacts the ground, and the analysis showed that the interaction between the rotating wheel and the surrounding airflow creates complex turbulence, especially where the tire contacts the surface of the road. This turbulence flow not only increases aerodynamic drag but will also affect the stability of the vehicle. The Model S design team's solution to this problem was to adopt a closed-rim design for the wheels, which would make the rim surface smoother, reducing the complex opening structure of traditional wheels. The study found that the wheel's influence on the vehicle outflow field could reduce the drag coefficient by nearly 0.015 (Peng, 2018).

The Model S chassis is also relatively flat. In the CFD simulation process, a flat chassis can significantly reduce the bottom of the car's turbulence and tail low-pressure area and improve the performance.

5. Conclusion

CFD technology provides significant support for innovation in the automotive industry realm. With improved computing power and accelerating technology integration, CFD technology will show more significant potential in auto driving, intelligence, and sustainable development. However, CFD technology will also face considerable challenges in the future. Improving CFD technology is also worth studying to pursue a lower coefficient. The accuracy of CFD technology still needs to be improved, even if engineers enter specific environmental parameters, but real-life environmental factors can change over time.

Extreme weather, such as heavy rain and snow, is also difficult to simulate. These unpredictable conditions make it challenging to model their effects accurately using traditional computational methods. As shown in Fig. 3, during heavy rain, the interaction between the jet stream and the raindrop can create a complex airflow and change the pressure distribution on the vehicle surface.

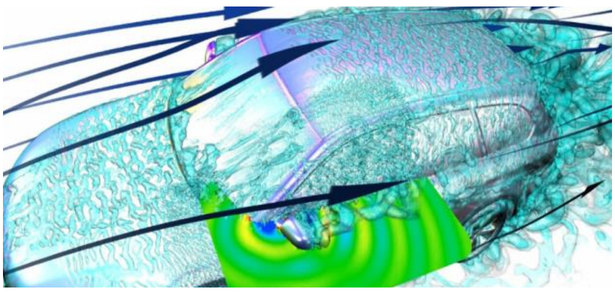

Fig. 3 Rain flow in vehicles [6].

However, in the future, the combination of artificial intelligence technology and CFD can further compensate for the shortcomings of traditional CFD methods. For example, generative adversarial networks (GANs) can synthesize accurate flow field data to train CFD models [7]. GANs can generate high-fidelity synthetic flow field data to provide more decadent training samples for CFD models. This will reduce reliance on physical experiments and costly wind tunnel testing, especially in extreme environmental conditions where data is scarce or difficult to obtain. For example, GANs can simulate the behavior of fluids in complex climatic conditions, such as heavy rains and strong winds, to optimize the performance of vehicle designs for extreme weather. At the same time, the simulation parameters can be adjusted in real-time so that the CFD simulation can adapt to the dynamically changing environmental conditions. Significantly improves computational efficiency under complex conditions.

In addition, using Digital Twin technology, it is also possible to create virtual models of physical objects or systems and monitor and simulate their behavior in real-time. In the field of CFD, digital twins will enable more accurate virtual testing and dynamic optimization. In the design of electric vehicles, digital twins can simulate the performance of vehicles in different climatic conditions and predict energy consumption and endurance.

In addition, with the rapid popularization of autonomous driving technology, multiple sensors on the body of autonomous vehicles, such as LiDAR, cameras, and antennas, will destroy the vehicles' original streamlined design, resulting in airflow interference and increased air resistance [8].

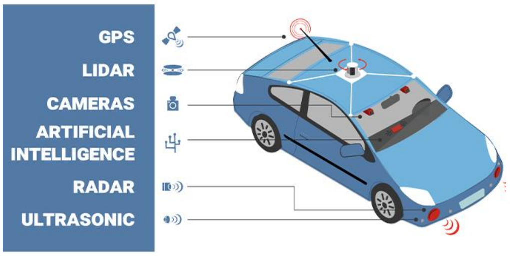

Fig. 4 Self-driving cars [10].

Reinforcement Learning (RL) technology will be an essential methodology to analyze the vehicle surface's airflow distribution through CFD simulation, optimizing the sensor's layout and shape design, thereby minimizing aerodynamic losses [9]. For example, the RL technology can analyze airflow distributions in real time and dynamically adjust the angle or position of the sensors to reduce turbulent flow and air resistance. RL technology also optimizes aerodynamic components, such as tail controls, for different driving conditions, like driving at speeds or facing extreme weather, further improving the vehicle's performance stability. This innovative method not only helps solve the problem of sensors but also provides a more efficient solution for optimizing the performance of autonomous vehicles.

Finally, CFD simulation technology has high requirements for users, who must have specific professional knowledge and techniques, leading to fewer professional CFD engineers who can use it. Moreover, many countries have also been promoting and marketing new energy vehicles, such as electric vehicles. However, the current domestic CFD simulation technology has not been widely popularized, which has led to the lack of professional engineers in this field and has also restricted the development of automotive technology to a certain extent.

As shown in Fig. 4, as the automotive industry moves toward low-carbon and environmental protection, the role of fluid dynamics in electric vehicles is more important than that of traditional internal combustion engine vehicles. This is mainly because electric vehicles can recover the braking energy into the battery during deceleration. However, the energy loss caused by air resistance cannot be recovered, so too much air resistance will reduce the recoverable energy ratio of electric vehicles. This is why Tesla invested vast sums of money in developing the Model S, using different technical means to reduce its air resistance.

The success of the Tesla Model S demonstrates the great potential of integrating fluid mechanics in the automotive realm. The application of fluid mechanics optimizes the energy efficiency of new energy vehicles and provides key technical support for their intelligent and sustainable development.

Looking to the future, as technology continues to advance, fluid mechanics will play an even more important role in driving automotive design innovation and performance improvements. Through collaborative development across disciplines, fluid mechanics will become a core driver for more efficient, environmentally friendly, and comfortable vehicles. In the context of eco-friendly development, the application of fluid mechanics will continue to provide solid technical support for the intelligent and low-carbon automobile industry and lead the global automobile industry to a more sustainable and intelligent future.

References

[1] Chen, Y., Sun, R., and Wu, X. Estimating Bounds of Aerodynamic, Mass, and Auxiliary Load Impacts on Autonomous Vehicles: A Powertrain Simulation Approach. Sustainability, vol. 13, no. 22, p. 12405, Nov 2021.

[2] FetchCFD. Tesla Model S Car Aerodynamic Analysis. 2019.

[3] Automobile dimensions. Tesla Model S dimensions, boot space and similars. Automobiledimension.com, 2021.

[4] Chen, Y., Sun, R., and Wu, X. Estimating Bounds of Aerodynamic, Mass, and Auxiliary Load Impacts on Autonomous Vehicles: A Powertrain Simulation Approach. Sustainability, vol. 13, no. 22, p. 12405, Nov 2021.

[5] Gene. Digital Wind Tunnel Tech Behind Tesla Model S Aerodynamics. TESLARATI, Feb 18, 2015.

[6] Ansys. What is Computational Fluid Dynamics (CFD)? | Ansys. www.ansys.com, 2024.

[7] Hu, L., Wang, W., Xiang, Y., and Zhang, J. Flow Field Reconstructions with GANs Based on Radial Basis Functions. IEEE Transactions on Aerospace and Electronic Systems, vol. 58, no. 4, pp. 3460–3476, Feb 2022.

[8] Kurz, M., Offenhäuser, Philipp, Viola, D., Shcherbakov, O., Resch, M., and Beck, A. Deep reinforcement learning for computational fluid dynamics on HPC systems. Journal of Computational Science, vol. 65, pp. 101884–101884, Nov 2022.

[9] Enteknograte. Aerodynamics Simulation: Coupling CFD with MBD, FEA and 1D - System Simulation for NVH Analysis, Drone, Wind Turbine, Race Car, and Cabin Thermal Comfort. - Enteknograte. Enteknograte, Oct 20, 2019.

[10] AAA. Self - driving cars, explained. www.ace.aaa.com, Sep 09, 2016. https://www.ace.aaa.com/automotive/advocacy/self - driving cars - autonomous - vehicles - explained.html?intcmp=homepage: promounit2x2: AVs