Methods on Adjusting Vehicle's Shape to Control Air Resistance

Jiarong Wu[1, *, †] Yiding Wen[2, †]

[1] Leicester International Institute, Dalian University of Technology, Panjin Campus of Dalian University of Technology, Panjin124221, China

[2] Pittsburgh Institute, Sichuan University, Sichuan6102205, China

* Corresponding Author Email: jw827@student.le.ac.uk

†These authors contributed equally to this work and should be considered co-first authors.

Abstract. Wind resistance is usually expressed by the following formula. Fx＝ CdA ρ v^2 /2. Where Cd is a constant. And A is the contact area between air and car. ρ is air density. v is the velocity. Because of the complex structure, mostly a very specific and accurate model cannot be obtained directly. Instead, engineers will use wind tunnel test to get a practical drag coefficient of a car. In the actual analysis, the influence of wind resistance cannot be ignored in the car dynamic system. Wind resistance is the main factor to counteract traction in the process of high-speed driving. Knowledge and literature about aerodynamics of vehicle is abundant in the world. However, this knowledge is not directly related to car design. Engineers empirically design vehicles in the perspective of aerodynamics based on what they have learnt. This paper will put forward some suggestions on vehicle structure according to the practical application of various aerodynamics. It will simultaneously emphasize how the knowledge in the literature is related to the car design in reality. Engineers can design vehicles combining with aerodynamic theory. In recent years, wedge-shaped vehicles have become the mainstream of design, which is the result of the pursuit of low wind resistance and higher stability. The streamlined body shape conforms to both aerodynamics and mainstream aesthetics. Improving speed and safety while saving energy is the most intuitive manifestation of aerodynamic application in vehicle field.

Keywords: Aerodynamics, ANSYS FLUENT, CATIA, CFD.

1. Introduction

In the industry III era, an indispensable part of designing vehicles is from the perspective of aerodynamics. Aerodynamics in vehicles is about studying the relationship and motion performance of the interaction force between the vehicles and the air surrounding the cars from the perspective of relative motion. In the various condition of vehicle traveling, the shape of vehicles can greatly affect this interaction force. For example, vehicles can drive at high speeds up to 120km/s. The vehicles drive at different angles to the wind direction. However, the car design is achieving the design requirement that the engineer controls the air resistance rather than reducing the air resistance continuously. Because engineers should design the vehicles from the perspective of aesthetics. Optimizing the shape continuously will affect the aesthetic structure of the vehicles. Consequently, reducing its economic benefits. Additionally, exhaust and ventilation of pipeline is an important portion in aerodynamics in vehicles because flowing at high speed in the pipeline, gases will have a great pressure impact on the pipeline. During the process of driving, the safety of the pipeline should be guaranteed, otherwise, it will lead to pipeline deformation, and affect the performance of the pipeline, consequently causing potential safety danger. Now there are many structural applications for aerodynamics. For example, Han et al. introduce the design logic of the racing car based on CFD including diffuser, rear wing, etc [1]. And Cui et al. talks about the Application of aerodynamics in ordinary car structure. In ref [2], the new applications of commercial CFD software are introduced to engineers. In cooperation with software companies, the adjoint method is added to the CFD code. When engineers use these commercial professional CFD software for vehicle design, the addition of methods will enable the software to provide engineers with very fine and detailed strategies. However,

in ref [2], the literature shows us a very powerful way to make CFD software more conducive to engineer design. Engineers themselves are very important to learn aerodynamic knowledge. At the same time, the real design process also needs sufficient aerodynamic design knowledge to support.

In 2014, it was concluded from ref [3] that progress in fuel was difficult to achieve in order to increase fuel efficiency. In order to increase the fuel efficiency of the whole vehicle, engineers currently take the optimization to reduce air resistance. It can be seen in ref [3] that for most vehicles, 50 % of the air performance of the vehicle depends on the air vortex generated at the tail of the vehicle during driving. The vortex will pressure the vehicle to reduce its motion performance and increase energy consumption. For engineers, it should be noted that the tail pressure control is related to the positive and negative lift of the vehicle [3]. The negative lift is to improve the stability of the vehicle. The positive lift is to reduce the driving resistance of the vehicle. Therefore, the design of the tail is designed to lift.

The aerodynamic design of tail pressure is very important in vehicle design [4], which is also mentioned in [3]. The existence of spoiler can make the vehicle obtain the maximum traction and manoeuvrability while reducing the driving resistance of the vehicle [4]. When the spoiler is used, the design of the inclination angle is the core of the spoiler. The design of the inclination angle can affect the design performance of the spoiler to meet the design requirements. If the angle of the spoiler is positive, the increase of the angle will increase the aerodynamic lift, and the relative will increase the aerodynamic drag. The setting of spoiler has great influence on aerodynamic performance of vehicle. Its influence on vehicles is both advantageous and disadvantageous. Suitable design of aerodynamic lift and aerodynamic drag will make the aerodynamic performance of the vehicle meet the design requirements. The use of vortex generators can make the vehicle perform better in aerodynamic performance [5]. Its function is the time node when the delayed flow of air separates from the vehicle during the driving process. The final effect is to change the air turbulence energy around the vehicle to control its aerodynamic performance. Similar to ref [4], the vortex generator can affect the aerodynamic performance of the vehicle by changing the air flow clearing. But unlike ref [4], the vortex generators used in ref [4] can be regulated differently according to the set mode. It can reduce or increase the turbulent energy around the vehicle. In reduced cases, the generator can perform well in the acceleration phase of the vehicle. With the increase, the generator can make the vehicle perform well in the deceleration phase.

2. Object and background

2.1. Introduction of Computational Fluid Dynamics

Computational Fluid Dynamics is the simulation method used by most aerodynamics at present. The method Computational Fluid Dynamics aims to study vehicle aerodynamics with the help of the computer, namely, to imitate the working conditions of wind tunnel tests. The air blowing is simulated on the computer, and the aerodynamic problems of the vehicle are calculated and simulated by using the method of numerical analysis, which provides a scientific basis for the study of more efficient vehicle shape and structure. More importantly, with the continuous development of code and algorithm, the function of Computational Fluid Dynamics has developed deeply. From the calculation of aerodynamic performance of vehicles in a specific field flow to now, it can automatically optimize the vehicle model and give optimization suggestions [2]. Simultaneously, it calculates and predicts the influence of changing any parameters on the aerodynamic performance of vehicles. With aerodynamic simulation software Computational Fluid Dynamics, engineers can quickly complete a vehicle shape design that meets the requirements when optimizing the vehicle shape to reduce wind resistance.

2.2. Introduction of Fluent

In this article, authors will first use CATIA to model (Figure 1) and conduct CFD analysis by importing a simple vehicle model into ANSYS fluent (Figure 2). FLUENT is a computer program of

ANSYS used to simulate fluid flow with complex shapes. ANSYS software mainly includes three parts: pre-processing module, analysis, and calculation module, and post-processing module.

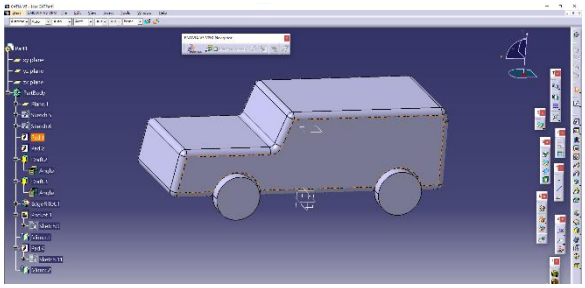


Figure 1. Car model of CATIA

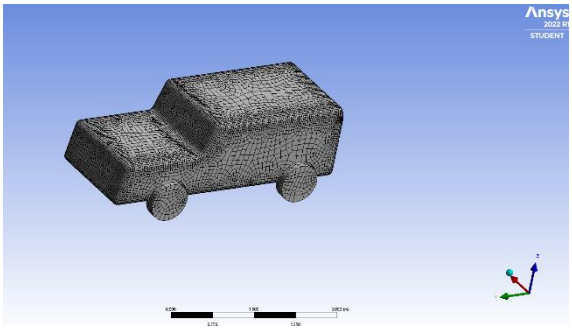


Figure 2. Car Grid by ANSYS FLUENT

The pre-processing module provides a powerful solid modelling and meshing tool, and users can easily construct finite element models.

The analysis and calculation module includes structural analysis (linear analysis, nonlinear analysis, and highly nonlinear analysis), hydrodynamic analysis, electromagnetic field analysis, sound field analysis, piezoelectric analysis, and coupling analysis of multiple physical fields. It can simulate the interaction of multiple physical media and has the ability of sensitivity analysis and optimization analysis.

The post-processing module can display the calculation results in the form of graphics such as colour contour display, gradient display, vector display, particle flow trace display, three-dimensional slice display, transparent and translucent display (you can see the interior of the structure), and can also display or output the calculation results in the form of chart and curve. Through modelling and meshing the geometry, the fluid is simulated and analysed. After the grid is established, the boundary conditions are determined for the whole calculation area, and the simulation results of the fluid are output after the boundary conditions are determined.

Xue and his team also simulated the model car through fluent, and found that the surface pressure of the car head was the largest. This was because the airflow velocity met the car head, and the airflow was blocked when it met the car head, which greatly reduced the airflow velocity and thus formed a positive pressure area in the car head [6]. The authors obtained a similar conclusion through the simulation of turbulence that the airflow accumulates at the vehicle head (Figure3) (Figure 4). And when touching the front windshield, the air velocity will decrease rapidly (Figure 5).

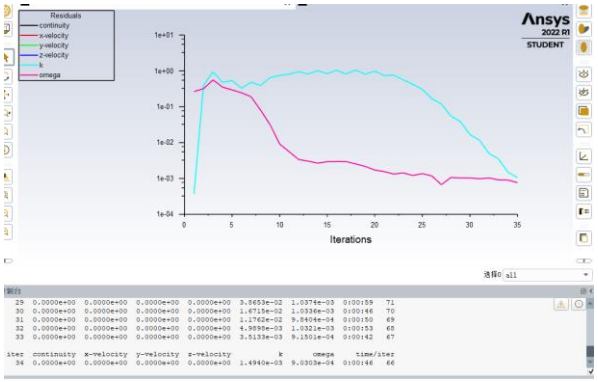


Figure 3. Data Simulation ANSYS FLUENT

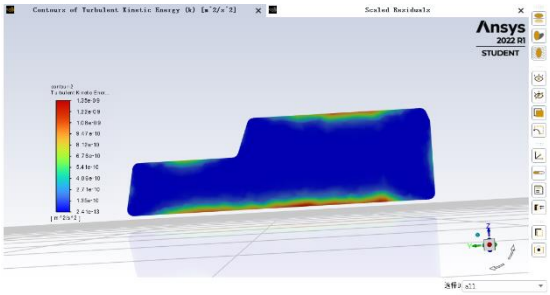


Figure 4. Turbulence Analysed by ANSYS FLUENT

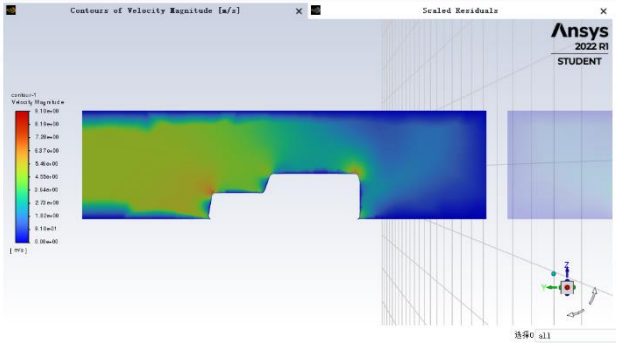


Figure 5. Velocity Analysed by ANSYS FLUENT

Compared with ordinary wind tunnel tests, CFD is not limited by wall interference and blocking effect, so more information can be obtained by solving hydrodynamic equations than ordinary wind tunnel tests [7].

For most of the racing cars, there are basically two ways to Improve performance by changing structure. First one is to decrease the drag, and the second one is to increase the downforce. With the increase of sideslip angle, the cornering potential of tire also increases. Increasing the cornering potential is to add more downforce to the vehicle. Which means it can get more friction between the tire and the ground and to provide more power.

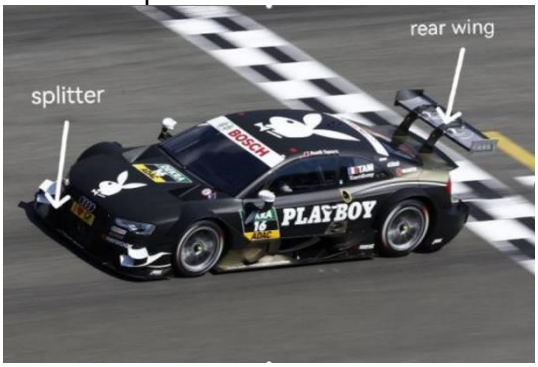


Figure 6. Structure of Racing Car

The front end of most cars has a 'splitter' that is relatively parallel to the ground. It tries to keep high-pressure air on the top of the car rather than flowing under the car which will increase the downforce. And its width makes it block in front of the front wheel, and the splitter surface can make the air flow bypass the front wheel as much as possible to reduce the resistance. Besides, racing cars have the structure called 'rear wing' (Figure 6). The rear wing has only one purpose, generating downforce. At the same time, reduce the aerodynamic resistance as much as possible. For the rear wing, the ways to obtain higher aerodynamic pressure are: increasing the surface area of the lift wing; Increase the lift wing radian. And there's also a structure called 'diffuser'. This is a device assembled at the bottom of the car, so that the air pressure at the upper part of the car is greater than the air pressure at the bottom, so as to form a pressure difference, so that the car generates a pressure towards the ground, so as to enhance the downward pressure of the car and make the car tire have good grip [1].

Different front-end shapes lead to complex front-end drag coefficient. Theoretically, a fully streamlined front end is the best, but it is impossible to adopt it in practical design. If we can round the edges and corners as much as possible, make the shape close to the streamline and reduce the projection area of the front of the car, we can get better aerodynamic effect.

At the same time, the shape of the roof has a great influence on the aerodynamic characteristics of the vehicle. The top cover is designed as an upper drum, which is conducive to the smooth flow of air in the vehicle. The aerodynamic resistance of a car is directly proportional to the projected area of the front. The airflow smoothly passes through the roof and reduces the air resistance coefficient Cd value. The side effect is that the front projection area increases, thus increasing the resistance. Therefore, these two factors should be comprehensively considered in the structural shape design.

The bottom of the car is usually uneven, which makes the air flow at the bottom complex and forms various complex vortices. When the ground clearance is small, the air flow between the bottom of the car and the ground may be blocked, so as to turn the front air flow to the upper surface of the car body, increase the air flow speed on the upper surface of the car, reduce the downforce, and increase the resistance and lift of the car. The underside of the car is made into a smooth shape to reduce friction and resistance on the body surface. When the ground clearance increases, the air flow between the lower body and the ground can flow smoothly, thereby reducing resistance and lift [8].

Moreover, Kulak and his team put forward an interesting concept - studying the aerodynamics of wheels. They found that up to 25 % of vehicle aerodynamic drag comes from wheels. The aerodynamic drag of wheels with grooves is obviously lower. They believe that sudden pressure drop in the groove promotes air flow on the tire surface and reduces the size of the vortex [9].

3. METHOD

Chronologically integrating several existing literatures, this article will reasonably and concisely introduce the advantages and disadvantages of these literatures and their feasibility in other fields. Additionally, evolution process of vehicle design from the perspective of aerodynamics will be shown by learning those literatures. What is more, the effect on other fields from these processes is researchable.

3.1. Difference from Traditional Analysis Methods

Conventional aerodynamic car design is preformatting in wind tunnel simulation in steady condition. However, when the vehicles are travelling, its aerodynamic parameters will change because of overtaking or following. In 1997, the research on aerodynamic of vehicles during overtaking or following was proposed [10]. From the wind tunnel test data in the literature ref [10], it can be concluded that the air drags coefficient of the vehicle which is travelling in platoon is less than the condition that the car travel alone. The air resistance of each vehicle can be reduced if those vehicles travel close to each other. Therefore, when a car leaves the platoon, the drag coefficient will increase. The advantages of this literature are that it focused on the car performance in the perspective

of aerodynamic when the car is travelling in the real situation. In reality, the aerodynamic performance of the car cannot always be in the situation which is a single car in the wind tunnel test. More realistically, in most cases, the vehicles are travelling complexly with the other vehicles. Therefore, engineers should take this situation into consideration when designing cars from the perspective of aerodynamic.

3.2. Case Analysis

During the car travelling, the rear-view mirror will increase the air resistance of the whole vehicle. In the car design, engineers can design the appearance of the vehicles to reduce the its air drag from the perspective which is the rear-view mirror proposed in ref [2] will greatly affect the aerodynamic performance of the vehicles.

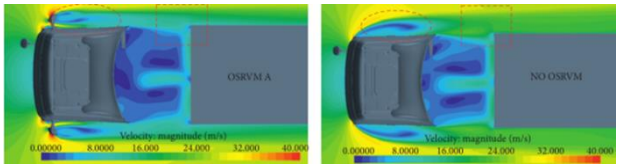


Figure 7. Different Air Resistances of Commercial Vehicles with or without Rear-view Mirrors

Referring to ref [11], what should engineer notice is that the commercial trailer has greater air drag without a rear-view mirror. That is a point that engineers need to pay attention to. Under different models, the installation location and installation of rear-view mirrors will have different effects on air resistance. What is more, commercial computational fluid dynamic software for simulation can be used to spot its influence, which can help engineers correct its car design. Ref [11] is a real case of car design. The purpose of this paper is to reveal the influence of the installation of rear-view mirror on the air resistance of commercial vehicles. Through this literature, it is pointed out that the design of the rear-view mirror is equally important in car design, simultaneously, its influence on air resistance is large, which requires careful design.

4. Result

These literatures show that throughout the application of aerodynamics in vehicles, it is the continuous development of simulation software and the application of fluid mechanics to vehicles after abundant contributing breakthrough. In the era of industrial 3.0, engineers only need to learn the use of simulation software such as CFD and aerodynamics knowledge, can quickly and meet the requirements of the vehicle shape optimization through CFD software. The function of Computational Fluid Dynamics software is constantly innovating, and the research on aerodynamics is also constantly innovating. Engineers need to constantly learn the knowledge of both and integrate them into the engineering of optimizing vehicle shape. Therefore, this paper first introduces the innovation of CFD software, reflecting the high applicability of CFD software and the convenience of design. However, the use of CFD also requires engineers ' own aerodynamics knowledge. This paper also introduces the application of aerodynamic knowledge in vehicle shape design.

5. Conclusion

This article researches several literatures to find the connection between them which is progressive based on the their former. Simultaneously, the research on vehicle based on aerodynamical design has also helped other related or unrelated industries. For example, aerodynamic design is used in racing cars, aircraft and rockets. Even these researches have great help on designing pipelines for transporting gases. Several aerodynamics knowledge is also presented in this article, and they are also mentioned for vehicle design.

REFERENCES

[1] Xiaoqiang Han, Hongyu Wang, Design of an Aerodynamic Package for FSAE Racing Car and CFD Analysis, in: Experimental science and technology, 2016,14(1):3-7. DOI:10.3969/j.issn.1672-4550.2016.01.002.

[2] Othmer C. Adjoint methods for car aerodynamics. Journal of Mathematics in Industry 2014 06;4(1):1-23.

[3] Wang Y, Xin Y, Zh. Gu, Sh. Wang, Deng Y, Yang X. Numerical and Experimental Investigations on the Aerodynamic Characteristic of Three Typical Passenger Vehicles. Journal of Applied Fluid Mechanics 2014 01;7(4):659-671.

[4] See-Yuan C, Mansor S. Influence of rear-roof spoiler on the aerodynamic performance of hatchback vehicle. MATEC Web of Conferences 2017;90.

[5] Shankar G, Devaradjane G. Experimental and Computational Analysis on Aerodynamic Behavior of a Car Model with Vortex Generators at Different Yaw Angles. Journal of Applied Fluid Mechanics 2018 01;11(1):285-295.

[6] Jinlu Xue, Bingzhi zhang, Guoying Xu. Application of Fluent in Aerodynamic Features of Automobiles [J]. Journal of Armored Force Engineering College, 2009,23(3):33-37. DOI:10.3969/j.issn.1672-1497.2009.03.008.

[7] Yuehua Jiang, Zhengqi Gu, Application of CFD in automotive aerodynamics, in: Journal of Hunan University: Natural Science Edition, 1997,024(004):52-56

[8] An Cue, Quan Kang, Haipeng Zhang, Aerodynamic shape design and CFD analysis of car, Proceedings of the 2009 International Symposium on Industrial Design. Machinery Industry Press (China Machine Press), 2009, 41-45.

[9] Leśniewicz, P., Kulak, M., & Karczewski, M. (2014). Aerodynamic analysis of an isolated vehicle wheel. Journal of Physics: Conference Series, 530(1) doi:https://doi.org/10.1088/1742-6596/530/1/012064

[10] Chen AL, Savas O, Hedrick K. Transient Vehicle Aerodynamics In Four-car Platoons. IDEAS Working Paper Series from RePEc 1997.

[11] Hu B, Lu Z, Cui Q, Tang R, Feng Z, Bi D. Prediction and Aerodynamic Analysis of Interior Noise and Wind Drag Generated by the Outside Rear-View Mirror for Commercial Vehicles. Shock Vibrat 2020;2020.