Simulation Analysis for Effect of Rear Structure of Hatchback Car on Rear Field Characteristics

Xian Qu* and Jun Xie

College of intelligent manufacturing and transportation, Chongqing vocational institute of engineering, Chongqing, China

*Corresponding author email: 516685355@qq.com

Abstract. The tail shape of the hatchback car is crucial to the rear field. To improve the tail aerodynamic design of the hatchback car, the orthogonal optimization method is used to reveal the rule of different tail structures affecting the aerodynamic drag and lift force considering their interference with each other. First, the different tail structures, such as the top, the side, the bottom and the tail structures, are analysed to reveal the influence of the wake based on the MIRA model. Then, the orthogonal optimization method is applied to investigate the tail shape optimal design considering the mutual influence of each tail structure. The aerodynamic drag coefficient and the lift coefficient are calculated to evaluate the aerodynamic characteristics in the orthogonal analysis. The results show that the angles of the rear windshield and the diffuser has a great influence on the aerodynamic drag. While the diffuser angle have the greatest influence on the lift coefficient. The influence of the rear windshield angle and roof angle is also noteworthy. In a word, the diffuser angle, the rear windshield angle and the roof angle are the key structures for the aerodynamic optimization of the tail of hatchback cars.

1. Introduction

For the vehicles running at 150km/h, the aerodynamic drag is equivalent to 2 ~ 3 times the rolling resistance [1]. The fuel consumption caused by aerodynamic drag is in direct proportion to the cube of the velocity [2]. As vehicle aerodynamic characteristics are closely related to the rear field, which is significantly affected by the tail modeling [3-5]. It is of great significance to investigate the influence of tail structure on the rear field for the aerodynamic design of automobiles.

Research on the automotive rear flow field attracted wide attention, achieving fruitful results. Some efficient active control methods, such as active air jetting, are studied to weak the wake vortex, which reduces the automotive aerodynamic drag effectively [6-8]. Besides, the rear wing or spoilers are commonly used to increase negative lift and thus improve driving stability [9]. These methods are developed based on the analysis of the rear flow field determined by the structure of the body tail. Kounenis et al. [10] investigated the rear-end flow structure on a sedan car and pointed out that it is comparable for the detailed flow structures on similar but distinct vehicles. Liu et al. [11] studied the wake structure of the hatchback car, presenting the location and development process of the separation of the wake flow. Yang et al. [12] conducted numerical simulation under typical angles including the diffuser angle and the rear windshield inclination angle, showing that the influence of these two angles on aerodynamic drag ranked second and third, respectively. Wu et al. [13] studied the influence of the vehicle rear plan on wake flow field based on a simplified SUV model. These studies contribute to improving the aerodynamic characteristics of the vehicle. However, the study about the specific rear structure affecting the rear flow is little considered, particularly from the aspect of the whole rear structure.
This paper focuses on the influence of specific rear structures on rear flow from an overall perspective. The main contribution is to reveal the rule of different tail structures affecting the aerodynamic characteristics by the orthogonal optimization method considering their interference with each other. The influence of different tail structures on aerodynamic drag and lift force was analyzed by using the orthogonal test method, which took the mutual effect of each tail structure into consideration, providing the basis for the design and optimization of the aerodynamic structure of the tail.

2. Body Modeling
In this paper, the internationally standardized hatchback MIRA model [14] (motor industry research association) is adopted with standard parameters shown as figure 1. The rear structures include the side windshield angle, the roof angle, the rear windshield angle, the trunk length, the rear end face angle and the diffuser angle. Considering the actual car body, a rough range is defined to make the change of the structures have certain practical significance. The structural parameters of each tail are defined as shown in figure 2. The ranges are listed in table 1.

![Figure 1. Standard MIRA model.](image)

![Figure 2. Parameters definition of the tail.](image)

| Symbol | Structural parameters | Range       | Symbol | Structural parameters | Range       |
|--------|-----------------------|-------------|--------|-----------------------|-------------|
| $\theta_s$ | Side windshield angle | 60°–80° | $l$ | Trunk length | 410–730mm |
| $\theta_f$ | Roof angle | -4°–+4° | $\theta_r$ | Rear end face angle | -20°–20° |
| $\theta_w$ | Rear windshield angle | 30°–50° | $\theta_d$ | Diffuser angle | 0°–20° |

3. Fundamentals of Numerical Calculation

3.1. The Governing Equation
The flow motion in the field conforms to the law of conservation of mass, the law of conservation of momentum and the law of conservation of energy, from which the Navier-Stokrs (N-S) equations are achieved [15]. In this work, the energy equation is neglected without considering the heat exchange and heat transfer phenomena [16]. And, the air is regarded as in-compressible since the Mach number is low during normal driving condition Therefore, Navier-Stokrs equations can be written as [15,16]

$$\frac{\partial u}{\partial t} + \frac{\partial (pu)}{\partial x} + \frac{\partial (pu)}{\partial y} + \frac{\partial (pu)}{\partial z} = 0$$

$$\frac{\partial}{\partial t} (\rho u_i) + \text{div} (\rho u_i u_j) = - \frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z} + F_i, \quad i = x, y, z$$

where, $p$ is the pressure on the fluid element; $\tau_{xx}$, $\tau_{xy}$ and $\tau_{xz}$ are components of viscous stress $\bar{\tau}$ on the surface of the micro-element caused by molecular viscosity. $F_x$, $F_y$, and $F_z$ are the forces on the element with the assumption that there is only gravity.

To close the N-S equations, some studies have shown that the SST $\kappa-\omega$ model with low-Re correction has a high accuracy, which gives a good prediction about the vortex core position under strong part-loading conditions [17,18]. In this study, SST $\kappa-\omega$ model is selected, which can be expressed as [18]

$$\frac{\partial (\rho W)}{\partial t} + \frac{\partial (\rho u_i W)}{\partial x_i} = \frac{\partial}{\partial x_i} \left[ \left( \frac{A}{\tau_{xx}} + \frac{B}{\tau_{xy}} + \frac{C}{\tau_{xz}} \right) \frac{\partial W}{\partial x_i} - \frac{1}{\tau_{xx}} \frac{\partial F}{\partial x_i} - \frac{1}{\tau_{xy}} \frac{\partial F}{\partial x_j} - \frac{1}{\tau_{xz}} \frac{\partial F}{\partial x_k} \right]$$

where, $A$, $B$, and $C$ are constants depending on the turbulence model parameters.
where, $\mu_t$ is the turbulence viscosity coefficient, $G_k$, $G_\omega$, and $Y_k$, $Y_\omega$ are the generation terms and the dissipative terms of the turbulent kinetic energy and turbulent kinetic energy dissipation rate, respectively, $D_\omega$ is the cross diffusion.

3.2. Meshing and Calculation Conditions
A semi-car model is used since the analysis is not involving asymmetric factors, such as unsteady flow and crosswind. In addition, the rear space of the computational domain is supposed to be long enough to meet the full development of the turbulence. Thus, the size of the computational domain is established as: 3 times the car length for the front domain, 7 times for the rear domain, 3 times the car width for the side domain, and 4 times the car height for the top domain. Considering the tire deformation on the road surface, the bottom surface of the calculation domain would be moved upward to improve the authenticity of the simulation.

The trimming meshing is adopted with 4 layers of layered encryption, whose sizes are defined as 15mm, 45mm, 90mm, 150mm and 240mm. And, the rear domain has additional local encryption. A 3mm boundary layer is set on the near wall and ground, where the thickness of the first layer was 0.1mm. Then, more than 4.8 million trimming grids were generated in the whole simulation computational field. the biggest grid far away from the body model area is 0.512m$^3$. For the near wall, the biggest grid is 2.6×10$^{-8}$m$^3$ while the smallest grid 1.72×10$^{-13}$m$^3$. The computational domain and the meshing model were shown in figure 3.

![Figure 3. Computational domain grids and the encryption.](image)

3.3. Calculation Conditions
The boundary conditions of the external flow field included the entrance, exit, wall surface and boundaries of the symmetric surface. The specific boundary conditions were set in table 2.

| Front boundary       | speed entrance 30m/s | half body | non-sliding wall |
|----------------------|----------------------|-----------|------------------|
| Rear boundary        | Pressure exit zero   | Boundary of the symmetrical surface of the half-body | Symmetrical surface |
| Top and side boundaries | Sliding wall      | Relaxation factor | Speed: 0.7, Pressure: 0.1 |
| Bottom boundary      | Non-sliding wall 30m/s | --        | --               |

4. Numerical Simulation Analysis
Numerical simulation showed that the resistance coefficient of the standard MIRA model was about 0.31313. Compared with the experiment data of the HD-2 wind tunnel of Hunan University and the IVK car model wind tunnel of the University of Stuttgart [19, 20], the error of the simulated values is approximate to 2.99% and 2.32% respectively. This error is mainly caused by the smooth surface set in the computer CFD simulation. It is believed that the simulation results are reliable, and the simulation work is effective for aerodynamic analysis of the MIRA model.

4.1. Influence of Tail Structures on the Rear Field
In order to reveal the tail structure to the rear field, the chosen tail structure is discussed individually first. Figure 4 shows the tail pressure clouds under various side windshield angles.
It is noted that the change of the side windshield angle has an impact on the wake field. There is an airflow separation at the C-pillar and the rear edge of the roof, which results in a long and narrow negative pressure center along the C-pillar and the rear edge of the roof. Due to the shielding effect of the car body, a large negative pressure area is generated between the upper trunk and the rear windshield. And, this airflow separation tended to be weakened with the side windshield angle increasing, which makes the pressure at the negative pressure center decrease. Compared with the pressure distribution, it can be inferred that the vortex caused by the separating airflow at C-pillar becomes stronger with the windshield angle reduce, which enhances the take-up effect and constrains the vortex core to the body.

Figure 5 showed the tail pressure clouds under different roof angles. The change of the roof angle had a direct impact on the airflow separation on the front and rear edges of the roof. As shown in figure 5, the negative pressure and its area on the surface of the rear windshield and the upper trunk decreased as the roof angle changes. The reason for this was that the rear roof curvature decreased with the roof angle change from positive to negative, which reduces the separation gradient of the airflow. In addition, the negative roof angle plays a guidance role on the top airflow, which is advantageous to the vortex reduction of the wake flow as well as the improvement of the aerodynamic performance.

Figure 6 showed the tail pressure clouds under various rear windshield angle.

As figure 6 shows the negative pressure center at the upper edge and the lateral edge rapidly decrease with the rear windshield angle increasing, the maximum negative pressure center appears at the 30° rear windshield angle. It is inferred that the upper flow separation was the strongest with a forward movement of the separation position. When the rear windshield angle reaches 40°, the strength of the upper flow separation quickly weakens. On the rear windshield, the negative pressure area showed a trend of first decreasing and then increasing, reaching the minimum at 35°. A small area of positive pressure appeared on the upper surface of the trunk, which gradually decreased as the rear windshield angle increased. It is concluded that the rear windshield may be designed with a proper small angle may reduce influence of upper airflow separation position.

Figure 7 showed the tail pressure clouds under different trunk length.

It is noted that the change of trunk length had little impact on the separation of the airflow at the top and the bottom. The negative pressure center at the rear edge of the roof and the rear edge of the side windshield with different trunk lengths is basically unchanged. The negative pressure area on the rear windshield surface increases first and then decreased with the growth of the trunk length, while the negative pressure area on the rear end surface decreased.
Figure 8 shows the rear airflow velocity vector on the symmetrical plane under various rear end face angles.

The change of the rear end face angle has a different influence on the separation flow of the upper part and lower part. When the tail end was positive, the airflow above the rear end face flows along the rear end face and the trunk surface. Two tail vortices formed at the rear space since the airflow converges lower separation flow. As the lower separation flow continued backward movement and interacted with the upper separation flow at the tail, generating a larger oval eddy current near the tail. When the tail end face angle was negative, the airflow was guided by the tail angle, which made the elliptic vortex elongate to the lower part of the tail without good aerodynamic performance.

Figure 9 is the flow velocity vector diagram of the symmetrical plane of the tail with different diffuser angles.

As shown in figure 9, the bottom airflow moved backward along the floor after separation resulting in a large triangular tail vortex area for the rear structure without diffuser angles. When the diffuser angle is 5°, the bottom airflow flowed slightly upward after separation. The rear vortex area is rapidly decreasing and forming several small elliptic eddies at the bottom. With the increase of the diffuser angles, the small elliptic eddies grow making the turbulent kinetic energy increase rapidly. When the diffuser angle is larger than 20°, the small oval vortex at the bottom of the tail fuses in together forming a large tail vortex center. At the same time, the upper vortex would be drawn into a long strip and the integration of the tail turbulent kinetic energy increased sharply, consuming a lot of energy.

4.2. Orthogonal Optimization Test Design

The airflow separation at the tail and the generation of wake vortices are the interaction of different structures. To analyze the aerodynamic characteristic realistically, the orthogonal test is applied to consider these interactions. Based on above aerodynamic analysis, the variation level of each structural factor is determined according to a commonly used orthogonal test table L25(5^6), as shown in table 3. The statistical results were listed in table 4.

| Level | \( \theta_s \) | \( \theta_f \) | \( \theta_w \) | \( l \) | \( \theta_r \) | \( \theta_d \) |
|-------|----------------|----------------|----------------|---------|----------------|----------------|
| 1     | 75°            | -1°            | 33°            | 670mm   | 4°             | 3°             |
| 2     | 77°            | -2°            | 35°            | 685mm   | 7°             | 4°             |
| 3     | 79°            | -3°            | 37°            | 700mm   | 10°            | 5°             |
| 4     | 81°            | -4°            | 39°            | 715mm   | 13°            | 6°             |
| 5     | 83°            | -5°            | 41°            | 730mm   | 16°            | 7°             |

The body models of the orthogonal test are established according to table 3. When completing the calculation, the average value of aerodynamic coefficients in each level can be obtained by results statistics. Figure 10 presents the specific average value of aerodynamic coefficients. From figure 10, it is easy to capture the optimal level of each of the factors, which can form a relatively good aerodynamic structure of the rear part of the vehicle. This contributes to improving the design of the vehicle aerodynamic characteristics. Meanwhile, it is noted in figure 10(a) in that \( c_w \) has a maximum change among the different levels of \( \theta_s \), which suggests \( c_d \) is major affected by the rear windshield angle. While it can found that \( c_f \) is a major influence by the diffuser angle \( (\theta_d) \) from figure 10(b).
To analyze the detailed influence of factors on the aerodynamic characteristics, the range and mean range (the aerodynamic coefficient range/the level range) of the aerodynamic drag and lift coefficient was calculated as shown in figure 11. It is noted that the range of $c_f$ under different levels is much larger than that of $c_d$, which indicates the rear structure has a larger influence on the lift. When analyzing the $c_d$ alone, it can be found that the change of $\theta_w$ and $\theta_d$ have a significant influence on the drag coefficient, which leads to the ranges about 0.01558 and 0.00943, respectively. The mean range of the aerodynamic drag coefficient is most influenced by $\theta_w$ while it is the most sensitive to $\theta_d$. The ranges of the drag coefficient of other tail structures are similar while the mean range value of the aerodynamic drag coefficient is: $\theta_f > l > \theta_s > \theta_r$. For the lift coefficient, the range under $\theta_d$ is much larger than others, which reveal the maximum influence of the diffuse angle. $\theta$ and $\theta_d$ have a similar change between the different levels, while the sensitivity of the lift coefficient to $\theta_d$ was about 2 times as much as that to $\theta_w$. This means it is easier to improve the vehicle aerodynamic by adjusting the roof angle. The influence of other tail structures on the aerodynamic lift was slightly smaller, and the sequence was $\theta_s > l > \theta_d$.

5. Conclusion

(1) The different part of the tail structures generated airflow separation at a different position, which make the wake complex due to the mutual influence of each airflow. It is important to consider these mutual influences in the rear field analysis.

(2) The tail structures have an obvious influence on the aerodynamic drag coefficient. The rear windshield angle has the maximum effect on the aerodynamic drag coefficient, while the diffuser angle has the maximum sensitivity to the drag coefficient. The influence of other tail structures on the drag coefficient is relatively little.

(3) The tail structures have a large influence on the aerodynamic lift coefficient. The influence of the diffuser angle on the aerodynamic lift coefficient was much greater than that of other tail structures. The aerodynamic lift coefficient was similarly influenced by the roof angle and the rear windshield angle; however, it was more sensitive to former change. Other tail structures had a relatively small impact.

Acknowledgments

This work was supported by the Science and Technology Research Program of Chongqing Municipal Education Commission (Grant No.KJQN201803408), Science and Technology Research Program of
Chongqing Municipal Education Commission (Grant No.KJ1603207) and Institution-level scientific research project of Chongqing vocational institute of Engineering (Grant No.KJA201903).

References
[1] Fu L M. Automobile design and aerodynamics[M].Beijing: China machine press,2011:50-70.
[2] Xu J M. Influence of Drag Reducing Device on Aerodynamic Characteristics of Van Type Truck[J]. Mechanical Science and Technology for Aerospace Engineering,2018,37(6): 941-948.
[3] Li S Q ,Zhao Y B, Feng X Y,et al. Optimization design of vehicle bottom structure based on the vehicle external flow field performance[J].Forest Engineering,2019,35(3): 100-106.
[4] Blocken B, Van Druenen T, Toparlar Y, et al. Aerodynamic drag in cycling pelotons: New insights by CFD simulation and wind tunnel testing [J]. Journal of Wind Engineering and Industrial Aerodynamics, 2018, 179: 319-337.
[5] Bell J R, Burton D, Thompson M C, et al. The effect of tail geometry on the slipstream and unsteady wake structure of high-speed trains[J]. Experimental Thermal and Fluid Science, 2017, 83: 215-230.
[6] Zhang Y C, Du G Y, Tian S, et al. Active flow control of 350 ahmed model to reduce aerodynamic drag with steady jet[J]. Journal of Jilin University(Engineering and Technology Edition) , 2019, 49(02): 351-358.
[7] Baek S W, Lee S W. Aerodynamic drag reduction on a realistic vehicle using continuous blowing[J]. Microsystem Technologies, 2020, 26: 11-23.
[8] Eulalie Y, Fourier E, Gilotte P. Active flow control analysis at the rear of an SUV[J]. International Journal of Numerical Methods for Heat & Fluid Flow, 2018, https://doi.org/10.1108/HFF-06-2017-0230.
[9] A. Buljac, I. Džijan, I. Korade, et al. Automobile aerodynamics influenced by airfoil-shaped rear wing [J]. International Journal of Automotive Technology, 2016, 17(3), 377-385.
[10] Kounenis C, Bonitz S, Ljunskong E, et al. Investigations of the Rear-End Flow Structures on a Sedan Car[J]. SAE Technical Paper 2016-01-1606, doi:10.4271/2016-01-1606.
[11] Liu C Y,Xie J F,Yuan C Y, et al. Numerical analysis of wake structure for hatchback car[J].Journal of System Simulation,2009,21(21):6897-6900.
[12] Wang J ,Yang Z G, Zhu H, et al. Simulation analysis of effect of typical angels on automotive aerodynamic drag[J].Computer Simulation,2013,30(10): 184-186.
[13] Wu L L, Xu X B, Zhang X L, et al. Study on the influence of the vehicle rear plan on wake flow field[J]. Journal of Physics: Conference Series, 2020, 1601, 062039.
[14] Su C ,Han Y ,Zhang Y C, et al. Study on the influence performance with wheel spoke design parameters of vehicle aerodynamic[J]. Journal of Jilin University(Engineering and Technology Edition) , 2020,https://doi.org/10.13229/j.cnki.jdxgxbl20190960.
[15] Zhou H, Li X L, Qi H, et al. Subgrid-scale model for large-eddy simulation of transition and turbulence in compressible flows[J]. Physics of Fluids, 2019, 31, 125118.
[16] Yang L, Shi Q S, Wei H, et al. CFD simulation of the flow field for SUV [J]. Beijing Automotive Engineering,2017, 1:39-42.
[17] Zhou L, Bai L, Li W, et al. PIV validation of different turbulence models used for numerical simulation of a centrifugal pump diffuser[J]. Engineering Computations, 2018, 35(1), 2-17.
[18] Zhang F R, Zhang J L, QU X, et al. Numerical simulation on the influence of different front window angle of automobile stability in crosswind[J]. Journal of Automotive Safety and Energy, 2015, 6(2):145-149.
[19] Gu Z Q, Wang S, Qiu J, et al. Wind tunnel tests of MIRA model group for study of vehicle’s rear shape[J].Science and Technology Review,2011, 29(8): 67-71.
[20] Han Y, Xie J P, Lu X P, et al. Parameter optimization of CRS’ seatbelt constraint paths based on design-of-experiment (DOE) [J]. Journal of Automotive Safety and Energy, 2015,6(3):230-236.