Aerodynamic Simulation and Optimization Design of Vehicle External Flow Field Based on Fluent

Yuan Lei[a,*], Ran Junjun

The Engineering & Technical College of Chengdu University of Technology, Leshan, Sichuan, 614000, China

[a] email: yuanlei11@163.com

*corresponding author

Keywords: Outflow Field, Air Lift, Automobile Body, Aerodynamic Performance

Abstract: Based on the vehicle parameters, the three-dimensional models of the body and wind tunnel are established and meshed, and the finite element model of the body aerodynamics is established. The K - ε turbulence model is used, and the coupling solver is used to simulate the flow field of the body, and the structural performance parameters that affect the flow field of the body are obtained. The structural factors that affect the aerodynamic changes of the body are simulated Based on the analysis, the aerodynamic modeling of the original car body model is optimized. The simulation analysis of the improved car body shows that the aerodynamic drag and the aerodynamic lift are reduced, which verifies the effectiveness of the improved method and provides a new reference method for the optimization research of the car body.

1. Introduction

As an important part of automobile, automobile body has always been the focus of automobile development. The power, fuel economy and handling stability of automobile are all affected by aerodynamic force[1]. Therefore, automobile aerodynamics has been paid more and more attention. How to obtain a good aerodynamic automobile body has become an important topic in modern automobile industry, and the analysis of automobile external flow field is An important way to optimize the aerodynamic modeling of automobile body

When the car is running at high speed, the aerodynamic resistance accounts for a large proportion of the total driving resistance, so it is necessary to reduce the aerodynamic resistance coefficient of the car body. A lower aerodynamic resistance coefficient can reduce the aerodynamic resistance of the car, effectively improve the power of the car, and reduce its fuel consumption and exhaust emissions. Due to the difference of air flow speed between the upper part and the lower part of the car, it is necessary to reduce the aerodynamic resistance coefficient of the car body In addition, the reasonable aerodynamic layout of the car can also improve the operation stability of the car at high speed

This paper mainly studies the design of automobile body based on aerodynamics and the analysis of its flow field. Driven by the rapid development of CAD / CAM and CFD technology, the design technology of automobile body is constantly improved[2]. The design of automobile body based on aerodynamics and the analysis of external flow field can make the body have better aerodynamic characteristics, So that the vehicle can have a lower aerodynamic drag coefficient and aerodynamic lift coefficient, improve the vehicle's power, fuel economy and handling stability

2. Simulation and Analysis of Vehicle External Flow Field

2.1. SIMULATION MODEL ESTABLISHMENT OF VEHICLE BODY AND WIND TUNNEL

Due to the symmetry of the car body, in order to improve the efficiency of the calculation, half of the car body is used for analysis, so as to reduce the amount of calculation and save time.

  DOI: 10.25236/cseem.2020.146

Because of the existence of multiple curved surfaces and small parts in the original car model, it will bring difficulties to the mesh division, and make the quality of the mesh reduced, so it is difficult to converge in the calculation. Therefore, in the modeling process, smooth curved surfaces are used to replace the air intake grille; the The vehicle bottom is simplified as a smooth surface; the small parts of the vehicle body, such as the door handle and the wiper, are removed; the small gaps of the vehicle body are filled with smooth surfaces; the wheels are replaced by simple cylinders, and steps are built at the places where the tire contacts the ground to simulate the bearing deformation of the tire, so as to prevent the grid distance between the tire surface and the ground too close to generate the correct boundary layer

When analyzing the flow field outside the car body, it is necessary to build a simulation wind tunnel outside the car body. Generally, the simulation wind tunnel model is a cuboid. The top of the wind tunnel is three times the height of the car, the side wall is three times the width of the car, the air inlet is three times the length of the car, and the air outlet is six times the length of the car, And the intersection of the wall and the body is divided

2.2. ESTABLISHMENT OF FINITE ELEMENT MODEL OF BODY WIND TUNNEL

The 3D model of wind tunnel is imported into ANSYS to mesh. The air outside the car body, that is, the volume between the wind tunnel and the car body, needs to be meshed. In this paper, the car body model is divided into nine parts: front face, engine hood, front window and roof, rear window, side window, trunk top surface, car body and tire[3], rear end and rear-view mirror, and according to the The minimum feature size is used to construct the surface mesh. Then the boundary layer mesh is stretched and filled. The boundary layer mesh of the body surface is set to 4 layers, the initial layer thickness is set to 0.2mm, and the growth rate is set to 1.2. Then the volume mesh is filled, and the total mesh number is 1346227.

2.3. SIMULATION ANALYSIS

Based on the standard k - ε turbulence model, the vehicle is set to run at the speed of 100km / h, and the fluent coupling solver is used to simulate the external flow field of the vehicle body. The iteration steps are set to 1000, and the iteration steps are 510 The residual value is basically stable, i.e. the mass at the entrance and the mass at the exit are basically equal, which can be regarded as convergence. The main pressure bearing areas of the car body are the intersection of the front face, rear-view mirror, engine hood and front window, and the intersection of the rear window and the top of the luggage compartment[4]. This may be caused by the sudden change of the direction of motion of the air flow when it encounters a prominent obstruction during the operation. The optimization of the car body It can be improved from these aspects

The air flow is divided into two parts in the front of the car. The upper air flow passes through the hood, which is affected by the resistance when turning to the front window to reduce the speed and form a separation area. Then, the air flow is combined above the intersection of the hood and the front window and flows to the roof. At this time, the air flow reaches the maximum value. The air flow is separated again when the rear window crosses the top of the luggage compartment to form a low-speed separation area, and then flows through the rear of the car and Separated from the vehicle body, it forms the maximum eddy current at the rear of the vehicle, while the lower air flow is separated from the vehicle at the rear through the vehicle chassis

3. Analysis of the Influence of Vehicle Structural Parameters on Aerodynamic Force

According to the above analysis, the main pressure bearing areas of the car body are the front face, rear-view mirror, the intersection of the hood and the front window, and the intersection of the rear window and the top of the luggage compartment. The three main vortex areas are respectively located at the intersection of the hood and the front window, the intersection of the rear window and the top of the luggage compartment[5], and the rear of the car. Among them, the rear-view mirror is not easy to modify as a car accessory, while the other eight feature areas are all It can be used as the optimization direction of body modeling

Feature optimization needs to ensure that the general layout of the car is not affected, and the basic shape does not change. Therefore, only eight features of the analysis are modified in a small range to carry out analysis and optimization. This analysis is a single factor analysis, each feature size is adjusted up and down, and then the simulation analysis is carried out, a total of 16 times of simulation is required. In order to facilitate statistics, to Each feature is numbered

Modify the two-dimensional model, import it into fluent for simulation operation, solve the parameters to get the simulation results of aerodynamic drag and aerodynamic lift, reduce the aerodynamic drag coefficient by increasing the inclination of the front and bottom of the car, increasing the inclination of the front window of the car, increasing the inclination of the rear window of the car, reducing the height of the trunk, increasing or reducing the inclination of the bottom of the car Among them, the most effective optimization is to increase the ground clearance

The most effective way to reduce the aerodynamic lift coefficient is to increase the inclination of the front and bottom of the car, reduce the inclination of the front and rear windows, increase the height of the trunk, increase the inclination of the bottom of the car tail, increase or reduce the deflection coefficient of the rounded corner of the car top[6], and increase or reduce the ground clearance.

It can be seen from the simulation that in order to improve the dynamic performance of the vehicle body, the aerodynamic lift coefficient can be reduced while the aerodynamic drag coefficient can be reduced, and the scheme of increasing the inclination of the front and bottom of the vehicle, increasing the deflection coefficient of the top corner of the vehicle body and increasing the ground clearance can be adopted [7]. However, increasing the ground clearance will reduce the internal space of the vehicle and reduce the comfort of the driver and the passenger The final optimization scheme of the car body is to increase the head bottom slope, the tail bottom slope and the upper deflection coefficient of the body top fillet[8].

4. Comparative Analysis before and after Optimization

The boundary conditions and solving parameters are the same as those of the original model. The eddy current at the tail of the optimized model is obviously smaller than the original model. Compared with the original model, the aerodynamic drag coefficient is reduced by 1.83%, and the aerodynamic lift coefficient is reduced by 12.8919% The comparison between the aerodynamic lift and the original model ,As shown in Figure 1[9].

parameter	Before improvement	After improvement
Speed	100	100
Windward area	1.045	1.04
Aerodynamic drag	133.25	130.81
Aerodynamic coefficient	0.26	0.26
Starting lift	54.93	48.17
Lift coefficient	0.11	0.09

Figure 1 Aerodynamic force of body simulation before and after improvement

5. Conclusion

In this paper, the aerodynamic optimization design of automobile body modeling is carried out based on aerodynamics. Firstly, through the three-dimensional model of vehicle body and wind tunnel, the original vehicle body model and local feature modification model are simulated by fluent solver, and the optimization scheme of aerodynamic modeling of the original vehicle body model is found out, that is, the inclination of the front bottom is 5 °, the inclination of the rear bottom is 11 °, and the inclination of the vehicle body is 5 °The top deflection coefficient is 0.028. Compared with the original model, the aerodynamic drag coefficient of the optimized model is reduced by 1.83%, and the aerodynamic lift coefficient is reduced by 12.8919%

References

[1] Xia Lu, Bingwen Long, Yigang Ding. (2018). Experimental Study and CFD-PBM Simulation of the Unsteady Gas-Liquid Flow in an Airlift External Loop Reactor. Flow Turbulence and Combustion, vol. 102, no. 3.

[2] Suzuki K, Yoshino M. (2017). Aerodynamic comparison of a butterfly-like flapping wing–body model and a revolving-wing model, vol. 49, no. 3, pp. 035512.

[3] Mirghorayshi M, Zinatizadeh A A, Van L M. (2018). Evaluating the process performance and potential of a high-rate single airlift bioreactor for simultaneous carbon and nitrogen removal through coupling different pathways from a nitrogen-rich wastewater, vol. 260, pp. 44-52.

[4] Mei Han, Arto Laari, Tuomas Koiranen. (2017). Effect of Aeration Mode on the Performance of Center- and Annulus-Rising Internal-Loop Airlift Bioreactors. Canadian Journal of Chemical Engineering, vol. 96, no. 1.

[5] Jian Zhang, Xinyan Deng. (2017). Resonance Principle for the Design of Flapping Wing Micro Air Vehicles. IEEE Transactions on Robotics, no. 99, pp. 1-15.

[6] Korkischko I, Konrath R. (2017). Formation Flight of Low-Aspect-Ratio Wings at Low Reynolds Number, vol. 54, no. 3, pp. 1-10.

[7] HASSANALIAN, R. SALAZAR, A. ABDELKEFI. (2019). Conceptual design and optimization of a tilt-rotor micro air vehicle. Chinese Journal of Aeronautics: English Edition, no. 2, pp. 369-381.

[8] Xiang Li, Yong Huang, Yi Yuan. (2017). Startup and operating characteristics of an external air-lift reflux partial nitritation- ANAMMOX integrative reactor. Bioresource Technology, vol. 238, pp. 657.

[9] Mei Han, Gerardo González, Marko Vauhkonen. (2017). Local gas distribution and mass transfer characteristics in an annulus-rising airlift reactor with non-Newtonian fluid. Chemical Engineering Journal, vol. 308, pp. 929-939.