ISSN: 2688-8645

Research on Aerodynamic Performance Optimization and Drag Reduction of mid-sized SUV on CFD

Xiaoxiao Yan[1], Zhuo Ma[1], Xuelong Liu[1], Li Jiang[2]

[1]CATARC (Tianjin) Automotive Engineering Research Institute Co., Ltd Tianjin 300300, China

[2]CATARC(Tianjin) Automotive Information Co., Ltd., Tianjin 300300, China

Abstract

With the increase of vehicle ownership, the environment and energy issues are highlighted, and reducing vehicle wind resistance has become the key to improving fuel economy. Based on the theory of fluid mechanics and project development experience, this paper develops a medium-sized SUV model for wind resistance reduction. Through the simulation modeling of the vehicle's external flow field, the proportion model wind tunnel test and the optimization scheme research of the exterior shape, the bottom of the vehicle and the engine room, the wind resistance is significantly reduced. Combined with the wind tunnel test results, the actual contribution of each local optimization scheme is verified, and the EGO algorithm is used for further optimization to further reduce the wind resistance coefficient. This paper summarizes the key parts and regularity experience of the aerodynamic drag reduction of the medium-sized SUV model, providing guidance for the development of high-precision vehicle performance in the future. The research methods and results of this paper are of great significance for improving the aerodynamic performance and fuel economy of vehicles.

Keywords

CFD; SUV; Aerodynamic Drag Reduction; Partial Optimization.

1. Preface

In order to improve the aerodynamic performance of cars, automobile companies around the world have introduced a large number of talents and spent a lot of costs to carry out research. The previous aerodynamic characteristics of wind tunnel research, can not fully understand the distribution of vehicle external flow rate, pressure and other parameters, there are shortcomings such as large investment, long test period, blockage effect, ground effect and other problems[1-3].Aerodynamics is a very important aspect in the development of automobile performance. With the improvement of computer simulation technology, more and more automotive research and development personnel began to realize the importance of CFD simulation analysis, and the method has gradually become one of the indispensable means in automotive development.

In recent years, with the continuous development of fluid mechanics theory, CFD has been widely used as one of the important simulation methods in the study of automotive aerodynamics. Many foreign car manufacturers, such as Ford, incorporated computational fluid dynamics into their Ford product development processes in 1993. Through years of experimentation, Ford has dramatically reduced the development cycle for new models. In Jilin University, wind tunnel test and numerical simulation were used to analyze the aerodynamic characteristics of a 1:5 reduction model of a pickup truck, and to make a comparative study of the SUV with a hardtop modified pickup truck. The study results show that the wake of pickup trucks is more complex than that of SUVs. The test provides an important basis for the research of drag reduction of these two models[4-7].China's research on automotive aerodynamic characteristics is relatively late, and is in the initial stage of development, but with the reform of the global automotive industry, automotive aerodynamic research has gradually attracted the attention of automobile manufacturers and even the whole country, in this context, China's research investment in automotive aerodynamics will be significantly increased, thus laying the foundation for promoting independent research and development in the automotive field.

Based on the background of a medium-sized SUV development project, the research content can be divided into two aspects, namely wind tunnel test and CFD numerical simulation calculation. The CFD simulation results of the developed vehicle base model were compared with those obtained from the wind tunnel test to verify the reliable numerical simulation methods, models and results. Then, based on the analysis model after the benchmark, CFD simulation analysis is used to perfect and optimize the aerodynamic characteristics of the vehicle. Combined with the existing project development experience, first of all, the vehicle outflow field of a medium-sized SUV model was simulated and analyzed. Under the premise of meeting the product definition and general layout requirements, the optimization of local external modeling details was studied, and the front and rear wheel choke plates were studied. Wheel well inner seal plate; The cabin, grille seal plate and other parts of the vehicle chassis, tire and cabin resistance reduction optimization plan, combined with the final results of wind tunnel test to obtain the actual contribution of each plan to reduce the aerodynamic resistance of the vehicle, through the summary, the similar medium-sized SUV model in the process of wind resistance reduction needs to focus on the content and form a certain regular experience. It provides effective support for the development of higher precision vehicle performance in the future.

2. Esearch Methods and Research Contents

There are three research methods for aerodynamics: numerical calculation, theoretical research and experimental research. Experimental research is the basis of aerodynamic research, but also a necessary method of testing, which can be verified for theoretical research and numerical calculation, so it plays an important role in the accuracy and reliability of research results. At the same time, the theoretical research is also of great significance to the research of automotive pneumatics, which can be used as the theoretical basis for numerical calculation and experimental research.

For theoretical research, its basic idea is to analyze the basic factors that affect the research problem, and find the main factors among all the factors, so as to build a reasonable theoretical model. Secondly, according to the relevant physical laws and experimental formulas, the corresponding integral differential equation is constructed to reveal the motion law of the medium. Finally, the solution of the equations is obtained by means of mathematical calculation tools and known initial conditions, and the change law of physical quantities is summarized according to the result of the solution, and the applicable conditions of the solution are obtained. Since the mathematical calculation technology of our country is relatively backward compared with other advanced countries, it is impossible to establish the aerodynamic characteristics model with high accuracy and complex structure at present, which leads to certain differences between the built model and the actual situation. The emergence of CFD method effectively solves the above problems. CFD method is the product of the rapid development of information technology and computer technology. As an important part of CAA (Automotive computer-aided Aerodynamics) design, CFD method plays an important role in test and theoretical analysis.This method has the following advantages: it can effectively solve complex flow problems, reduce development cost, do not need test support and short development cycle. Before using CFD method, we should first fully understand the various physical characteristics of the research object and establish more accurate mathematical equations. However, in general, the above processes need to carry out necessary experimental and theoretical research, in addition, this method is only applicable to the study of specific problems, and can not get conclusions on similar problems. Wind tunnel test is the basis for the study of automobile aerodynamic characteristics, and the test results are also applicable to the study of similar problems. At the same time, the test can also be used to verify the accuracy and application range of calculation and theoretical research results, which has important guiding significance for the construction of models and the revelation of motion laws. However, the cost of wind tunnel test is high, so it is not suitable for the study of some problems. It can be seen that the three methods of experiment, theoretical research and numerical calculation complement each other and promote each other, and these three methods jointly promote the progress of vehicle starting characteristics research.

3. Simulation Analysis of Vehicle Outflow Field

3.1. Simulation Analysis Process

With reference to the development process of aerodynamics, the aerodynamic characteristics of automobile outflow field are analyzed. In this process, CFD numerical simulation technology is used, and the analysis model is benchmarked by wind tunnel test. Figure 1 shows the specific analysis process.

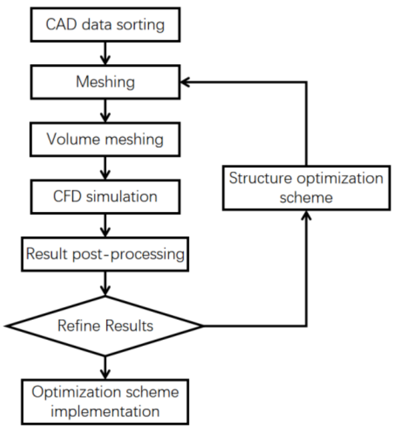


Figure 1. simulation analysis process

3.2. CFD Model Building

A cuboid is selected as the calculation domain, the car is in the cuboid, the entrance to the front end of the car and the exit to the back end of the car are 3 times and 6 times the length of the vehicle, the total width and total height are 7 times the width of the car and 5 times the height of the car. The calculation domain diagram is shown in Figure 2.

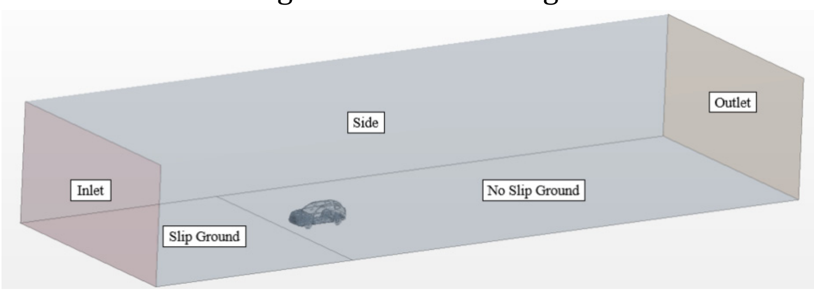


Figure 2. Calculation domain view

3.3. CFD Simulation Results and Analysis

Model convergence judgment,the calculation steps of the entire model are 4000 steps. After the calculation is completed, the sum of the calculation results should be judged to ensure the convergence. CFD analysis software generally uses several methods to determine whether the calculation is convergent, such as residual value and drag coefficient. Using a single method to judge convergence may be effective in some cases, but it may lead to wrong conclusions in other cases. Therefore, the convergence of the calculation should be tested, and the test indexes are the comprehensive resistance coefficient and residual value. The value of the residual curve is the average value of the whole solving process and the result of CFD simulation. The residual value has a strong correlation with the number of iterative steps, and its value decreases and increases with the number of steps, and the fluctuation is less than 1count.

Body surface pressure analysis,the pressure on the surface of the car is related to a variety of factors, and it will have a certain impact on the aerodynamic characteristics of the entire car to a certain extent. The analysis results show that the position of the intake grille and the bottom of the front windshield (as shown in Figure 3) are positive pressure areas, which play an important role in the generation of aerodynamic resistance, and at the same time, they are also important areas to withstand the impact of air flow. As shown in Figure 4, areas such as front and rear wheels, rear edge of the body and A-pillar are negative pressure areas. Under the influence of air separation, these areas may produce vorticity, which will cause energy loss and ultimately reduce the aerodynamic performance of the car.

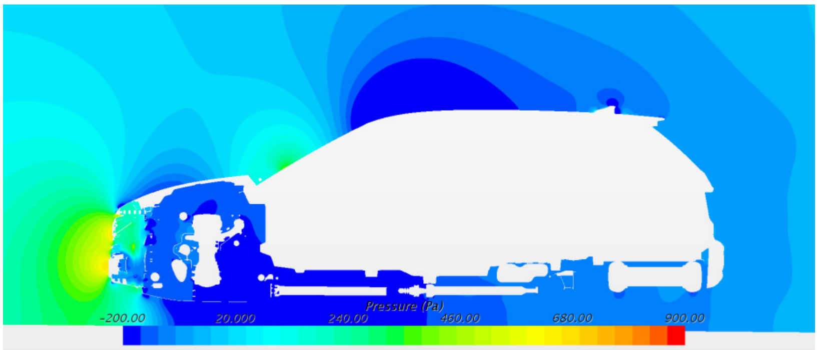
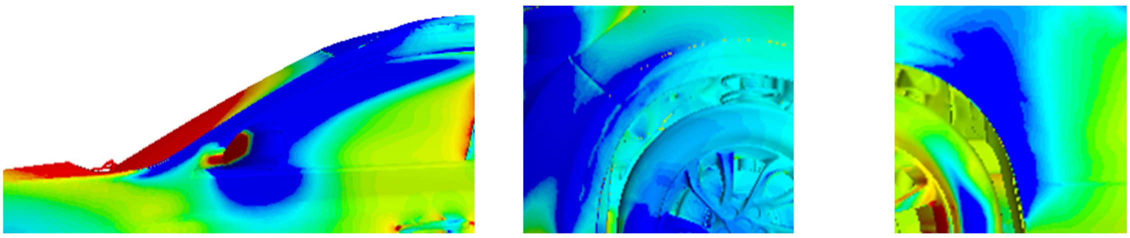


Figure 3. center (Y=0) velocity cloud image



(a) Pressure cloud image of the front windshield and pillar A area (b) Front rim pressure cloud map (c) Rear rim pressure cloud map

Figure 4. Cloud image of vehicle surface pressure

The rear of the car body forms a large area of negative pressure. It is mainly caused by the shielding effect of the SUV, and the emergence of the negative pressure zone leads to the vortex in the tail of the car, which causes a lot of energy loss. As the vortex develops further, the negative pressure zone will continue to spread to the rear of the vehicle.

Wind tunnel model calibration,In order to verify the validity of the CFD model built, the development project adopted 40% full detail scale model to calibrate the CFD simulation analysis model. The wind tunnel of the model test is an American vehicle scale model wind tunnel laboratory, which is a single belt moving road simulation system, mainly focusing on the development of racing aerodynamics. Six component force and surface tests were carried out

ISSN: 2688-8645

during the test, the main purpose of which was to benchmark the wind resistance coefficient of the developed model.

Wind tunnel test, the original wind resistance coefficient of wind tunnel test is less different than that of simulation model wind resistance system, which is within the error range, thus demonstrating the effectiveness of the model. The subsequent reduction and optimization schemes of CFD simulation are carried out on the basis of this scale model to verify its effectiveness.

4. Vehicle Aerodynamic Performance Optimization

There are many factors influencing vehicle aerodynamic performance that can be optimized, including vehicle modeling, local detail modeling, chassis layout, engine compartment layout, etc. The optimization method adopted in this paper is as follows: First of all, the vehicle modeling is analyzed and optimized, and then the details of the modeling are designed and optimized, and then the external modeling is confirmed. At the same time, the chassis and engine compartment are analyzed and optimized, and finally the optimization scheme meeting the aerodynamic characteristics of the vehicle is formed. In the process of analysis and optimization, these programs need to constantly communicate with modeling, design and other departments and be recognized by everyone, and gradually superimpose the optimization programs of each part according to the optimization process of the vehicle outflow field to form the final implementation plan. The development flow chart of vehicle aerodynamics is shown in Figure 5.

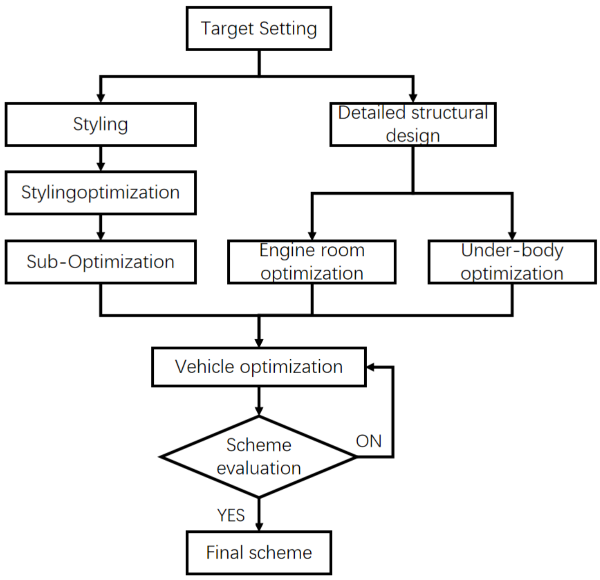
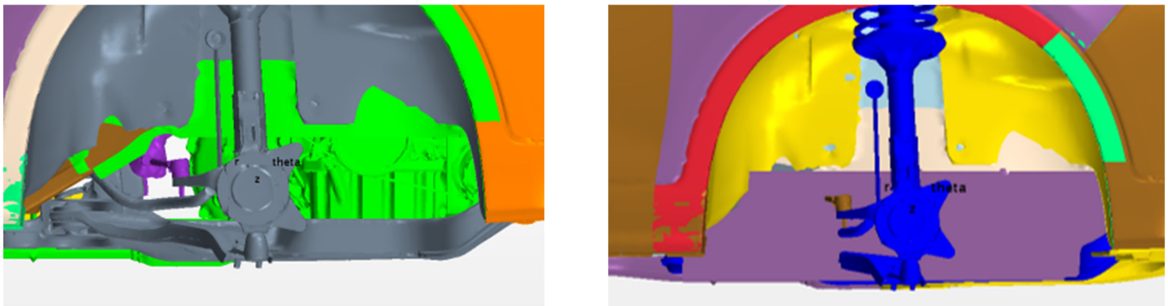


Figure 5. Development flow chart of vehicle aerodynamics

4.1. Optimization of Vehicle Bottom

After the optimization of the external shape of the vehicle is completed (the shape is basically set), the next work is mainly through the systematic optimization of the vehicle chassis layout, so as to further reduce the external aerodynamic resistance caused by the impact of air flow on the bottom of the vehicle. The principle of the chassis optimization scheme is to reduce the air flow through the bottom of the car, smooth the bottom of the car to reduce the impact of the air flow, and advance the separation of the air flow to achieve the purpose of reducing the drag.

Sealing plates on both sides of the wheel well: there are many exposed parts on both sides of the wheel well of this SUV model, and the wheel well part is a part with complicated air flow. Increasing the sealing plates on both sides of the wheel well can effectively reduce the gas leakage in the wheel well, thus reducing the disturbance of the tire on the air flow and reducing the energy loss at the same time to achieve the effect of reducing drag. Through simulation analysis, the sealing plates on both sides of the wheel well can reduce the vehicle wind resistance by 1count. The specific scheme is shown in Figure 6.

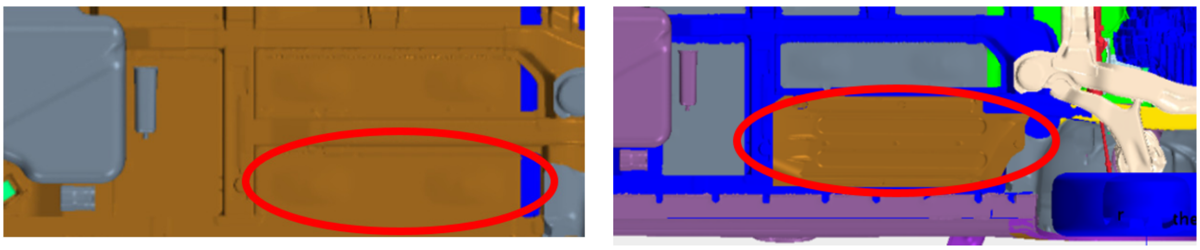


(a) Base version model (b) Add an optimization scheme model for sealing plates

Figure 6. Model comparison before and after implementation of optimization scheme

The wind tunnel test shows that the wind resistance coefficient does not change, so the conclusion combined with simulation and test is that the sealing plates on both sides of the wheel well have no effect on the wind resistance coefficient, and the scheme is treated as invalid.

The left deflector at the bottom of the car body: there are many parts at the bottom of the vehicle, which are uneven and can not make the air flow smoothly through the bottom of the vehicle. In order to reduce the impact of air flow on the chassis, it is necessary to hide the components of the chassis, one of the means is to increase the front choke plate at the front bumper so as to reduce the wind resistance of the vehicle. In addition, the chassis deflector can be added, so that the bottom air flow of the car is smooth, to achieve the purpose of reducing drag. The comparison of simulation models before and after the implementation of the optimization scheme of the left guide plate at the bottom of the car is shown in Figure 7. Through the simulation results, the wind resistance of the vehicle is reduced by 3count, and the effect is obvious.



(a) Base simulation model (b) Add the simulation model of the left deflector at the bottom of the car body

Figure 7. Comparison of simulation models before and after the implementation of optimization scheme

Wind tunnel test verification: The test verification also measured that the vehicle wind resistance coefficient decreased by 4count, and the simulation and test results were basically consistent, which proved that the optimization scheme was effective.

Rear wheel choke plate, According to the simulation analysis results of the Base version (without adding the rear wheel choke plate simulation analysis results are shown in Figure 8), the pressure difference between the front and side of the two rear wheels is relatively large,

ISSN: 2688-8645

which will cause the air flow to produce a large pressure difference resistance to the rear wheels, which will significantly increase the resistance coefficient of the vehicle. Installing rear wheel choke plate is one of the effective means of drag reduction optimization design. The final analysis result of specific optimization scheme is shown in Figure 9 below.

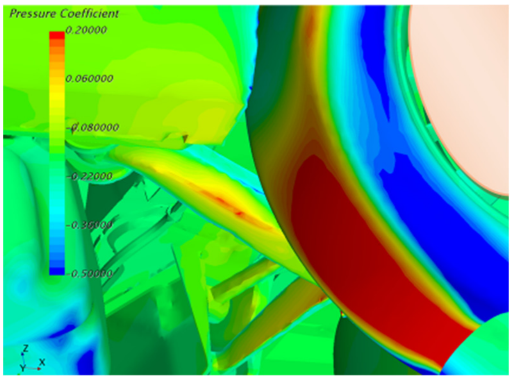


Figure 8. Simulation analysis results without adding rear wheel choke plate

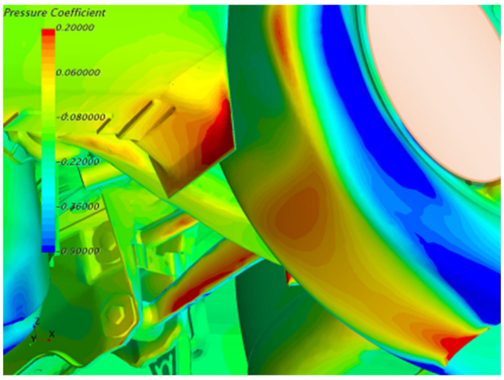


Figure 9. Simulation analysis results of adding rear wheel choke plate

After adding the rear wheel wind deflector, the pressure difference of the front and side of the rear wheel is reduced, and the vehicle drag coefficient is reduced significantly. Therefore, the rear wheel spoiler is installed in the wind tunnel test model. With the help of previous engineering experience, the preliminary front wheel choke size is 155mm*45mm, and the optimization scheme is simulated and analyzed, and the simulation reduces 5count.

The wind tunnel test results show that the effective wind resistance coefficient decreases by 4count, and the simulation results are consistent with the test results.

4.2. Vehicle Cabin Optimization

The air flow inside the vehicle cabin has a great impact on the vehicle wind resistance and engine cabin cooling. While optimizing the external shape and bottom of the vehicle, it is also necessary to systematically optimize the engine cabin air flow. The main principle of cabin optimization is to make the greatest efforts to reduce the frontal air flow in the engine room on the premise of fully meeting the heat dissipation conditions of the internal parts of the cabin, so as to avoid the impact of the frontal air flow on the internal parts of the cabin as far as possible, so as to achieve the purpose of reducing the wind resistance coefficient.

Heat sink upper seal plate,Through the analysis of the heat sink upper seal plate of the Base version, it can be seen that it fails to play a good role in sealing and guiding flow (as shown in Figure 10), resulting in air flow from the back of the heat sink to the front of the heat sink, which is not conducive to drag reduction. By extending the upper sealing plate of the radiator to the bumper, the phenomenon of air flow from the back of the radiator to the front of the radiator is effectively blocked (see Figure 11), and the wind resistance coefficient of the simulation result is reduced by 4count.

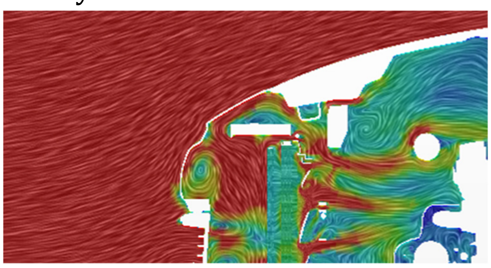


Figure 10. Base result of the sealing plate on the radiator

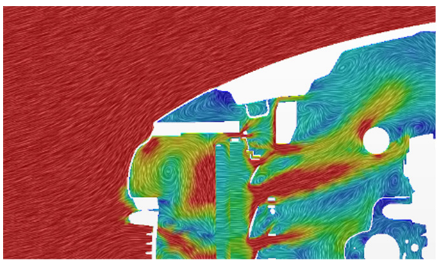


Figure 11. Simulation results of sealing plate on extension radiator

Seal plate at the bottom of the radiator: same as the above scheme, through the simulation analysis results of the whole vehicle, the air flow back in the cabin is formed, resulting in the air flow not passing through the heat dissipation core, thus increasing the resistance coefficient of the whole vehicle. Therefore, the optimization scheme is to add the intercooler seal plate scheme, and the simulation results are shown in Figure 12. When the bottom seal plate of the radiator is added, the resistance coefficient of the whole vehicle is reduced by 1count.

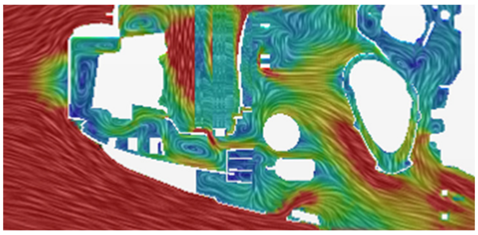


Figure 12. Simulation results of adding the bottom seal plate of the radiator

Figure 13. Wind tunnel test to add radiator bottom seal plate

The verification model of wind tunnel test is shown in Figure 13. The wind tunnel test results show that the wind resistance coefficient decreases by 1count, while the radiator ventilation can be increased by 9.6%. This scheme can be considered for implementation.

After optimizing the exterior shape of the vehicle, optimizing the details of the exterior structure and optimizing the layout system of the components at the bottom of the vehicle, the engine room layout was optimized. Through the sealing plate of the front cooling module, the right sealing plate of the radiator, the sealing plate of the intercooler and the intake grille of the blocked part, the wind resistance coefficient of the vehicle was effectively reduced. At the same time, it also effectively increased the intake volume of the engine compartment (which also played a positive role in the heat dissipation of the cabin).

After the superposition of all effective drag reduction schemes is completed, the CFD simulation analysis result is reduced from 0.402 to 0.358, and the wind tunnel test result shows that the vehicle wind resistance coefficient is reduced from 0.405 to 0.368. The wind tunnel test and simulation errors meet the requirements and basically reach the target value of the vehicle wind resistance coefficient.

5. Conclusion

By establishing the basic air flow calculation model of a medium-sized SUV, this paper conducts CFD simulation analysis on its aerodynamic characteristics related to the outflow field. Meanwhile, the analysis results obtained from the simulation calculation (pressure and velocity cloud map, etc.) are fully studied. In order to verify the effectiveness of the simulation, this paper adopts 40% full detail scale model for wind tunnel test. The results are verified. Then a series of optimization simulation analysis is carried out on the basis of the basic model, and the wind resistance coefficient is effectively reduced. In the course of the research, the conclusions are as follows:

1. Based on the basic theories and concepts of automotive aerodynamics, the main influencing factors of automotive aerodynamics are analyzed to provide theoretical basis for CFD simulation analysis and optimization of target models.

2. Combined with the existing vehicle wind resistance optimization process, CFD simulation research was carried out on the vehicle digital model of an SUV model, the existing modeling process was optimized, and the initial wind resistance coefficient of the target vehicle was obtained through simulation calculation, and the simulation analysis results were simply

ISSN: 2688-8645

analyzed. The production of wind tunnel laboratory and wind tunnel model is briefly introduced. The wind tunnel test of 40% full detail scale model is carried out, and the wind resistance coefficient obtained is compared with the CFD simulation coefficient, which meets the deviation requirements, proving that the CFD simulation model built in the early stage is real and effective, can be used to optimize the analysis work, and can effectively save project costs and shorten the development cycle.

3. The vehicle was optimized based on aerodynamic theory and numerical simulation results. Through various optimization analyses on vehicle modeling, local modeling details, vehicle bottom layout and engine compartment layout, the measured wind resistance coefficient of the vehicle model was reduced by 35count, from 0.404 to 0.369. Thus, to achieve the purpose of effectively reducing the vehicle wind resistance, improve the vehicle aerodynamic characteristics.

References

[1] Simone Sebben. Numerical simulations of a car underbody: effect of front-wheel deflectors. SAE paper 2004-01-1307.

[2] Vitali. D., Casella, M. and Bergamini, P., CFD Prediction of the Effect on Aerodynamic Drag of Vehicle Underbody, ECCOMAS 96 Proceedings, John Wiley & Sons, UK, 1996.

[3] Skea, A.F., Bullen, P.R. and Qiao, J., Review of underbody Aerodynamics Test Techniques, Airflow Characteristics, CFD Contribution, Ford Technical Journal 1998.

[4] Thomas Hallqvist. The Cooling Airflow of Heavy Trucks--a Parametric Study[J]. SAE Technical Paper 2008-01-1171.

[5] Timothy P. Nobel & Sunil K. Jain. Improving truck underhood thermal management through CFD[J]. SAE Technical Paper 2002-01-1027.

[6] Adrian Tentner, Paul Froehle & Chung-Yi Wang, Modeling and analysis of transient vehicle underhood thermo-hydrodynanic events using computational fluid dynamics and high-performance computing[J]. SAE Technical Paper 2004-01-1-511.

[7] Taylor, D. O. and Chu, a. c., Wind Tunnel Investigation of the Effect of Installation Parameters on Truck Cooling System Performance[J], Society Automotive Engineers, SAE Technical Paper 760832.