Research on automatic aerodynamic optimization for a SUV based on RBF model

L L Wu1,*, Y Fu1, X B Bu1, X R Li1 and X L Ma1

1CAE Performance Division, Vehicle Crash Testing and Research Department, CATARC Automotive Test Center (Tianjin) Co., Ltd, Tianjin, 300300, China
*wulinlin@catarc.ac.cn

Abstract. Considering the continuous shortening of automobile R & D cycle, mathematical methods have been more used in automotive aerodynamic optimization design for their capability to fit the law and calculate the optimal results. In this paper, the deformation software, fluid analysis software and optimization software are connected to build an automatic aerodynamics optimization platform of automotive shape. Based on the RBF model, the automatic aerodynamic optimization for a SUV is completed with the front windshield angle, rear windshield angle, engine cover angle, approach angle and departure angle as design parameters and the drag coefficient as design objective. The drag coefficient of the optimal model is 9.40% lower than that of the initial model, and remarkable drag reduction effect is achieved.

1. Introduction

The resistance of the automotive can be roughly divided into two parts: rolling resistance and aerodynamic resistance. When the driving speed is 60km/h, the aerodynamic resistance is equivalent to the rolling resistance. With the increase of the driving speed, the proportion of the aerodynamic resistance keeps rising. If the driving speed is over 150km/h, the aerodynamic resistance is 2~3 times of the rolling resistance [1]. Note that shape resistance accounts for 60% of aerodynamic resistance [2]. Therefore, the aerodynamic resistance can be reduced through the optimization design for automotive shape, thus to improve the fuel economy of traditional automotive and the range of new energy automotive.

In the traditional optimization process of the automotive shape, aerodynamic engineers put forward optimization schemes based on the results of simulation, test and their engineering experience. Usually, it needs to go through a lot of blind attempts, consuming more energy and time, which can’t get satisfied results [3]. For this reason, mathematical methods are much more used in the automotive aerodynamics optimization design owing to their capability to fit the laws and calculate the optimal results [4].

In this study, an automatic aerodynamics optimization design platform for automotive shape is constructed. Based on the RBF model, the automatic aerodynamic optimization for a SUV is completed, which can effectively reduce the drag coefficient and shorten the optimization time, as well as meet the needs of automotive development.

2. Automatic optimization design process

The automatic aerodynamic optimization design platform is composed of deformation software, fluid analysis software and optimization software. The automation of parametric deformation and CFD
simulation are accomplished by writing script files. The automatic optimization design process is shown in Figure 1.

![Figure 1. Automatic optimize design process](image)

3. CFD simulation and analysis of the initial model

3.1. Governing equation

The governing equations of computational fluid dynamics include mass conservation equation, momentum conservation equation and energy conservation equation. In the simulation of the drag coefficient of automobile, the influence of heat transfer on flow field is usually ignored, and only the mass conservation equation and momentum conservation equation are solved [5].

(1) Mass conservation equation

\[ \text{div}(\rho) = 0 \]  

(2) Momentum conservation equation

\[ \frac{\partial u}{\partial t} + \text{div}(uu) = -\frac{1}{\rho} \frac{\partial p}{\partial x} + \text{div}(\text{grad}u) \]  

\[ \frac{\partial v}{\partial t} + \text{div}(vu) = -\frac{1}{\rho} \frac{\partial p}{\partial x} + \text{div}(\text{grad}v) \]  

\[ \frac{\partial w}{\partial t} + \text{div}(wu) = -\frac{1}{\rho} \frac{\partial p}{\partial x} + \text{div}(\text{grad}w) \]

here, \( \rho \) is the density, \( t \) is the time, \( u, v, w \) are the three components of the velocity vector, \( p \) is the static pressure of the flow field in this area.

3.2. Meshing model

In this study, the half model of a SUV is taken as the research object. In the early design stage, simple plane is used to replace the complex structure of chassis, and the grille only retains its shape. The calculation domain is set as 10 times of the model length, 5 times of the model width and 5 times of the model height. The boundary layer mesh is set on the body surface, and the mesh encryption is set in the separation region, as shown in Figure 2.

![Figure 2. Mesh model](image)
3.3. Physical model and boundary condition
Assuming that the problem is a three-dimensional, steady-state and constant density problem. The separation algorithm and realizable \( k-\varepsilon \) turbulence model are selected. The settings of boundary conditions are illustrated in Table 1.

| Position | Type of boundary | Type of wall | Value       |
|----------|------------------|--------------|-------------|
| Inlet    | Velocity Inlet   | /            | 120 km/h    |
| Outlet   | Pressure Outlet  | /            | 0 Pa        |
| Slip     | Wall             | Slip         | /           |
| No-slip  | Wall             | No-slip      | 120 km/h    |
| Side     | Wall             | Slip         | /           |
| Sym      | Symmetry Plane   | /            | /           |
| Body     | Wall             | No-slip      | /           |

3.4. CFD simulation results of initial model
After 5000 iterations of CFD simulation, the drag coefficient of the initial model is 0.3342. The cloud diagram of pressure on symmetry plane is shown in Figure 3.

![Figure 3. The cloud diagrams of pressure on symmetry plane](image)

According to the simulation results of initial model, the front windshield angle, rear windshield angle, engine cover angle, approach angle and departure angle are selected as design parameters, and their initial value and variation range are shown in Table 2.

| Parameter                  | initial value | Variation range  |
|----------------------------|---------------|------------------|
| approach angle             | 15°           | 10°~ 20°         |
| engine cover angle         | 6°            | 3°~ 12°          |
| front windshield angle     | 55°           | 50°~ 60°         |
| rear windshield angle      | 42°           | 32°~ 52°         |
| departure angle            | 15°           | 10°~ 20°         |

4. Analysis of aerodynamic optimization results

4.1. DOE
There are many kinds of DOE methods. Choosing the right DOE method can effectively improve the fitting accuracy of surrogate model. Since optimal Latin hypercube design (Opt LHD) optimizes the non-uniform phenomenon caused by the random, the generated sample points are evenly distributed in the whole space [6]. The overall performance of the surrogate model constructed by Opt LHD is acceptable [7]. In practical engineering, it is usually regarded as the best choice. Therefore, Opt LHD is used to select 50 sample points in this paper.

4.2. RBF model and its accuracy analysis
The Radial Basis Function (RBF) model consists of input layer, output layer and hidden layer. It has the ability of fitting complex nonlinear functions with high accuracy, and does not need to determine the
relevant information of approximate functions in advance. The hidden layer is like a black box to establish the mapping relationship between the input layer and the output layer [8]. Generally speaking, under the same design conditions, the fitting accuracy of RBF model is better than Response Surface Methodology (RSM) model, and the model establishment is faster and easier than Kriging model [9]. So in this paper, RBF model is used to build the relationship between design parameters and design objective.

In the optimization process of practical engineering, the value of $R^2$ is often used to evaluate the accuracy of surrogate model, and its range is $0 \sim 1$. It is generally considered that when the value of $R^2$ is greater than 0.9, surrogate model has high reliability [10]. Based on the DOE matrix after CFD simulation, $R^2$ of the surrogate model built by RBF model is 0.91, which means the accuracy of the surrogate model can meet the requirement.

4.3. Analysis of the optimization results

NLPQLP algorithm is used to optimize the surrogate model. The prediction value of the drag coefficient of the optimal model is 0.3012, while its CFD simulation value is 0.3028. And the error between them is only 0.53%, illustrating that the optimization result is reliable. The parameters and drag coefficients of the optimal model and the initial model are shown in Table 3. The drag coefficient of the optimal model is 9.40% lower than that of the initial model.

Table 3. The parameters and drag coefficients of the initial model and the optimal model

| Model    | approach angle | engine cover angle | front Windshield angle | rear windshield angle | departure angle | drag coefficient |
|----------|----------------|--------------------|------------------------|-----------------------|----------------|------------------|
| Initial  | 15°            | 6°                 | 55°                    | 42°                   | 15°            | 0.3342           |
| Optimal  | 15°            | 12°                | 60°                    | 43°                   | 15°            | 0.3028           |

It can be seen from the cloud diagrams of velocity vector on symmetry plane in Figure 4 that the velocity of the initial model is higher than that of the optimal model at the front of the engine cover and the roof, and the separation phenomenon is more obvious. In the meanwhile, at the junction of the front windshield and the engine cover, the velocity of the optimal model is faster than that of the initial model, which indicates that the air through this area is more smoothly. Compared with the optimal model, the eddy current in the rear of the initial model is larger, which will consume more energy and make the pressure lower.

Figure 4. The cloud diagrams of velocity vector on symmetry plane

The cloud diagrams of pressure on body surface are shown in Figure 5, from which it is clear that after optimization, the positive pressure area of the front and front windshield of the optimal model is smaller than the initial model due to the increase of engine cover angle and front windshield angle. In the negative pressure area of the rear of the body, the pressure value of the optimal model is larger than that of the initial model. Therefore, the pressure difference resistance of the optimal model is less than that of the initial model, so that the drag coefficient of the optimal model is lower.
From the cloud diagrams of turbulent kinetic energy on symmetry plane in Figure 6, it can be seen that there is a high turbulent kinetic energy area at the tail of the initial model, which indicates that the air flow in this area is unstable and there is a strong pressure fluctuation. Compared with the initial model, the intensity of turbulent kinetic energy at the tail of the optimal model is obviously reduced, so that the energy consumed is less and the aerodynamic drag is smaller.

5. Conclusion
Based on the RBF model, the automatic aerodynamic optimization of an SUV is completed with the automatic aerodynamics optimization design platform of automotive shape in this work. From the results obtained, specific conclusions can be summarized, including:

(1) By comparing and analysing the velocity, pressure and turbulent kinetic energy between the optimal model and the initial model, it is found that adjusting the engine cover angle and the front windshield angle can effectively decrease its drag coefficient.

(2) Based on the surrogate model constructed by RBF model, the error between the optimal prediction value and the CFD simulation value is only 0.53%. The error is satisfying, demonstrating that RBF model can be used for the optimization of automotive shape.

(3) The drag coefficient of the optimal model is 0.3028, which is 9.40% lower than that of the initial model, and good drag reduction effect is achieved.
(4) Since there are many parameters involved in the optimization of automotive shape, and there is a strong correlation among them. Therefore, whether the optimization method proposed in this paper is applicable to other models remains to be further verified.

Acknowledgement
This research was financially supported by the Research on common basic technology of CATARC Automotive Test Center (Tianjin) Co., Ltd, (NO.TJKY1920011).

References
[1] Hu Y 2017 Computer Simulation 34 127-131.
[2] Tian Y Y, Hong H H 2015 Tractor & Farm Transporter 42 40-43.
[3] Huang R S, Wang Y, Cui S H, Liu X L 2019 Automobile Applied Technology 3 101-104.
[4] Zhang H X, Qiu Y F, Zhang L B, Fu J 2014 Computer Aided Engineering 23 6-11.
[5] Han Z H, Wang J, Lan X P 2005 An Example and Application of Fluent in Fluid Engineering Simulation (Beijing: Beijing University of Technology Press) p 76-78.
[6] Chen D J, Xiong S F 2017 Journal of Systems Science and Mathematical Sciences 37 53-65.
[7] Ye P C, Guang P, Shan G 2019 Journal of Northwestern Polytechnical University 37 714-723.
[8] Wang Y P, Guo C Q, Wang T, Li S, Yao Y L 2018 Transactions of Beijing Institute of Technology 38 221-228.
[9] Gao D W, Zhang N, Chen H F 2018 Journal of Highway and Transportation Research and Development 35 128-136.
[10] Ming Z, Guo Q T 2015 Machinery & Electronics 2 7-11.