Research on Underbody Flattening of a Passenger Vehicle towards Aerodynamics Optimization

HE Xiaona¹,², LI Tao³, XI Chunfu², and Zhang Honghui¹

¹ State Key Lab. of Vehicle NVH and Safety Technology, Chongqing 401122, P. R. China
² China Automotive Engineering Research Institute Co., Ltd., Chongqing 401122, P. R. China
³ Beijing New Energy Automobile Co., Ltd., Beijing 102606, P. R. China
E-mail: hexiaona@caeri.com.cn

Abstract. A CFD method for the full-scale external flow of a passenger vehicle is established. Based on the comparison between the simulation and wind tunnel test results, the reliability to use this model and method for the aerodynamic drag coefficient (Cd) and lift coefficient (Cl) prediction of the vehicle is verified. The trend of Cd and Cl for five models with different underbody flattening degree shows the significance of underbody flattening on the aerodynamic performance, with nearly 18% on the drag and over 30% on the lift coefficient optimization.

1. Introduction
CFD is an effective and widely used method for the automotive aerodynamics, and has achieved the transition from the theoretical research into the practical engineering [1]. Using commercial software STAR-CCM+, full-scale models and the external flow field of a passenger vehicle was established in the literatures [2-4] to verify the effectiveness of CFD method by the contrast between the calculation and wind tunnel test, and based on this, the drag coefficient optimization has been carried out by underbody flattening of the passenger vehicle.

2. Calculation method
2.1. Geometric Model
Due to the complex shape and structure of the vehicle, the body surface of the vehicle model and the interior of the engine compartment are properly simplified. The model should reflect the actual state as much as possible, especially the reaction on the aerodynamic performance of the vehicle when the underbody changed. Therefore, the model includes not only the integrity of the upper body geometry, but also completely retains the engine compartment and underbody components, which is shown in Figure 1.

Figure 1. CFD calculation model of the vehicle body
The accuracy of CFD is closely related to the size of the calculation grid, but it is necessary to refine the grid of the body details in combination with hardware limitations. The main parameters of the calculation grid are shown in Table 1.

| Type               | Description                                                                                                                                 |
|--------------------|---------------------------------------------------------------------------------------------------------------------------------------------|
| Surface unit       | The basic size of the surface unit is 2-8mm, the size of encrypted parts (such as rear view mirror, rear wing, etc.) are 1-4mm, the size of the engine compartment interior parts are 4-16mm and the air intake grille size of the Engine compartment is 1-2mm. |
| Body unit          | The minimum size of body unit encryption area is 16mm and the total grid number is up to 35 million.                                           |
| Boundary layer unit| Body outer surface and accessories: 10 layers in total, the first layer thickness is 0.05mm and the total thickness is 6mm.                   |

In the process of calculation, an area surrounding the vehicle is established to simulate the wind tunnel to monitor the state of the flow field, which is called the fluid calculation domain\(^5\). The calculation domain size is set to 66*12*10m, the front end of the passenger vehicle is 20m from the front end of the calculation domain, and the number of the body unit grid is about 35 million. The Realizable K-E turbulence model is used to set the boundary conditions, the Y+ value of the vehicle body surface is approximately 4-6. The front of the vehicle is set as the calculation domain entrance, the inlet turbulence ratio is 1%, and the speed is 33.33 m/s. The calculation domain outlet is the pressure outlet boundary, the ground is the non-slip wall boundary and the other calculation domain wall is the sliding wall boundary. The condenser and radiator use adopt porous media models and cooling fan is set not to rotate.

2.2. Calculation results
The pressure at each point in the calculation domain can be calculated, and the space corresponding to which called the pressure field. In order to describe the airflow visually, different colors which represent the pressure value are used to reflect the flow field pressure distribution. Figure 2 shows the pressure coefficient of vehicle body surface (Cp) and equipotential surface of the total pressure equaling to zero.
The airflow is blocked when it encounters the front of the vehicle so that the speed of the airflow is greatly reduced and a positive pressure zone is produced, then the airflow flows in two directions. A part airflow flows to the top of the vehicle and when passing through the front windshield, the airflow once again is blocked and weakened to form a positive pressure zone. When reaching the highest point of the vehicle body, the larger airflow speed turns sharply to form a negative pressure zone. When arrived at the rear trunk lid, a sudden change in direction of airflow causes the airflow to separate and a small negative pressure zone is formed. Then the airflow accelerates tangentially along the upper surface of the trunk lid. The other part airflow flows from the bottom of the vehicle to the rear. When passing through the tire, a negative pressure zone is formed due to the effect of tires. When entering the bottom of the vehicle, a negative pressure zone is formed due to the narrow aisle and the fast speed of airflow.

The two parts of the airflow merge at the rear of the vehicle to form a vortex resulting in energy loss, which is one of important influence factors for aerodynamic drag. The calculation result predicts the flow field of the vehicle correctly and reflects the local airflow separation of the wheel cavity and the rearview mirror, etc.

2.3. Model reliability analysis and evaluation

In order to further verify the credibility of the geometric model and calculation method, a wind tunnel test is carried out in the aero-acoustic wind tunnel in Shanghai Ground Vehicle Wind Tunnel Center (shown in Figure 3).

The wind tunnel test is performed under the same conditions as CFD and the wind speed is set to 120km/h. By comparing the calculation result with the test result in three states (shown in Table 2), it is found that the test result is in good agreement with the CFD results, the relative deviation of the drag coefficient (Cd) is within 3%, and the CFD result is lower than the test result. The relative deviation of the lift coefficient (Cl) is slightly larger, but the maximum relative deviation is also about 5%. The overall evaluation shows that the calculation model and method are credible, and the model can be used to research the flattening underbody.

| No. | Model for CFD and wind tunnel test | Cd  | Cl  | Relative deviation | | Relative deviation |
|-----|-----------------------------------|-----|-----|-------------------|------------------|
| 1#  | Basic model (consistent with model 5 below) | 0.315 | 0.323 | 2.5% | 0.161 | 0.166 | 3.0% |
2# Plugging cooling grille based on 1#
   0.295  0.299  1.3%  0.107  0.111  3.6%
3# Removing the rearview mirror based on 2#
   0.282  0.285  1.1%  0.159  0.151  -5.3%

3. Flattening underbody optimization

3.1. Model establishment
The unevenness of underbody hinders the airflow at the bottom of the vehicle. The underbody shape is more complex and the aerodynamic resistance is the greater, which will increase sharply with the increase of the vehicle speed. Thus it is useful to reduce the vehicle aerodynamic resistance, induced resistance and frictional resistance by optimizing the flattening underbody [6]. The underbody parts studied in this paper is shown in Figure 4.

Five full-scale external flow field calculation models were established (Figure 5). The specific differences between models are: Model 1: Basic model. Model 2: Add four front bottom boards on Model 1. Front bottom boards is below the cooling module, engine compartment and the chassis central channel. Model 3: Add bottom boards below both sides of the central channel of the chassis, both sides of the tank and the rear suspension based on Model 2. Model 4: Add deflector before the front wheel based on Model 3. Model 5: Add deflector before the rear wheel based on Model 4.

3.2. The cloud map of pressure coefficient
The cloud map of pressure coefficient for key parts of five models are obtained by calculations, as shown in Figure 6 and Figure 7.

![Figure 6](image1.png)

**Figure 6.** The cloud map of pressure coefficient of the bottom of Model 1, Model 2 and Model 4 (from left to right)

The left picture of Figure 6 shows that: owing to the underbody of the model 1 not optimized, the pressure distribution below the engine and the cooling module are more complicated and not uniform. Air pressure increasing at the bottom of the vehicle when airflow passes through both sides of the central channel of the chassis, both sides of the tank and the rear suspension area. The pressure of the front and rear wheels is large which causes energy consumption of wheels. Contrasting the three pictures in Figure 6 shows that the pressure distribution of model 2 and model 4 becomes even and the pressure decreases by adding bottom boards below the engine, chassis center, tank, rear suspension, etc.

![Figure 7](image2.png)

**Figure 7.** The cloud map of pressure coefficient of front and rear wheels with and without deflectors installed before (from left to right)

The Figure 7 shows that the airflow pressure and the turbulence before front and rear wheels of model 4 and model 5 are reduced by adding deflectors, then reaching to reduce the resistance to wheels.

3.3. Contrast to Cd and Cl

The calculation results of the Cd and Cl of five models are shown in Table 3, which shows that the aerodynamic drag and lift of the vehicle have been variously varied by series of flattening optimization of underbody. The optimal optimization has an effect of 18% on drag and 31% on lift.

**Table 3.** The results of calculating the Cd and Cl of five models

| Model | Cd     | Relative value of Cd | Cl     | Relative value of Cl |
|-------|--------|----------------------|--------|----------------------|
| 1     | 0.387  | 1.000                | 0.199  | 1.000                |
| 2     | 0.350  | 0.904                | 0.149  | 0.749                |
| 3     | 0.335  | 0.868                | 0.136  | 0.683                |
| 4     | 0.317  | 0.819                | 0.168  | 0.844                |
| 5     | 0.315  | 0.814                | 0.161  | 0.809                |

Analyzing the relative values of Cd and Cl in Figure 8 shows that the effect on the aerodynamics of the vehicle of different components. Both bottom boards and deflectors before wheels have the effect of reducing the aerodynamic drag, and the most obvious effect comes from the rear four-piece boards.
and the deflector before front wheels. All bottom boards are beneficial to reduce aerodynamic lift, which is mainly due to that the flattening underbody lets the airflow accelerated and the pressure reduced at the bottom of the body. It is noteworthy that the deflector before the front wheel increases the aerodynamic lift of the vehicle greatly, but the deflector before rear wheels slightly reduces the aerodynamic lift of the vehicle.

![Figure 8. Relative values of Cd and Cl](image)

4. Conclusions
Based on the effectiveness of CFD on the aerodynamics calculation, the drag and lift coefficients \( C_d \) and \( C_l \) of the vehicle were predicted with error not more than 5%. With underbody flattening, the drag coefficient \( C_d \) was reduced 18% and the lift coefficient \( C_l \) was reduced 31% maximum in five models with different flattening degree, which shows the significance on the aerodynamic performance of the vehicle. It is noteworthy that the deflector before wheels will reduce the aerodynamic drag of the vehicle, but which has an adverse effect on the lift.

5. References
[1] GU Zhengqi. Automobile Aerodynamics, M. People's Communication Press, 2005.
[2] Sun S, Chang Y, Fu Q, et al. Aerodynamic Shape Optimization of an SUV in early Development Stage using a Response Surface Method, J. SAE Technical Paper 2014-01-2445.
[3] Thomas Blacha, Moni Islam. The Aerodynamic Development of the new Audi Q5, J. SAE Int. J. Passeng. Cars-Mech.syst.10(2):2017, doi: 10.4271/2017-01-1522.
[4] ZHANG Yangjun, LV Zhenhua, XU Shian, et al. Advances in Numerical Simulation of Automobile Aerodynamics, J. Automotive Engineering, 2001, 23(2), pp.82-91.
[5] ZHANG Jianli, ZHANG Guozhong, Application of CFD in Automotive Aerodynamics Research, J. Journal of Shenyang University (Natural Science), 2006, 18(4), pp.15-18.
[6] Andreas Kremheller. The Aerodynamics Development of the New Nissan Qashqai, J. SAE Technical Paper. 2014.

6. Acknowledgments
This work is supported by the Key Special Technology Innovation Fund of Chongqing Science and Technology Commission (cstc2015zdcy-ztxx60011) and the Opening Fund of State Key Laboratory of Vehicle NVH and Safety Technology (NVHSKL-201402).