Numerical Simulation of the Influence of Intake Grille Shape on the Aerodynamic Performance of a Passenger Car

Longwei Liu[1,a], Chang Jing[1,b], Xingjun Hu[1,c](Corresponding Author)

[1]State Key Laboratory of Automotive Simulation and Control of Jilin University, Changchun,130022, Jilin Province, China

[a]18844198211@163.com, [b]changjingnihao123@126.com, [c]hxj@jlu.edu.cn

Keywords: Intake grille; Grille shape; Aerodynamics; Numerical simulation

Abstract. The influence of the front intake grille shape on the aerodynamics of a passenger car is studied,and a numerical simulation is made on a full-scale CAD automobile model with different grille shapes, including straight, convex, concave, and M and W-shaped. The results show that the drag coefficient Cd and the radiator mass flow rate Q have a rough correlation, and the straight grille has the best effect, with smaller Cd by 0.05% and larger Q by 0.19%. Although it is numerically insignificant, the straight grille uses the least amount of materials as well, thus it is considered the best.

Introduction

The front intake grill of an automobile is usually made of metal, which has three main roles. Firstly, it affects the cooling of components inside the engine compartment, such as radiator, engine, and air conditioner. Secondly, it protects engine compartment from damage by foreign objects. Thirdly, it has aesthetic function. Different cars have grilles with different sizes, materials and mounting positions. Being the only intake of the engine compartment apart from the bottom intake, it is especially important. It can control both intake flow rate and flow details. We know from aerodynamics that the shape, position, orientation, size and density are important for intake quality and the inner flow field[1-3].

Because the components inside the engine compartment is complexly shaped and tightly packed, and the computational resources is limited, the numerical simulations and the wind tunnel experiment of the engine compartment was not much studied until the late 1990s, when CFD and the turbulence theory became more mature[4-5]. The numerical simulation is fast, widely applicable, not limited by Reynolds number and boundary conditions, and thus widely used in automobile aerodynamics[6].

In this paper, numerical simulation is used to study how different intake grilles affect aerodynamic performance. This research will support the future development of low-drag automobiles.

Geometric Model

A full-scale passenger car CAD model with engine compartment is studied. Because there are too many components in the engine compartment and the computational resources are limited, we only consider the components that affect air flow or cooling, such as the radiator, or the fan. The simplified model is shown in Fig.1, and the simplified engine compartment is shown in Fig.2.

The original grille is convex, as shown in Fig. 3. Four different grilles were designed: straight, concave, M-shaped and W-shaped, as shown in Fig. 4.

Numerical Simulation

Computation Domain. A rectangular domain is chosen to simulate a wind tunnel. To ensure that the flow is well developed, we define the domain as shown in Fig.5, where L is the length of the car, W is the width, H is the height. The domain's length is 6L, width is 9W, and height is 5.5H. The car is centered side-to-side. Blockage ratio is 2.0%, within the acceptable ranges[7-8].

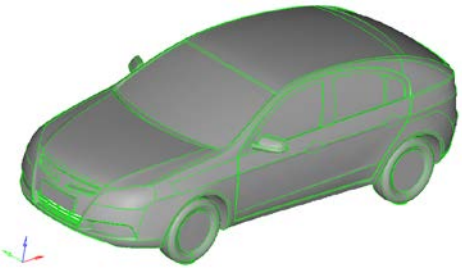


Fig.1 Simplified CAD model of the whole car

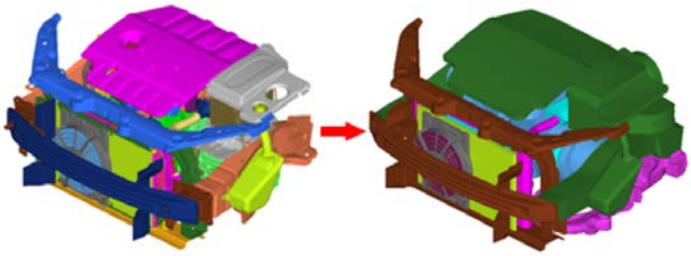


Fig.2 CAD model of the engine compartment before/after simplification

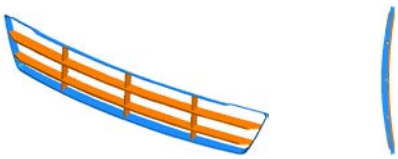
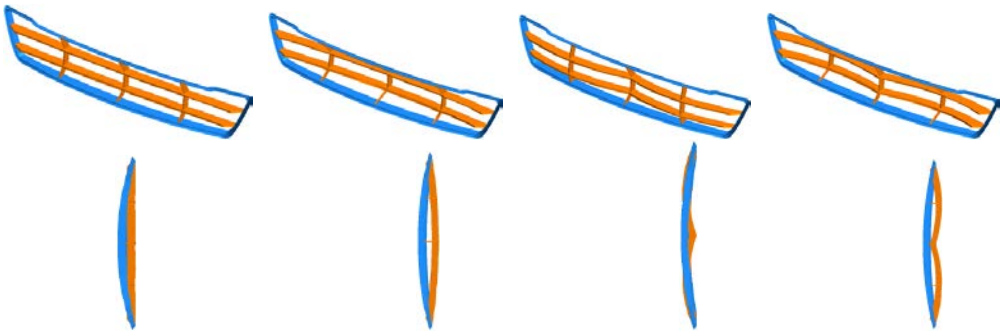


Fig.3 Original (convex)



（a）straight （b）concave （c）M-shaped （d）W-shaped

Fig.4 Four different grille shapes

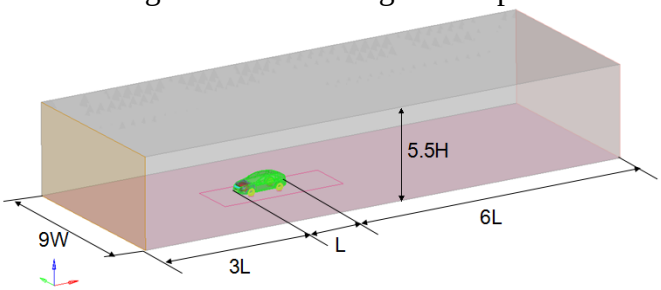


Fig.5 Computation domain

Mesh Generation. Hypermesh is used to compute triangular surface mesh, with different mesh size for different parts. In order to reduce mesh size, we use different types of elements in the volume mesh, using prisms, polyhedral, and cut-cell mesh. The main part is meshed by using a cut-cell mesher, while the porous volumes, such as the condenser and the radiator, is meshed by using polyhedrals. In order to increase the accuracy of the simulation while not having excess elements, we increased the density in key areas. The total mesh size is about 15 million volume elements. A cut plane at Y=0m is shown in Fig.6. The red rectangle is the engine compartment, zoomed in in Fig.7.

Boundary Conditions. The boundary conditions can be divided into the internal and the

external. The boundary conditions of the external flow field is as follows: inlet is a velocity inlet, and velocity is 27.78m/s ,that is perpendicular to the surface, and the temperature is 25`C. Outlet is a pressure outlet under atmospheric pressure, and the temperature is 25`C. The ground is a moving wall, with the same velocity as the inlet, adiabatic. The side and top is symmetric, adiabatic. The car surface is an adiabatic wall.

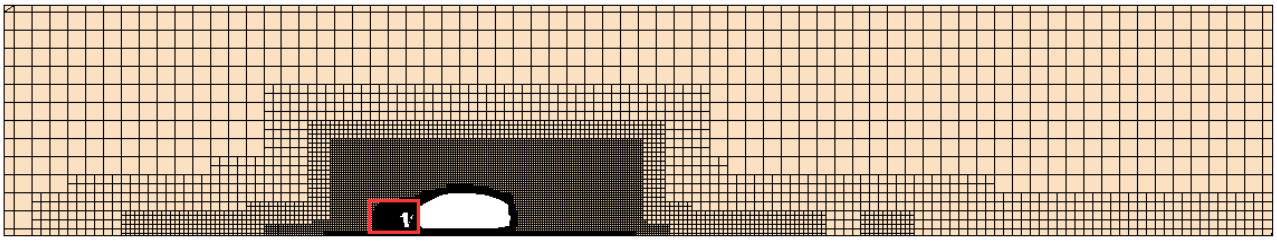


Fig.6 Cut plane at Y=0m

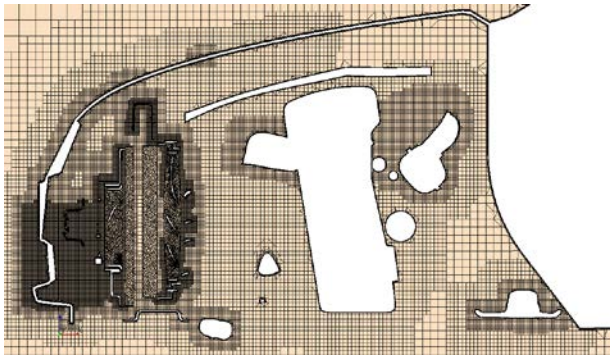


Fig.7 Cut plane of the engine compartment at Y=0m

The boundary conditions of the internal flow field is as follows: the cooling modules(condenser, radiator) is simplified as a porous area with certain damping on its inlet and outlet. The cooling fan is rotating. A local rotational coordinate frame is set by using multiple reference frames(MRF), with interfaces on its inlet, outlet and side surfaces. The engine block and connectors is set as a heat source.

Turbulence Model. The velocity is 27.78m/s, far lower than 100m/s. It is in the incompressible flow region, and a 3D steady incompressible viscous RANS model is used.

Standard k-e and realizable k-e model is both used in STAR-CCM via a two-layer method. It solves the viscous bottom-layer problem by fine meshing. However the turbulence viscosity equation is changed for the realizable k-e model, resulting in vorticity and curvature. The advantages of satisfying mathematical constraints under normal stress and being in agreement with experimental results. This make the realizable k-e model better than the standard k-e model in many applications, especially for rotational uniform shear flow, pipe flow, boundary layers and separated flows. And the realizable k- ε model can lead to more accurate results which make it more credible. Thus the realizable k-e model is used to simulate the internal flow field inside the engine compartment.

Result and Analysis

Numerical simulations for all five configurations are carried out under the same process. The result is shown in Table 1. The results are evaluated using the drag coefficient Cd and the radiator mass flow Q.

Table 1 Results for different configurations

No.	Grille shape	Cd	Q
1	Convex	0.36551	0.8910
2	Straight	0.36534	0.8927
3	Concave	0.36765	0.8874
4	M-shaped	0.36636	0.8903
5	W-shaped	0.36834	0.8869

The Fig.8 about Cd and Fig.9 about Q for different configurations are plotted by using the data in Table.1.

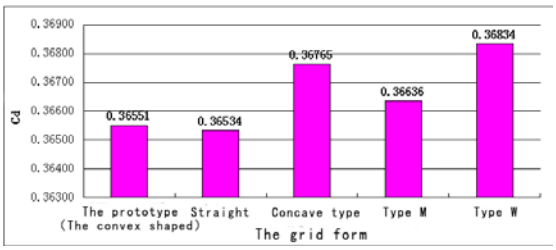


Fig.8 Cd of different grilles

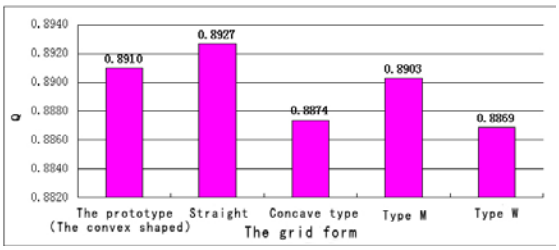


Fig.9 Q of different grilles

As shown in Fig.8, the effects of shape change on the Cd are reflected on the three decimal place. Because a smaller Cd is better, the orders of excellence are followed as such: straight > original(convex) > M-shaped > concave > W-shaped. As shown in Fig.9, the change of Q is on the order of 10^{-2}. Since a bigger Q is better, we rank the grills as such: straight > original(convex) > M-shaped > concave > W-shaped. Generally, they change in the same way. The straight grill has the best effect, with a lower Cd by 0.05% and a more Q value by 0.19%. Although the number may seem insignificant, it also uses the least amount of material, thus it's the best solution. Now how they change using some graphics will be analyzed.

Streamlines. The streamlines for each configuration, from best to worse, are given in Fig.10 (a) to (e). From them we can see that, although the streamline from the lower grille is highly irregular due to the blockage of the components, It's clear that the streamlines increase in turn according to (a)-(e).The streamlines are respectively produced from the grille inlet to the top of first fan, the top of radiator, the top of second fan and the top of engine. That is the streamlines increase that aren't going through cooling modules, and thus leading to the decrease of mass flow Q for radiator.

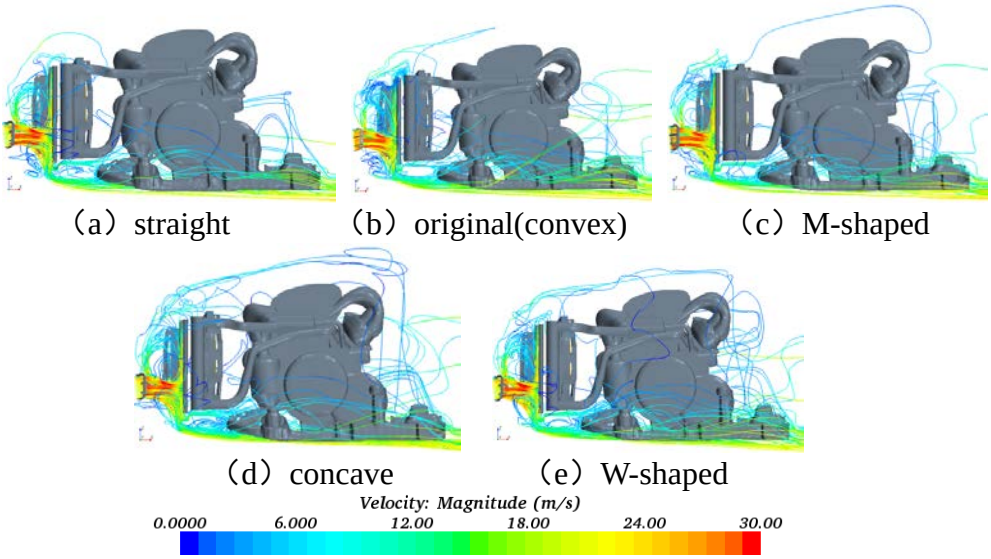


Fig.10 Streamline inside the engine compartment for different configurations

Velocity Distribution. In order to better illustrate the differences between the flow fields of different grille shapes, a plane where they have the greatest differences is chose, which is the plane where the M or W-shaped grill have the greatest curvature, Y=0.14, as shown in Fig.11.

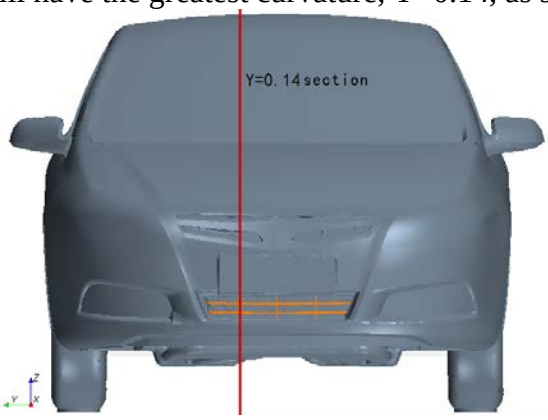
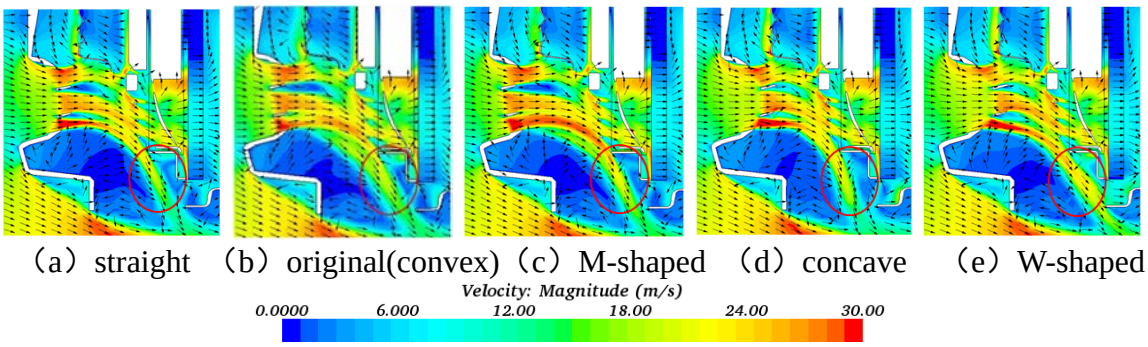


Fig.11 The cut plane Y=0.14

From the best to the worst, vector plots of velocity of each configuration is shown in Fig.12 (a) to (e). The colors illustrate the magnitude of velocity, and the directions of arrow represent the directions of velocity.



（a）straight （b）original(convex) （c）M-shaped （d）concave （e）W-shaped

Fig. 12 Velocity vector plot at lower grille for different configurations

From the plots we can find the air flow illustrated by a red circle that flows out of the engine compartment directly without going through the cooling module. The plots show that the velocity of such flow increases from (a) to (e). Together with the streamline plot, we can deduce that the decrease of Q value is caused by more air flowing directly out of the engine compartment without going through the cooling module.

Conclusions

The results have shown that the drag coefficient Cd and the radiator mass flow rate Q have a rough correlation, and the straight grille has the best effect, with a smaller Cd by 0.05% and a larger Q by 0.19%. Although it is numerically insignificant, the straight grille uses the least amount of materials as well, thus it is considered the best.

As shown in previous figures, grills with outward curvature(convex, M-shaped) are better than those with inward curvature(concave, W-shaped), and grills with a single curvature(convex, concave) is better than those with double curvatures(M,W-shaped).

In this paper we have only studied four different grill styles without a more detailed study of its exact shape, which shall be followed.

Acknowledgment

The authors are grateful for the support provided by the Key Laboratory of Road Construction Technology and Equipment (Chang'an University) of the Ministry of Education.

References

[1] Limin Fu. Automotive Aerodynamics [M].Beijing: Machinery Industry Press, 2006.

[2] Xingjun Hu. Automotive aerodynamics[M]. Beijing: The people's traffic press, 2014.

[3] Wang Dong, Dengyun Pan, Jiyun Yi. Simulation analysis of influence of grille to vehicle front air intake [J]. Computer aided engineering,2013,22(6):18-22.

[4] Kaiyao Hu. The bus engine management research and structure improvement [D].Hunan: Hunan University,2012.

[5] Fujun Wang. Computational fluid dynamics analysis-The principle and application of CFD software[M]. Beijing：Tsinghua University Press, 2014.

[6] Doddegowda P, Bychkovsky A L, George A R. Use of computational fluid dynamics for the design of formula SAE race car aerodynamics[C] .SAE Paper, 2006-01-0807.

[7] Xingjun Hu, Jing Chang. Influence of Engine Cabin Simplification on Aerodynamic Characteristics of Car[C]//Advanced Materials Research. 2014, 1042: 188-193.

[8] Xinyu Wang, Xingjun Hu, Tengfei Li， etc. The effect of a new skirt on the dynamic characteristics of heavy vehicle. [J]. Journal of Jilin University: Engineering Science, 2012(S1):37-40.