Numerical Simulation and Experimental Research on Engine Room Heat Dissipation Performance of a Hybrid Car

WANG Jianpeng¹, YAN Tingsong², ZHANG Zhijun²

(1. BYD Company Limited, Product Planning and New Auto Technologies Research Institute, Shenzhen, 518118, China; 2. School of Mechanical Engineering and Automation, Northeastern University, Shenyang, 110819, China)

987179244@qq.com

Abstract. Compared with the fuel cars, the hybrid cars has higher thermal load of engine room and has higher requirements for heat dissipation performance of the cooling module. Under idling condition, a hybrid car has an actual problem that higher temperature of engine room and inlet air temperature of cooling module. In this paper, a mathematic model of car thermal management is established, numerical simulations on flow and temperature fields of car were performed by using STAR-CCM+ software, the feasibility of the simulation results was verified by the thermal balance test of the cars. The results show that the reflow of hot air and the inlet air temperature of the cooling module are reduced by sealing the cooling module, the front end module and optimizing the structure of deflector, thus, the heat dissipation efficiency of the cooling module is improved.

1. Introduction

Compared to fuel cars, the engine room of hybrid cars need to be equipped with parts such as engine, generator, gearbox, motor and controller at the same time, the heat dissipation condition is worse. Therefore, research on engine room heat dissipation performance of hybrid cars has become the focus for domestic and foreign scholars. CFD approach is applied to all aspects of car design, it can significantly shorten the product development cycle and save R&D costs [1-3]. Kim J M [4] performed a numerical simulation on the shape of the grille opening, the influence of structural parameters such as the vertical height, horizontal width and position of the grille on the air intake flow is analyzed, which provides reference for the improvement of heat dissipation performance of car engine room. Ma W S [5] tested the heat dissipation efficiency of a hybrid car radiator, the influence of different air intake speed and temperature under multiple working conditions on the heat dissipation efficiency of the cooling system is regularized. Vivek Kumar [6] analyzed the flow and temperature field in the engine room and verified the simulation results through test, it provided a theoretical basis for the optimization of the car cooling system. Shi Pengfei [7] used CFD approach to simulate the flow field of engine room and found that the air intake flow of the cooling module could be effectively increased by adding deflectors. Patidar [8] found that the reflow of hot air was an important reason for low heat dissipation efficiency of cooling module and overheating of engine room. Li Zhelong [9] restrained the reflow phenomenon of hot air and improved the cooling efficiency of the condenser by configuring the condenser bossing and adjusting the opening of the grille.

Most of the research is mainly to simulate the air flow and temperature field of traditional fuel cars,
the research on the heat dissipation characteristic of the hybrid car engine room is rare. This paper aiming at the actual problems of high inlet air temperature of the cooling module and high temperature of engine room under idling condition, CFD approach is used to analyze the causes of problems and propose improvements. Moreover, the correctness of the simulation results are verified through the thermal balance test so that making the thermal management performance of the engine room meets the requirements.

2. Establishment of numerical simulation model

2.1 Numerical model

Under idling condition, mach number is less than 0.3, the air density around the car can be approximated as a constant. The air flow field in the engine room is complicated, the standard \( k-\varepsilon \) model is selected. The basic equations are as follow.

Continuity equation:

\[
\frac{\partial}{\partial x_j} (\rho u_j) = 0 \tag{1}
\]

Momentum equation:

\[
\frac{\partial}{\partial x_j} (\rho u_i u_j - \tau_{ij}) = -\frac{\partial p}{\partial x_j} \tag{2}
\]

Energy equation:

\[
\frac{\partial}{\partial x_j} \left( \rho u_i h - k \frac{\partial T}{\partial x_j} \right) = -u_j \frac{\partial p}{\partial x_j} + \tau_{ij} \frac{\partial u_i}{\partial x_j} \tag{3}
\]

Where \( \rho \) is the density, \( T \) is the temperature, \( u_i \) is the velocity component of \( i \) direction on the coordinate axis, \( p \) is the pressure, \( \tau \) is the viscous stress, \( k \) is the heat transfer coefficient, \( h \) is the fluid enthalpy.

2.2 Simulation model

The size of computational domain is consistent with the thermal balance laboratory. The whole car has 35 million grids. The computational domain is shown in Figure 1. The boundary conditions of the simulation model are set according to the idling condition of thermal balance test. The computational domain uses velocity inlet and pressure outlet. The surfaces of exhaust system, engine, gearbox and other use temperature wall boundary condition. The heat protection shields are set as solid area. Each radiator is set as porous medium area. MRF model is applied to the fan. Radiation model uses S2S model.

![Figure 1. Computational domain](image)

3. Analysis of simulation result

Under idling condition, the airflow tends to form partial reflow around the cooling module. The cooling airflow passes through the grille, motor radiator, condenser, low temperature radiator, high temperature radiator and fan and then blows to the surfaces of engine, motor, controller, gearbox and
exhaust pipes. The airflow cools the radiators firstly and then carries out convective heat transfer with the engine room parts to achieve the goal of the heat dissipation.

3.1 Analysis of original model

(1) The maximum spacing between the upper, lower, left and right faces of the cooling module and the front end module is 20mm, 24mm, 23mm and 22mm respectively, as shown in Figure 2. (2) There is a spacing between the radiator groups. (3) The deflector is divided into four pieces on the left and right sides, which is fixed on the radiator water chamber and the front end module. The front of deflector is connected with anti-collision beam and front bumper, there is gap between the deflector and water chamber, anti-collision beam and front bumper. The deflector is shown in Figure 3.

![Figure 2. Cooling module and front end module](image)

![Figure 3. Deflector](image)

Figure 2 shows the air reflow in the Y=0 mm section of engine room. After airflow passes through the radiators and fan, part of airflow passes through the gap between the top and bottom of cooling module and the front end module and flows back to the upper front and lower front of cooling module. Figure 5 shows the air reflow in the Z=260 mm section. Part of the airflow passes through the gap between the cooling module and the front end module, the front end module and front bumper, in the end, it flows back to the front of cooling module.

![Figure 4. Y=0 mm](image)

![Figure 5. Z=260 mm](image)

Monitor the air intake flow and inlet air temperature of the grille and radiators separately, analyze the influence of hot air reflow on cooling effect. The monitoring air intake flow position of grille is 5 mm away from the grille, the monitoring air intake flow position of radiators is 2 mm away from the surface of radiators.

Table 1 shows the hot air reflow rate around the cooling module is 38.47%, which seriously affects the Wind absorption efficiency of the fan. Table 2 shows the inlet air temperature of the grille is 45.17℃, the motor radiator is 50.85℃, which are higher than the corporate standard. After the cooling air which is heated by the radiators enters the engine room, part will reflow to the radiators for reheating again, which obviously worsens the inlet air temperature of cooling module.

| Grille | Motor radiator | Condenser | Low temperature | High temperature |
|--------|----------------|-----------|-----------------|-----------------|

![Table 1. The air intake flow of grille and radiators(kg/s).](image)
| Grille | Motor radiator | Condenser | Low temperature radiator | High temperature radiator |
|--------|----------------|-----------|---------------------------|---------------------------|
| 45.17  | 50.85          | 58.28     | 69.40                     | 71.25                     |

Table 2. The inlet air temperature of grille and radiators (°C).

3.2 Thermal balance test

In order to verify the correctness of the simulation results, it is necessary to test the original car according to the thermal balance test scheme. The collection is stopped after 5 minutes of thermal balance, as shown in Figure 6.

![Figure 6. Thermal balance test](image)

Some test data and simulation results are shown in Table 3. The error between the simulation results and the experimental data is less than 15%. It can be considered that the CFD simulation results are feasible.

| Position    | Grille | Motor radiator | Condenser | Rear bumper | Rear lifting lug |
|-------------|--------|----------------|-----------|-------------|-----------------|
| Simulation  | 45.17  | 50.85          | 58.28     | 46.14       | 93.13           |
| Test value  | 49     | 58             | 65        | 52.6        | 105.4           |
| Error       | 7.82%  | 12.33%         | 10.34%    | 12.28%      | 11.64%          |

Table 3. Part of test data and simulation results (°C).

4. Optimization of heat flow field in engine room

4.1 Optimization analysis

Under idling condition, not only the hot air reflow around the cooling module be considered, but also the air leakage between radiator groups. (1) Seal the gap between the radiator groups with sponge sealing strips. (2) Seal the gap between the cooling module and the front end module with sponge sealing strips, as shown in Figure 7. (3) Figure 8 shows the new deflector is based on the original deflector by adding two pieces of upper and lower ones. In addition, the gap between the deflector and the radiator water chamber, the anti-collision beam and the front bumper is filled. Compared with Fei hongqing[10], the optimization scheme proposed in this paper has higher feasibility and lower cost to increase the grille intake area and adjust the relative position of the cooling module.
Figure 7. Sealing strip

Figure 8. New deflector

Figure 9 shows the flow field distribution in the engine room of the optimization. After the new deflector was replaced and the sponge sealing strips were added, the hot air reflow at the top and bottom basically disappeared. Figure 10 shows that after the gaps between the deflector and the radiator water chamber, the anti-collision beam, the front bumper are filled with sponge sealing strips, the reflow of hot air on the left and right sides is obviously suppressed.

Figure 9. Y=0 mm

Figure 10. Z=260 mm

The air intake flow of the optimized grille and each radiator is shown in Table 4. The air intake flow of grille, motor radiator, condenser and low temperature radiator has increased obviously and the high temperature radiator is slightly reduced. The hot air reflow rate around cooling module is 11.92% because of the reflow phenomenon was suppressed. Table 5 shows the inlet air temperature of the optimized grille and each radiator has declined in varying degrees;

| Table 4. The air intake of the optimized grille and radiators(kg/s). |
|-----------------|-------|-------|---------------|---------------|
| Grille          | Motor radiator | Condenser | Low temperature radiator | High temperature radiator |
| 0.554           | 0.256          | 0.629    | 0.629          | 0.629          |

| Table 5. The inlet air temperature of the optimized grille and radiators (℃). |
|-----------------|-------|-------|---------------|---------------|
| Grille          | Motor radiator | Condenser | Low temperature radiator | High temperature radiator |
| 45.08           | 46.44          | 51.91    | 61.44          | 62.05          |

4.2 Comparison of results

From the comparison of figures 11 and 12, the average temperature of airflow in the engine room is significantly reduced after optimizing, the inlet air temperature of the motor radiator, condenser, low temperature radiator and high temperature radiator is reduced by 4.41℃, 6.37℃, 7.96℃ and 9.2℃ respectively. The air intake flow of grille has increased by 25.06%. The hot air reflow rate of the cooling module has been reduced by 26.55%. Reference of 10 shows the inlet air temperature of
cooling module drops by 2°C. Therefore, the optimization scheme in this paper has obvious advantages.

5. Conclusions
In this paper, a combination of numerical simulation and experimental research was used to study the influence of the sealing condition of a hybrid car cooling module on the heat dissipation characteristic of the engine room. Comparing the simulation results of the original status and the optimization with the data of thermal balance test. The conclusions are as followed:
(1) In the original state, the hot air reflow rate around the cooling module is 38.47%. By sealing the gaps of cooling module and front end module and optimizing the deflector structure, the hot air reflow can be effectively suppressed.
(2) After optimizing, the air intake flow of grille has increased by 25.06%, which can effectively improve the heat exchange efficiency of the radiator. The inlet air temperature of each radiator of the cooling module is significantly reduced. The average temperature of the airflow in the engine room drops by about 10°C.
(3) Comparing the thermal balance test data with simulation results, the errors of the simulation are within 15%, the correctness of the simulation results is verified.

6. References
[1] Kepeng Z, Peiyu C, Jun L, et al. Analysis on Thermal Management of a Hybrid Vehicle Engine Compartment[J]. Auto Engineer, 2019.
[2] Llhan Bayraktar. Computational simulation methods for car thermal management[J]. Applied Thermal Engineering, 2011, 36.
[3] Fortunato F, Damiano F, Matteo L D, et al. Underhood Cooling Simulation for Development of New cars[C]// car Thermal Management Systems Conference & Exposition. 2005.
[4] Kim J M, Kim K M, Ha S J, et al. Grille design for passenger car to improve aerodynamic and cooling performance using CFD technique[J]. International Journal of Automotive Technology, 2016, 17(6):967-976.
[5] Ma W S, Shen W X, Zhang L W. Heat rejection efficiency research of new energy car radiators[J]. Iop Conference, 2018, 324.
[6] Kumar V, Kapoor S, Arora G, et al. A Combined CFD and Flow Network Modeling Approach for car Underhood Air Flow and Thermal Analysis[C]// SAE World Congress & Exhibition. 2009.
[7] Shi P, Yu J,Li F, Liu H 2018 Auto Eng 01 41.
[8] Patidar A, Gupta U, Marathe N. Optimization of Front-End Cooling Module for Commercial car Using CFD Approach[C]// Symposium on International Automotive Technology. 2013.
[9] Li Z, Li T and Li X 2019 Chin. J. Auto. Eng. 6 400.
[10] Fei H, Tian J, Lei S, Chen Y and Yan W 2014 Internal Combust. Engine Powerplant 06 25.