Available online www.jocpr.com

Journal of Chemical and Pharmaceutical Research, 2015, 7(12):920-925

Research Article

ISSN : 0975-7384
CODEN(USA) : JCPRC5

Heat exchange coupling simulation of vehicle underhood based on Lattice-Boltzmann Method

Xiong Chun Ying, Liu Shu Ying, Yu Xian Zhong and Li Xiao Hua

Key Laboratory of Vehicle Noise and Vibration, Jiangxi Province, Jangling Motors Corp Ltd, Nanchang Jiangxi

---

ABSTRACT

To guide the design of front bumper opening and grille pattern, study the heat exchange performance and flow distribution of cooling system under specified conditions, whole vehicle with detailed underhood geometry was modeled. By adopting Lattice-Boltzmann method (LBM hereinafter) to simulate the thermal performance in Digital wind tunnel, underhood flow and temperature field as well as Radiator exit temperature were modeled, which the result shows good correlation to the test data. Therefore, LBM is verified to be quite suitable for evaluating heat exchange of underhood which has complex geometry. Finally, based on the analysis model, proposals on front bumper opening and grille pattern optimization were put forward, and 6.94% of radiator heat dissipation capability improvement was achieved through several rounds of iterations.

Key words: Cooling system; Lattice-Boltzmann; Heat exchange; Coupling simulation

---

INTRODUCTION

Following the enforcement of new emission regulations and application of new technologies in automotive industry, heat dissipation of cooling system under hood has become a hot topic attracted attention of researchers at home and abroad. Respond to the needs of shortening vehicle development cycle and cutting testing expenses, CFD numerical simulation method is adopted in the early stage of product development to analyze the performance of radiator cooling system. Based on traditional analysis methods, simplified analysis is done to numerous parts under hood, which reduces analytic precision of underhood flow distribution and results in poor computational accuracy of heat exchange between cooling system and external environment. While the characteristic of LBM determines that it is well suited to analyze underhood flow distribution featured with complex geometry. There are quite a lot underhood heat dissipation analysis been conducted with this method in abroad, providing effective ways to resolve such problems[1][2][3]. In China, such study is normally done with traditional fluid analysis method [8][10], thus, less literature based on LBM can be referenced to. In this paper, the model established has reserved whole vehicle model with detailed features of all underhood components maintained, giving consideration to the heat exchange of cooling system. The model simulated underhood velocity and temperature distribution, based on which to analyze the influence brought by the cooling air to the cooling system, and identified the primary causes of this influence. Accordingly, optimization proposal was raised and heat dissipation performance of cooling system was effectively improved.

MATHEMATICAL MODEL

Complex geometric details processing is a main difficult when simulating flow distribution under hood. Finite Volume Method is adopted to discrete flow field under traditional N-S solution, which raises higher requirement on grid quality and mesh size. Complex geometric details may result in significantly distorted element that will affect the calculation accuracy and even lead to computations convergence problem. Therefore, normally, simulation of underhood geometry details is not allowed or must be moderately simplified. As a result, the accuracy of underhood flow distribution is directly impacted. Even so, mesh quality still needs to be adjusted repeatedly, involving heavy labor intensity with long cycle and requiring users to be proficient in grid processing.

Differs from N-S equation, LBM does not require extra pressure correction equation to supplement the conservation, momentum and energy equation. Therefore, its numerical solution is more efficient and robust. Such higher efficiency is embodied in the use of larger number of grid cell. LBM-based fluid solution gains further improvement in boundary layer treatment that generates more flexible surface mesh, enabling its interactions with surrounding body-fitted mesh grid. With this method, surface details of complicated shape can be reserved without any simplification.

Applying LBM to simulate fluid flow field has made remarkable advances in recent years [4][5][6]. A brief description of this method has been given in this paper.

Expression of Boltzmann equation is as follows:

$$\frac{\partial}{\partial t}+\vec{v}\cdot\frac{\partial f}{\partial \vec{x}}=\mathbf{\Theta}$$

In which, the probability distribution functions for the velocity, $\mathbf{\Theta}$ represents the collision operator. In the lattice, this equation f_i can be unfolded into algebraic expression.

$$f_i\left(t+\Delta t,\vec{x}+\vec{e}_i\Delta t\right)=f_i\left(t,\vec{x}\right)+\mathbf{\Theta_i}\left(\mathbf{t},\vec{\mathbf{x}}\right)$$

Among them:

$$\mathbf{\Theta_i}\left(\mathbf{t},\vec{\mathbf{x}}\right)=-\frac{\Delta t}{\tau}\left[f_i\left(t,\vec{x}\right)-f_i^{eq}\left(t,\vec{x}\right)\right]$$

τ is the relaxation time and grid equilibrium distribution function, f_i^{eq} expressed as the function of velocity:

$$f_i^{eq}\left(t,\vec{x}\right)=t_p\left[1+\frac{e_{i\alpha}e_{i\beta}}{c_s^2}+\frac{u_\alpha u_\beta}{2c_s^2}\left(\frac{e_{i\alpha}e_{i\beta}}{c_s^2}-\delta_{\alpha\beta}\right)\right]$$

Through the definition of distribution function, hydraulic parameters, such as density, velocity can be expressed as:

$$\rho\left(t,\vec{x}\right)=\sum_i f_i\left(t,\vec{x}\right)$$
$$\vec{u}\left(t,\vec{x}\right)=\frac{1}{\rho\left(t,\vec{x}\right)}\sum f_i\left(t,\vec{x}\right)\vec{e}_i$$

Lattice Boltzmann solver to perform mass conservation and momentum equation, make all state in the collision term i sum to zero:

$$\sum_i \Theta\left(t,\vec{x}\right)=0$$
$$\sum_i \Theta\left(t,\vec{x}\right)\vec{e}_i=0$$

The influence of turbulence, using modified $\kappa-\varepsilon$ model based on RNG equation to establish.

$$\rho\frac{Dk}{Dt}=\frac{1}{\sigma}\frac{\partial}{\partial\vec{x}}\left[\left(\mu+\mu_t\right)\frac{\partial k}{\partial\vec{x}}\right]+P-\rho\varepsilon$$

The change in temperature is acquired by solving the following partial differential equation:

$$\rho c_p \frac{DT}{Dt} = \frac{\partial}{\partial \vec{x}} \left[\left(\frac{\mu c_p}{P_r} + \frac{\mu_t c_p}{P_{r_t}} \right) \frac{\partial T}{\partial \vec{x}} \right] + Q$$

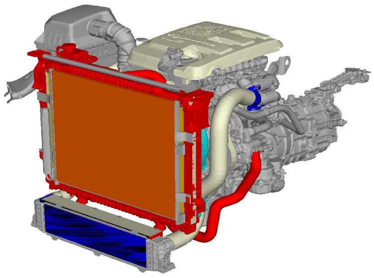


Fig.1: Engine with cooling system

PHISICS MODEL

When a vehicle is running on an open road, cold air enters into the engine compartment through the upper and mid grille of the front bumper, and flows out from the back of the vehicle. In order to accurately simulate the route of airflow through the engine bay, whole vehicle model was built. The engine bay detailed geometry and front bumper model was maintained without any simplification. Detailed engine cooling system model was also built as shown in figure 1.

Close-fitting between grid and Geometric faces should be achieved in Meshing. In order to accurately simulate the complex flow field in the engine bay, grids around key components including engine, grille opening, cooling system and cooling fan should be refined, so as to improve the calculation accuracy. The minimum mesh size around the cooling system is 1.25 millimeters. The volume mesh was generated by software automatically without human intervention; finally 90 million effective volume meshes was generated.

Boundary Condition

In order to simulate the vehicle driving on an open road, the vehicle model built was put into a digital wind tunnel model, the inlet boundary was set as 90 km/h of velocity boundary, and the outlet was set as pressure boundary. Fan and tires were set to the rotating wall boundary with MRF way, and the ground was set as moving wall. Except for flow field simulation, heat transfer between the flow field and the heat exchanger was also modeled. 1D tool was adopted to couple the heat exchange computation of cool air medium flows through inside and outside of heat exchanger. With this method, the internal flow of heat exchanger was treated as 1D flow and the flow distribution was reasonably simplified, so to simulate heat transfer process of heat exchanger alongside the airflow direction. Input parameters are as shown in table 1.

Tab.1: Boundary parameters

Ambient temperature	43℃
Velocity	90 km/h
Radiator mass flow	1.33 kg/s
Radiator inlet temperature	108 ℃
CAC mass flow	0.106 kg/s
CAC inlet temperature	168 ℃
Condenser heat rejection	12.9 kW

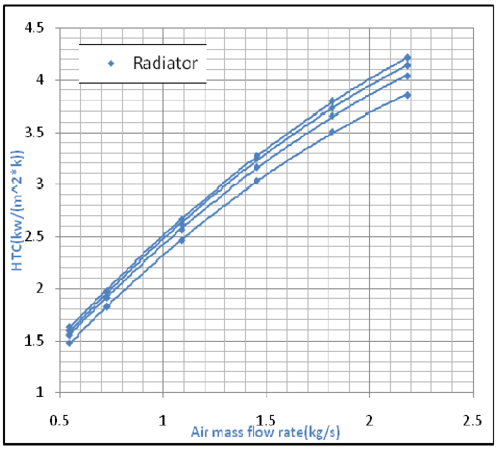


Fig.2: Heat transfer coefficient curve

Heat exchanger was modeled with porous media, and the resistance characteristic can be gained through tested relation between pressure drop and velocity and corrected through the Darcy's law. 1D tools were adopted to simulate the heat transfer progress between the heat source inner side with the air flow outer side. Heat transfer coefficient, gained through test data conversion, may have 5% error due to measurement error in measuring value point. Such error shall be eliminated through data processing technique. Usually two kinds of interpolation methods are used to fit experimental data. Double linear interpolation method[7] was adopted to fit the radiator's experimental data , as shown in figure 2.

Analysis and Validation

When analyzing underhood heat dissipation, rated power point of engine under the worst environment conditions was selected as input to be computed on 64 CPU cluster. After 30,000 time steps, coupling with 1D tools began, and then, with pace of one coupling every 5000 time steps, the whole calculation lasted about 5 days.

The temperature of cooling medium entering into radiator was our primary concern. In the process, the target temperature was set as Input. And after test data gained, it was substituted into 1D tool for coupling calculation, which show a only 0.1 ℃ gap with the measured temperature of cooling medium entered into radiator, as shown in table 2:

Tab.2: Analysis result vs. Test data

Parameters		simulation	test
Radiator	Top tank temperature	109.6℃	109.6℃
	Bottom tank temperature	102℃	101.9℃

Calculated value was well consistent with experimental values. This can be attributed to higher resolution of underhood geometry model, since it provides more accurate prediction on the flow field under hood as well as the air mass flow rate passing through the heat exchanger. Correspondingly, the prediction on the outlet temperature of cooling medium gets more accurate. Since the precise model consumed long computer time, it was not conducive to get the analysis result promptly and was unfavorable for timely evaluation to optimization proposals. Therefore, based on the correlation between digital prototype and physical prototype, we need to reduce the fineness of benchmarking analysis model to a reasonable degree. Compute cycle can be shortened with acceptable calculation accuracy guaranteed. As a result, computing time can be controlled within 2 days, which greatly improved the working efficiency.

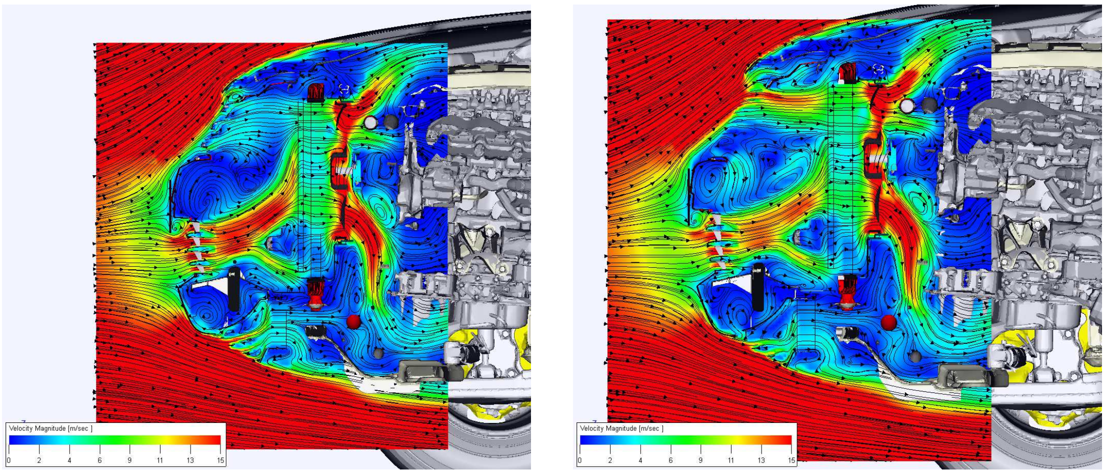
Result Analysis

The flow field structure in the underhood is as shown in the figure 3 (a). Airflow accelerated to pass through the grille opening, entered into the condenser and radiator, and was separated into up and down flow influenced by the front cross beam and formed a declination angle. Impacted by the shape of upper grille and angle of baffler, the airflow from upper grille entered into the engine bay at lower speed. A vortex was formed in the cavity between front bumper and the cooling system, which greatly affect the airflow entering into the cooling system. At the same time, poor sealing between cooling frame and surrounding parts led to fluid leak, which also affected the flow of air into the cooling system.

Since air cooler was located behind the bottom end of the front bumper, it was not favor for airflow inlet. Therefore, a guiding device in front of the Charge air cooler was installed. In the diagram, we can see that this device guide high-speed air into the inter cooler.

Optimization

In view of above problems, within the acceptable range of engineering design, proposals on grille shape optimization, grille openings optimization, seal chamber optimization and a variety of combination solutions were put forward, and the one with best effects was selected. As shown in the figure below, the grille opening was increased by 20%, shape and angle of the grille were scoped to a more reasonable range, and the chamfer between



（a）benchmarket **（b）optimization proposal**

Fig.3: Velocity distribution comparison in slice

the license plate and the grille was smoothed. As shown in figure 3 (b), it is clearly that the air velocity into the grille and into the cooling system increased obviously, and the direction of airflow becomes more reasonable. Making sure unobstructed airflow into the cooling system is key to cooling system package design. Table 3 below indicates the result before and after optimization;

Tab.3: Parameters comparison(baseline vs. opt)

Results		Benchmarket	Optimization
Radiator heat rejection		36.0KW	38.5KW
Upper grille	mass flow	0.12kg/s	0.46 kg/s
	Averaged velocity	3.39m/s	5.1m/s
Lower grille	mass flow	1.22kg/s	1.1kg/s
	Averaged velocity	10.9m/s	9.7m/s

It can be seen from the table that, after upper grille optimization, air inflow and average velocity gained significant improvement. Air flew into the radiator at a more even speed. Impacted by the flow field structure of upper griller, the air inflow and average velocity passing through the lower grille were decreased. Nonetheless, the cooling efficiency of radiator was improved with heat dissipating capacity increase of 2.5 KW .

CONCLUSION

LBM was adopted to conduct underhood heat dissipation 1D/3D coupling analysis, and comparative verification was done with experimental data. On this basis, optimization proposals were raised which improved underhood flow field characteristics and cooling capacity of cooling system. Therefore, we came to conclusions as below:

(1)By using CFD software, a detailed model of vehicle and engine bay was built to analyze and simulate the underhood cooling system performance. With this model, the velocity field and temperature field under hood were simulated which provided valuable reference for engine bay packaging study and body style design .

(2)The consistency of correlation between the analysis results and the experimental data provides accurate and effective guidance to the evaluation of cooling system performance through virtual method .

(3)Unobstructed air inlet passage and uniformity of air velocity distribution will directly impact the heat dissipation capacity and its efficiency.

REFERENCES

[1] Alajbegovic A, Sengupta R Jansen W. *SAE Int*, 01-3478, **2006**.
[2] Bhatnagar B, Schlesinger D, Alajbegovic A et al. *SAE Int,* 01-4111, **2007**.
[3] Lu L, Zhang L, Liu S et al. *Optimization of Aerodynamics and Engine Cooling Performance of a JMC Mid-Size Truck using Simulation*, *SAE Int*, 01-2032, **2010**.
[4] Frisch U, Hasslacher B, Pomeneau Y. *Physical Review Letters*, 56(14), 1505-1508, **1986**
[5] Chen S, Doolen GD. *Annual Review of Fluid Mechanics*, 30, 329-364, **1998**
[6] Succi S. The Lattice Boltzmann Equation for Fluid Dynamics and Beyond, Series Numerical Mathematics and Scientific Computation, New York: Oxford University Press Inc., Oxford, **2001**, 120-140.
[7] PowerCOOL User`s Guide, Release 4.0,Exa Corporation, Boston,Massachusetts,**2006**.
[8] Yuan Xiayi, Gu Zhengqi, Yang Yi et a, *Automotive Engineering*, **2009**, 9(31), 843-847.
[9] Zhang Xiaojun, Liu Zuyuan, Wan Mingfang. *Journal of WuHan university of Technology*, **2002**,3(24), 46-49.
[10] Wang Fujun. Computional fluid dynamics simulation, Beijing：Tsinghua University Press, **2004**, 50-140.