## INTEGRATED TESTING OF ELECTRIC VEHICLE THERMAL MANAGEMENT AND OPTIMIZATION OF FLOW FIELD UNIFORMITY IN AIR SUPPLY SYSTEM

#### by

### Yu FU<sup>a,b</sup>, Haitao MIN<sup>a</sup>, Weiyi SUN<sup>a\*</sup>, and Yang FANG<sup>c</sup>

<sup>a</sup> State Key Laboratory of Automotive Simulation and Control, Changchun, China <sup>b</sup> CATARC New Energy Vehicle Test Center (TianJin) Co., Ltd., Tian Jin, China <sup>c</sup> General Research and Development Institute, China FAW Corporation Limited, Changchun, China

> Original scientific paper https://doi.org/10.2298/TSCI230926045F

To meet the requirements of integrated testing and performance calibration for the early development of electric vehicle thermal management, this paper conducted research on the configuration scheme design of the electric vehicle thermal management integrated testing platform and simulation and optimization of the air supply system flow field. Firstly, a requirement analysis of the testing environment and the overall design of the testing platform was performed. Secondly, simulation techniques were applied to optimize the uniformity of the intake air-flow field in the test section, which was then validated through real testing. Finally, a comparative study of the tested system was conducted through CFD simulations in two different environments, the testing platform and the actual vehicle underhood. The results indicate that the intake air-flow field uniformity in the test section falls within  $\pm 3\%$ . The intake flow rate errors for the radiator and condenser, compared to those in the actual vehicle, were found to be 0.039% and 2.116%, respectively. These findings confirm that the testing platform offers good consistency with the flow field in real vehicle testing scenarios.

Key words: thermal management, bench test, integrated design, air supply system, flow field simulation

### Introduction

To enhance the safety and comfort of electric vehicles while reducing energy consumption, the design of the electric vehicle thermal management system is trending towards integration, with a highly unified system structure, strong inter-subsystem energy coupling, and intricate control strategies [1]. The strong coupling of energy between the power electronics, electric drive, and thermal management systems increases the complexity of development. In the development process, extensive integrated thermal management performance testing at the system level is required as a prerequisite for testing. Considering the strong coupling between various systems in pure electric vehicles, the integrated testing should encompass the core systems of power electronics, electric drive, thermal management, and control. These system components should be integrated into a test rig, replicating the actual operating environment of the vehicle's components, thus creating an *equivalent to a small real vehicle*. This allows for the comprehensive testing of system performance at the system level, guiding development optimization. Feng [2] developed

<sup>\*</sup>Corresponding author, e-mail: swy\_18@jlu.edu.cn

a vehicle-mounted thermal management integrated testing system based on physical components in commercial vehicle cabins. This system can monitor variables like flow rates, temperatures, pressures, vehicle speed, engine speed, and throttle position, establishing relationships between these elements. However, it focuses on testing actual vehicles, and the testing phase still lags behind. Zhao [3] conducted an in-depth study on the thermodynamic model of a thermal management system test bench, enabling the calculation of parameters like dry bulb temperature, wet bulb temperature, pressure, air-flow rate, cooling capacity, and heating capacity. However, this test bench only exists as a model and lacks physical testing hardware for specific experiments. Wang et al. [4] modified a heat pump test bench to verify the performance of the heat pump system in pure electric vehicles. They added some water circuits to the test bench to test the heat pump system's performance and calibrate optimal control parameters such as compressor speed, expansion valve opening, and subcooling. However, this test bench only focuses on the heat pump system's performance and does not include the electric drive system and battery system's thermal management circuits, making it unable to test multi-system coupling performance. Xue [5] developed a multi-system integrated test rig based on real components of electric vehicles, including the power electronics, electric drive, thermal management, and control core systems. However, this test rig does not control the flow field and temperature field, allowing only for functional and room temperature performance testing and preventing it from conducting high and low temperature performance testing. Ma et al. [6] placed the tested system in an environmental chamber where the relative positions of the components could be maintained. The environmental chamber provides environmental boundary conditions such as temperature, humidity, and flow field for the tested components. However, the differences in temperature, humidity, and flow field environments in which the electric drive system, battery system, and air conditioning system operate in real vehicle operation are significant. Therefore, the environmental chamber can only provide a single environment and cannot simulate the working environment consistent with the actual vehicle for the tested components. In summary, the existing research breaks the relative positions and coupling relationships between components, with test equipment providing only a single test environment for the tested system without controlling the flow field and temperature field in different component areas. In actual driving, the environmental conditions in different areas inside the vehicle vary significantly. Therefore, there is an urgent need to develop an electric vehicle thermal management integrated test system that can carry the core components of the vehicle, maintain the real relative positions of the components, and simulate the environments in which the components operate. This is to meet the initial development requirements of electric vehicles for integrated thermal management performance testing and performance calibration, to validate system performance early, reduce development risks, and significantly shorten the development time. In response to the shortcomings of existing research and the design principles of testing environment scenario and testing condition operationalization, this paper first starts with an analysis of the working principles of the tested system and the environmental requirements for key components' testing. It proposes and designs the overall configuration and operating principles of the electric vehicle thermal management integrated test platform, completing the construction of the testing environment scenario. A particular focus is placed on the simulation analysis and optimized design of the air-flow system in the test platform, and the optimization and empirical verification of the uniformity of the inflow air-flow field at the entrance to the test section. This provides a foundational assurance for precise control of the inflow air volume. Finally, to verify the consistency of the flow field environment provided by the test platform and the actual vehicle environment, a comparative CFD simulation study is conducted on the tested system based on both the test platform and the actual vehicle underhood environment.

#### **Design requirements analysis**

The environmental conditions in different areas of vehicles vary greatly, with differences in the underhood environment, air conditioning core environment, and battery environment [7-9]. The following will provide a detailed analysis of the demand environment for core components in the three major spaces.

## Analysis of environmental requirements for underhood internal components

The electric drive system and front-end cooling module are the core components in the cabin. The process of converting electrical energy into mechanical energy in the electric drive system will generate energy loss, mainly including iron core loss, winding loss, and mechanical loss. The internal cooling cycle of the motor carries away a portion of the heat, and it is calculated as:

$$P_{w} = \rho_{w} C_{w} A_{w} v (T_{\rm in} - T_{\rm out}) \tag{1}$$

where  $\rho_w$  is the density of the coolant,  $C_w$  – the specific heat capacity of the coolant,  $A_w$  – the cross-sectional area of the cooling pipe-line, v – the flow rate of the coolant, and  $T_{in}$ ,  $T_{out}$  refer to the inlet and outlet temperatures, respectively.

The dissipation power loss of the stator and its winding to the surrounding environment:

$$P_{s_{air}} = \frac{\delta A_s (T_s - T_0)}{t_2 - t_1}$$
(2)

where  $\delta$  is the convective heat transfer coefficient,  $A_s$  – the convective heat transfer area on the surface of the stator and winding,  $T_s$  – the surface temperature of the stator and winding,  $T_0$  – the ambient temperature, and  $t_2$ ,  $t_1$  are the starting and ending time points within a unit time, respectively.

The convective heat transfer coefficient is calculated using the following empirical equation:

$$\delta = 9.73 + 14V^{0.62} \tag{3}$$

where V is the air-flow velocity of the heat dissipation surface.

In summary, it can be seen that the heat generation of the motor is directly related to the operating conditions. The dissipated power of the stator and its winding to the surrounding environment is affected by the environmental temperature and convective heat transfer coefficient, while the convective heat transfer system is also related to the air-flow velocity on the heat dissipation surface. Therefore, an integrated thermal management testing system needs to provide operating load conditions, temperature and flow field conditions similar to those in the engine compartment during actual vehicle operation for the electric drive system. To ensure that the flow field distribution around the electric drive system is consistent with the actual vehicle, and to ensure accurate air intake, the air-flow velocity distribution at the entrance of the test section should be consistent with the air-flow velocity distribution entering the engine room, as uniform as possible.

### Analysis of environmental requirements inside the air conditioning box

The core components inside the air conditioning box are evaporators, built-in condensers, or heater cores. Taking the heater core as an example, the heat transfer calculation equations are [10]:

$$\Phi = KA\Delta t_m \tag{4}$$

$$K = \left[\frac{\beta}{h_1} + (\beta\lambda_3) + \left(\frac{1}{\eta_0}h_2\right) + R_f\right]^{-1}$$
(5)

$$h = \mu_u \frac{k}{L} \tag{6}$$

$$\mu_u = C \operatorname{Re} \tag{7}$$

$$\operatorname{Re} = u \frac{d}{v} \tag{8}$$

where  $\Phi$  is the heat transfer coefficient, K – the heat transfer coefficient, A – the heat transfer area,  $\Delta t_m$  – the logarithmic average temperature difference between the cold and hot fluids,  $\beta$  – the rib coefficient,  $h_1$  – the surface heat transfer coefficient on the water side,  $h_2$  – the surface heat transfer coefficient on the air side,  $\lambda_3$  – the thermal conductivity coefficient of the pipe material,  $R_f$  – the total thermal resistance between the water side and the air side, h – the surface heat transfer coefficient, L – the characteristic scale of the heat exchange surface, Re – the Reynolds number, d – the pipe diameter, u – the average flow velocity in the pipe, and v – the viscous system of liquid motion. From the principle of core heat transfer, it can be seen that the heat transfer performance is directly related to the ambient air temperature and the flow rate through the core. The heat transfer coefficient of the gas passing through its core surface. Therefore, an integrated thermal management testing system needs to provide target temperature and precise air supply for the components inside the air conditioning box.

## Analysis of environmental requirements around the power battery system

Taking lithium iron phosphate battery as an example, the mathematical control equation for its 3-D electrically thermally coupled mode is [11]:

$$\rho_p C_p \frac{\partial T}{\partial \tau} = k_{px} \frac{\partial^2 T}{\partial x^2} + k_{py} \frac{\partial^2 T}{\partial y^2} + k_{pz} \frac{\partial^2 T}{\partial z^2} + \dot{q}_p - \dot{q}_{fp}$$
(9)

$$\rho_n C_n \frac{\partial T}{\partial \tau} = k_{nx} \frac{\partial^2 T}{\partial x^2} + k_{ny} \frac{\partial^2 T}{\partial y^2} + k_{nz} \frac{\partial^2 T}{\partial z^2} + \dot{q}_p - \dot{q}_{fn}$$
(10)

$$\rho_r C_r \frac{\partial T}{\partial \tau} = k_{rx} \frac{\partial^2 T}{\partial x^2} + k_{ry} \frac{\partial^2 T}{\partial y^2} + k_{rz} \frac{\partial^2 T}{\partial z^2} + \dot{q}_p - \dot{q}_{fr}$$
(11)

$$\dot{q}_{p} = J_{p} \left( U_{0} - U - T \frac{\mathrm{d}U_{0}}{\mathrm{d}T} \right) + I_{p}^{2} R_{pp}$$
(12)

$$\dot{q}_{n} = J_{n} \left( U_{0} - U - T \frac{\mathrm{d}U_{0}}{\mathrm{d}T} \right) + I_{n}^{2} R_{pn}$$
(13)

$$\dot{q}_r = 0 \tag{14}$$

$$J_{p} = \sigma_{cp} \Delta \phi p \tag{15}$$

2417

$$J_n = \sigma_{cn} \Delta \phi n \tag{16}$$

$$\Delta\phi_p = \frac{I_{tp}}{\sigma_{cp}^h p^s p} \tag{17}$$

$$\Delta \phi_n = \frac{I_{tp}}{\sigma_{cn}^h n^s n} \tag{18}$$

where  $\dot{q}_p$ ,  $\dot{q}_n$ ,  $\dot{q}_r$  [Wm<sup>-3</sup>] are the heat production rates of the positive electrode current collector, the negative electrode current collector, and the battery plate, respectively,  $R_{pp}$ ,  $R_{pn}$  [ $\Omega$ ] – the equivalent polarization internal resistance of the positive and negative plate current collectors,  $\sigma_{cp}$ ,  $\sigma_{cn}$  [Sm<sup>-1</sup>] – the conductivity of the positive and negative electrode plates,  $\Delta\phi_p$ ,  $\Delta\phi_n$  [V] – the potential differences within the positive and negative electrode plates are, respectively, and  $I_{lp}$ ,  $I_{ln}$  – the currents flowing through the positive and negative electrode plates.

From the heat generation model of the power battery system, it can be seen that the heat generation of the battery is influenced by operating conditions and internal resistance. The size of the internal resistance of the battery is related to temperature [12]. Research has shown that when the battery temperature decreases by 5 °C, the internal resistance increases by  $35 \text{ m}\Omega$  [13]. The lower the ambient temperature of the battery, the greater its internal resistance value, and the higher its heat generation [12]. Therefore, the heat generation of the battery is related to the ambient temperature. The heat transfer methods of batteries can be divided into three types, thermal conduction, thermal convection, and thermal radiation. Currently, the mainstream cooling methods for batteries in the industry are liquid cooling and direct cooling. The bottom of the battery is separated from the battery surface by a partition, and the convective heat transfer between the battery surface and the external air-flow can be ignored. However, the high temperature of the asphalt ground in summer cannot ignore its thermal radiation the battery. In summary, the integrated thermal management testing system needs to provide the power battery system with temperature environment, ground radiation, and real operating load similar to that of the battery under actual vehicle operation.

Based on the aforementioned analysis, it can be concluded that the testing system needs to provide real operating loads for the electric drive system, target temperature, humidity, and precise and uniform air supply for the internal components of the cabin. Provide target



Figure 1. Test environment requirements and system configuration for each system

Fu, Y.,	et al.: Integrated	Testing of Ele	ectric Vehicle	Thermal M	anagement
	THERMAL SO	CIENCE: Year	2024, Vol. 28	3, No. 3B, p	p. 2413-2431

temperature, humidity, and precise air supply for the heat exchanger in the air conditioning box, and provide temperature and ground radiation environment for the power battery system. Based on the analysis of the environmental conditions required for each region, the platform needs to be equipped with a temperature control system, humidity control system, and air supply control system. Each system is independently controlled to provide target environments for each region. The platform requirements and system configuration are shown in fig. 1.

## Research on the overall design of the system

#### Overall system configuration and operating principles

According to the aforementioned analysis conclusion, the system scheme includes simulated underhood environment air duct that is independent of each other, simulated HVAC (heating ventilation and air conditioning) environment air duct, simulated battery temperature control box, and two sets of dynamometers. The lay-out fully considers the true position of electric vehicle components, pipe-line length, and compact structure. The specific structural form is shown in fig. 2.



Figure 2. Schematic diagram of system lay-out structure

The environmental conditions in the system are achieved by the air supply system, the temperature control system and humidity control system, and the operating principal diagram is shown in fig. 3. The temperature control of the simulated underhood environment air duct and the simulated HVAC environment air duct adopts intercooling control mode, while the simulated battery temperature control box adopts direct cooling control mode. The refrigeration unit and heater in the temperature control system of the simulated underhood environment air duct and the simulated HVAC environment air duct control the temperature of the constant temperature water tank. The cooling water in the constant temperature water tank is connected to the inner core of the temperature and humidity control section of the air duct through pipelines, and the temperature of the simulated underhood environment air duct and the inner core of the temperature and humidity control section of the simulated HVAC environment air duct are controlled through the opening of the electromagnetic valve body. The coupling effect of the power fan and the bypass fan is to supply air to the temperature control section core and the test section, in order to achieve temperature control in the testing section, simulate the underhood environment air duct and simulate the HVAC environment air duct. Dehumidification and humidification devices are installed behind the core of the temperature control section achieve humidity control. The simulated battery temperature control box control system is equipped with a cooling unit and heater, and the internal temperature of the simulated battery temperature control box is controlled through a mixture of cold and warm air.



Figure 3. System operation principal diagram

## Design of air duct structure for simulating underhood environment

The simulated underhood environment air duct provides constant temperature, humidity, and precise air supply for the internal components of the underhood, such as the electric drive system and front-end cooling module, consistent with the test conditions. To restore the flow field inside the actual vehicle underhood, the inlet and outlet positions of the test section, the width and axial length of the test section, and the lay-out and installation of the tested parts all refer to the design of the actual vehicle underhood. The air inlet of the actual engine compartment is located at the front of the radiator and condenser, and the air outlet is located on both sides below the rear of the engine compartment. Therefore, the air inlet of the test section is also designed at the front end, and the return air outlet is designed on both sides below the rear end. Considering the need for temperature and humidity control in the air duct, in order to reduce system operation energy consumption, the simulated underhood environment air duct is designed as a closed double circulation structure. As shown in fig. 4, the simulated underhood environment air duct is equipped with a power section, rectification section, testing section, side pipe section, convergence section, and temperature and humidity control section. The tested component is installed in the testing section, and the array fan is installed in the testing section. The heat transfer capacity of the temperature and humidity control system is directly related to the air-flow velocity passing through the system's heat exchanger. Under low wind speed conditions, the lower air-flow through the temperature and humidity control system results in a decrease in the system's heat transfer capacity. If the operating conditions of the tested piece are in good agreement at this time and its transient heat production increases, the temperature and humidity control system cannot take away excess heat in a short period of time, resulting in an increase in the inlet temperature of the test section and unable to provide stable temperature air supply for the tested piece. To solve this problem, this article adds a bypass section above the temperature and humidity control section of the simulated underhood environment air duct. The installation of a circulating fan in the section aims to increase the surface flow rate of the heat exchanger, ensure the heat exchange capacity of the temperature control system at various operating points, and achieve stable temperature control.



Figure 4. Top and oblique side view of simulated underhood environment air duct structure; (a) top view and (b) oblique side view

The air supply range in the platform design requirements is 500~14000 m<sup>3</sup> per hour. The maximum wind speed change rate is 3 m/s, corresponding to a flow rate change rate of 1080 m<sup>3</sup> per hour. Increasing the rotational inertia of the fan will reduce its sensitivity [14, 15]. According to theoretical equation:

$$J = \frac{mr^2}{2} \tag{19}$$

where  $J \,[\text{kgm}^{-2}]$  is the moment of inertiam,  $m \,[\text{kg}]$  – the mass, and  $r \,[\text{m}]$  – the distance from motor axis to external assembly.

A single large fan motor mass is 40 kg, a fan blade mass of 20 kg, a motor rotation radius of 0.02 m, and a fan shaft radius of 0.5 m. Therefore, acorresponding to the eq. (20) the inertia of A single large fan motor is  $2.5 \text{ kg/m}^2$ :

$$T = J \frac{\Delta \omega}{\Delta t} \tag{20}$$

where  $\Delta \omega$  is the rotation angle and  $\Delta t$  [second] – the time, determine the speed change as 100 r/s based on the change in air volume, acorresponding to the eq. (20), the torque is 1575.024 Nm. In addition, it also needs to overcome aerodynamic resistance and mechanical friction, and the



Figure 5. Schematic diagram of array fan

total torque provided by the motor is relatively large. To reduce the rotational inertia of the fan and reduce the working load of the motor under rapidly changing wind speed conditions, it is considered to discretize a fan into several small fans. Meanwhile, considering the uniformity of the system's air supply flow rate, the number of fans should be 4, 9, and 16. Considering the cost of the fan and the lay-out size of the air duct, a comparative study was conducted on 4 fan and 9 fan schemes. After calculation, in order to achieve rapid changes in wind speed, the torque required for each of the four fan schemes is 14.2 Nm, while the torque required

for each of the nine fan schemes is only 1.73 Nm. Both options require a significant reduction in the torque required for the fan, which is acceptable. But by comparing the flow field distribution at the entrance of the test section between the two schemes through simulation methods, taking into account both. After theoretical calculation, using traditional environmental cabin axial flow fans, the uniformity coefficient of 9 fans became 0.9420, and the uniformity coefficient of 4 fans became 0.4298. Therefore, 9 fan schemes were ultimately selected, the array fan model is shown in fig. 5.

### Structural design of air ducts in simulated HVAC environments

The simulated HVAC environment air duct is a closed single cycle structure, which provides target temperature, humidity, and flow intake for the evaporator or heating core in the air conditioning system. As shown in fig. 6, the simulated HVAC environment air duct includes a temperature and humidity control section, a power section, a rectification section, and a testing section. The tested core is installed in the testing section, and the array fan is installed in the power section. The air-flow enters the power section after temperature and humidity control treatment through the temperature and humidity control section section ensure uniform and consistent air-flow towards the test section. After passing through the testing section, the air-flow circulates to the temperature and humidity control section, and so on, continuously providing the required air supply for the tested object. Similar to the simulated underhood environment air duct, in order to ensure the heat transfer capacity of the temperature and humidity control system at various operating points, a bypass section was also designed for the simulated HVAC environment air duct, and a bypass fan was installed.



Figure 6. Top and oblique side view of simulated HVAC environment air duct structure; (a) top view and (b) oblique side view

#### Structural design of battery temperature control box

As shown in fig. 7, the battery temperature control box provides the target temperature environment for the battery system. At the same time, to simulate the thermal radiation of the battery on the asphalt ground in summer, a heating plate is installed at the bottom of the box and the temperature of the heating plate is adjustable.

The aforementioned system structure is designed based on the actual vehicle lay-out, which can carry real components and retain their true relative position relationships, providing a prerequisite for accurate testing. The platform relies on air supply systems, temperature control systems, and humidity control systems to provide environmental conditions that are consistent with the actual vehicle operation for the tested components. In order to restore the real environmental conditions around the components and ensure that the air-flow flow, temperature, and humidity passing through each component are consistent with the real situation, it is necessary to conduct in-depth research on how the environmental air duct provides accurate



Figure 7. Top and oblique side view of battery temperature control box; (a) top view and (b) oblique side view

temperature, humidity, and flow field environment. However, considering that temperature and humidity control technology is relatively mature, this article will not conduct in-depth research on temperature and humidity control, but will conduct in-depth research on the flow field of the integrated testing system flow field testing section. Based on the analysis of the demand environment for internal components in the cabin, the simulation of HVAC environmental air ducts and the simulation of HVAC environmental air ducts are different from environmental wind tunnels, and flow field related indicators such as boundary-layer and axial static pressure gradient are temporarily not considered. The uniformity of the inlet air in front of the tested piece directly affects the velocity distribution on the surface of the heat exchanger, thereby affecting the heat transfer performance. During the actual operation of the vehicle, the intake air into the engine compartment is uniform. Therefore, in order to restore the flow field distribution on the surface of the heat exchanger, the platform needs to provide uniform intake air for the tested object. But as mentioned in section Design of air duct structure for siulating underhood environment, to ensure the heat exchange capacity of the heat exchanger, a bypass fan has been added to the system. The coupling between the bypass fan and the active power fan during operation directly affects the uniformity of the inlet air of the test section, and the uniformity of the inlet air in the system is the research focus. The simulated cabin environment air duct is a closed double circulation structure, with numerous internal tested components and a complex overall structure. The flow field in the system is complex, and the side ventilation fan has high power and operating speed, which greatly affects the uniformity of the test section inlet. Therefore, this article takes the simulated cabin environment air duct as an example to conduct in-depth research on the uniformity of the flow field at the entrance of the test section.

### Optimization and verification of flow field uniformity at the entrance of the testing section

The entrance of the simulated cabin environment air duct test section is connected to the test piece, and the uniformity of the inlet air directly affects the surface velocity distribution of the heat exchanger of the test piece. Therefore, the uniformity of the flow field at the test entrance should be a key research area. In the following text, simulation analysis and measurement methods will be used to solve the problem of the uniformity of the flow field at the entrance of the test section.

# Optimization of velocity uniformity at the entrance of the testing section

Simulation study on flow field uniformity

- Model building

In order to improve the uniformity of the inlet air-flow field of the test section, this paper applies CFD simulation analysis technology to calculate the flow field of the Simulate the underhood environment air duct. Simulation software selection STARCCM<sup>+</sup> software. The model includes all the structures of the wind tunnel body, main fan, bypass fan, temperature and humidity control system heat exchanger, and corner guide structure. The main fan is an array fan composed of 9 small fans, installed in the power section. The specific model is shown in fig. 8(a). As shown in in fig. 8(b), the calculation domain of the model is the area inside the air duct, and the cut volume grid is selected. The grid size is  $6\sim12$  mm. In order to obtain detailed and accurate flow field information, boundary-layers are set up on the air duct and fan walls. Based on the *Y*+ value, the thickness of the first boundary-layer is calculated to be 0.1 mm, with a total thickness of 10 mm and 8 layers, the grid size is 26 million.



Figure 8. Wind tunnel model; (a) geometric model and (b) grid model

The physical model is a 3-D, steady-state, gas, separated flow,  $k-\varepsilon$  turbulence model [16-21], with an array fan speed of 4000 rpm.

Simulation results of flow field before optimization

As shown in fig. 9(a), the air-flow is blown out by the array fan, and after passing through the test section, two connection sections, and corner section, it is re converged in the power section, which can achieve the function of providing continuous air supply for the tested object. However, from fig. 9(b), it can be seen that the air-flow enters the stable section after being blown out by the array fan, but the air-flow streamline directions are different, mixed and interfering with each other, resulting in poor flow field quality at the entrance of the testing area.



Figure 9. Flow line diagram of simulate the underhood environment air duct; (a) whole and (b) detail

From fig. 10, it can be seen that the air-flow velocity distribution at the entrance of the test section is extremely uneven, with a uniformity coefficient of -0.0485 and a turbulence de-

gree of 4.1228. Additionally, there is a *reflux* phenomenon in the lower right area of the section, which cannot provide uniform inflow for the tested object.



Figure 10. Distribution of mainstream velocity direction velocity at the entrance of the testing section

## Research on simulation and optimization of flow field uniformity

#### Model building

Based on the aforementioned analysis, it is necessary to sort out the air-flow after the fan blows out, improve the uniformity of the inlet speed of the test section, and reduce the turbulence of the incoming flow. Adding a *honeycomb guide* structure in the air duct can suppress lateral air-flow movement, cut largescale vortices, and streamline air-flow. This ar-

ticle selects a hexagonal honeycomb structure to comb the air-flow, in order to predict the effect of adding honeycomb devices on improving the quality of the flow field. This article establishes a real honeycomb model installed in the Simulate the underhood environment air duct.



Figure 11. Simplified model of honeycomb

– Optimization plan design

The structural size of a honeycomb directly affects the uniformity of velocity and turbulence. In order to design the optimal honeycomb structure, it is necessary to study the relationship between the width and length of the honeycomb cell, uniformity, and turbulence separately. This article uses the CFD method to simulate the flow field of honeycomb devices with different structural sizes. Due to the large number of honeycomb cells, the simulation calculation requires a lot of time and computational resources. To improve simulation efficiency, this article simplifies the simulation model and only models the rectifier region where the honeycomb is located, as shown in fig. 11.

As shown in fig. 12, the core cell widths were calculated to be 10 mm, 15 mm, 20 mm, and 25 mm, with a honeycomb cell wall thickness of 1.5 mm and a length of 100 mm. The simulated grid size ranges from 67 million 83 million. The core cell lengths are calculated to be 100 mm, 150 mm, 200 mm, and 250 mm, respectively. The honeycomb cell has a wall thickness of 1.5 mm, a core cell width of 10 mm, and a simulated grid size of 83 million 160 million.

#### - Simulation optimization analysis

The velocity uniformity coefficient and turbulence intensity at the outlet position were obtained through simulation calculation under different core cell widths. From figs. 13(a) and 13(b), it can be seen that as the width of the core cell decreases, the uniformity of velocity increases, the turbulence level decreases, and the quality of the flow field improves. However, if the width of the core cell is too small, it will increase costs, increase the pressure loss of the air duct, and increase operating energy consumption. Taking into account the improvement of

Fu, Y., et al.: Integrated Testing of Electric Vehicle Thermal Management ... THERMAL SCIENCE: Year 2024, Vol. 28, No. 3B, pp. 2413-2431



Figure 12. Schematic diagram of core cell; (a) width and (b) length

flow field quality, production process, and manufacturing costs, this article selects a core cell width of 10 mm.



Figure 13. Trend chart of variation in velocity uniformity coefficient and turbulence of core cells with different widths; (a) velocity uniformity coefficient and (b) turbulence

From figs. 14(a) and 14(b), it can be seen that as the length of the core cell increases, the uniformity of velocity increases, the turbulence level decreases, and the quality of the flow field improves. But when the length increases to 150 mm, the rate of turbulence decrease and the rate of velocity uniformity increase both decreases. The total length of the rectification section of the simulate underhood environment air duct is 800 mm. Considering that this section needs to be equipped with honeycomb and damping mesh structures, and a certain space area should be reserved for buffering the air-flow. Therefore, this article selects a core cell length of 150 mm.



Figure 14. Trend chart of variation in velocity uniformity coefficient and turbulence of core cells with different lengths; (a) velocity uniformity coefficient and (b) turbulence

A honeycomb cell with a width of 10 mm and a length of 150 mm was installed in the simulate underhood environment air duct for simulation calculation. The simulation results are shown in fig. 22. Compared to fig. 15, it can be seen that after adding a honeycomb filter, the



Figure 15. Flow line diagram of stable section after adding honeycomb in simulate the underhood environment air duct

air-flow is *sorted* more smoothly, with higher consistency in flow direction, and there is no *reflux* phenomenon at the entrance section of the test section. Its uniformity coefficient has been changed to 0.9420, and the turbulence degree has been changed to 0.0247. On the one hand, it can ensure uniform flow for the tested object, and on the other hand, it can provide a high quality flow field foundation for accurate air supply.

## *Experimental verification of uniformity of flow field at the entrance of the testing section*

After designing the platform structure and optimizing the flow field simulation, the testing platform was built and debugged. The entity of the testing platform is shown in fig. 16.



Figure 16. Front and side view real time of test platform; (a) front view and (b) side view



Figure 17. Testing shrinkage tooling

ment points under different main fan speeds, the uniformity of the flow field at the necking section of the test section was obtained.

Figure 18. Schematic diagram of the measurement points for necking speed in the test section



To investigate the uniformity of the flow field before entering the grid of the tested object, this article arranges 7 flow measurement points at the outlet of the diversion necking, as shown in fig. 18. Measurement Point 3 is the center position of the necking, and measurement Points 1, 2, and 6 are symmetrical with measurement Points 5, 4, and 7, respectively. By testing the flow rate at different measure



The test results are shown in tab. 1. This article tested the necking flow rate of the main fan at the operating points of 500 rpm, 1000 rpm, 1500 rpm, 2000 rpm, and 2150 rpm, and obtained the flow deviation relative to the average value at different positions. The test results show that the flow deviation at Points 1-7 is less than  $\pm 3\%$ , with good uniformity and meeting the testing requirements.

Fan speed [rpm]	Flow rate at different measuring points [m <sup>3</sup> per hour]					Uniformity of different measurement point positions [%]								
	1	2	3	4	5	6	7	1	2	3	4	5	6	7
500	1250.61	1239.41	1257.06	1261.68	1268.27	1250.55	1260.53	-0.39	-1.28	0.13	0.5	1.02	-0.39	0.41
1000	2889.61	2803.56	2802.11	2812.15	2841.50	2767.37	2865.46	2.25	-0.79	-0.84	-0.49	0.55	-2.07	1.40
1500	4540.44	4374.85	4428.51	4397.57	4454.27	4299.71	4454.83	2.69	-1.05	0.16	-0.54	0.74	-2.75	0.75
2000	6148.41	5926.34	5970.16	5980.58	6056.74	5816.48	6047.28	2.61	-1.10	-0.37	-0.20	1.08	-2.93	0.92
2150	6398.68	6380.57	6459.77	6458.38	6525.43	6275.67	6364.80	-0.16	-0.44	0.79	0.77	1.82	-2.08	-0.69

Table 1. Test section uniformity measurement results

#### **Research on the consistency of surface** flow field distribution of tested parts

The distribution of air-flow entering the engine compartment during actual operation of the vehicle is uniform, and after passing through various components in the engine compartment, the distribution on the surface of the heat exchanger is no longer uniform, which directly affects the heat transfer performance. To reflect the true heat transfer performance of the components, the platform should not only provide uniform air supply for the tested parts, but also ensure that the velocity distribution of key components on the surface of the test platform should be consistent with the velocity distribution on the actual surface of the engine room. Therefore, the following will conduct flow field simulation analysis on the tested key components installed on the testing platform and the actual vehicle cabin environment to verify the consistency of flow field distribution.

### Simulation model

## Components mounted on the simulate underhood environment air duct model

Through the optimization of the flow field in the previous section, it can be seen that after adding a *honeycomb* steady flow structure, the air-flow velocity distribution is uniform and the flow direction is consistent. To improve simulation efficiency, the fan structure was removed from the wind tunnel underhood flow field simulation model and uniform velocity was used as the input condition for the inlet. The boundary condition is set to the mass-flow inlet without specifying the pressure outlet. The inlet flow rate is set at 1.72 kg/s based on the actual engine room intake air volume. Install a real model of the interior components of the vehicle underhood in the wind tunnel test section, and its geometric model is shown in fig. 19(a).



Figure 19. Geometric model geometric model of components mounted in the simulate underhood environment air duct; (a) simulate underhood environment air duct and (b) real vehicle

### Components mounted in the actual vehicle underhood model

Select a certain actual vehicle and establish a full detail model of the entire vehicle, which includes the exterior design of the vehicle body, detailed components in the engine compartment, floor and chassis components, *etc.*, as shown in fig. 19(b).

### Comparison and analysis of simulation results

#### Analysis of the calculation results of the inlet air volume of the heat exchanger

The intake air volume of the heat exchanger directly affects the heat transfer performance of the heat management system. Through simulation calculations, the intake air volume of the heat exchanger in the Simulate underhood environment air duct and the actual vehicle underhood are obtained.

	Simulate underhood environment air duct	Vehicle	Error [%]
Radiator inlet air volume [kgs <sup>-1</sup> ]	0.5011	0.5013	0.039
Inlet air volume of condenser [kgs <sup>-1</sup> ]	0.8743	0.8928	2.116

#### Table 2. Comparison of heat exchanger inlet air volume

As shown in tab. 2, the intake air volume of the radiator and condenser in the Simulate underhood environment air duct environment is 0.5011 kg/s and 0.8743 kg/s, respectively. The intake air volume of the radiator and condenser in the actual engine compartment environment is 0.5013 kg/s and 0.8928 kg/s, respectively. The error in the intake air volume of the radiator is 0.039%, and the error in the intake air volume of the condenser is 2.116%.

#### Analysis of calculation results of flow field around key components

As mentioned earlier, the intake air volume of the heat exchanger directly affects the heat transfer performance of the thermal management system and is a key component in the underhood. In addition, the electric drive assembly, as the main heat source in the underhood, is also a key focus of the thermal management system. Therefore, this article focuses on comparing the surface velocity distribution of the radiator and condenser, as well as the flow field distribution around the electric drive assembly. As shown in fig. 20, due to the shape of the grille in front of the radiator, the flow velocity in the middle area of the radiator is extremely low, and the flow velocity on the left and right sides is high, with a velocity distribution between 8 m/s and 10 m/s. The radiator is located in the simulate underhood environment air duct, and its surface velocity distribution is highly consistent with the surface velocity distribution of the radiator in the actual engine compartment environment.



Figure 20. Velocity distribution of radiator; (a) air duct environment and (b) real vehicle environment

As shown in fig. 21, the condenser is located in both the simulate underhood environment air duct environment and the actual engine compartment environment, with a speed range of between 6 m/s and 9 m/s, and its surface velocity distribution trend remains basically consistent.



Figure 21. Velocity distribution of condenser; (a) air duct environment and (b) real vehicle environment

As shown in fig. 22, the electric drive assembly is installed in the simulate underhood environment air duct and the actual vehicle underhood, with a velocity distribution range of 4 m/s to 8 m/s, and the velocity distribution trend remains consistent in the simulate underhood environment air duct and the actual vehicle underhood environment.



Figure 22. Velocity distribution of electric drive assembly; (a) air duct environment and (b) real vehicle environment

Based on the aforementioned analysis, it can be concluded that the flow field environment provided by the Simulate underhood environment air duct is highly consistent with the environment inside the actual vehicle underhood, which can ensure that the system's heat generation and dissipation performance is close to the real situation and meet the requirements of thermal management system integration testing.

### Conclusions

- This paper, based on the working principles of key components in the thermal management system, identifies the environmental conditions that the testing system needs to provide for the test subject and proposes a novel integrated thermal management testing system configuration. To enhance the uniformity of the velocity distribution at the entrance of the test section, simulations were conducted to optimize different structural dimensions of the honeycomb. Ultimately, the width of the honeycomb was determined to be 10 mm, with a length of 150 mm.
- Comparative analysis using simulation methods was performed to assess the degree of alignment between the flow field within the components in the test section and the flow field inside a real vehicle cabin. The results show a high degree of alignment between the flow field of the test subject within the system and the flow field inside a real vehicle cabin. The overall intake air-flow error for the radiator was 0.039%, and the condenser intake air-flow error was 2.116%. Furthermore, the flow field distribution around the radiator, condenser, and electric drive assembly exhibited good consistency. Additionally, this system can accommodate actual components while retaining their real relative positional relationships, meeting the requirements of integrated testing for thermal management systems.
- This testing system is primarily used to simulate the real operating conditions of a vehicle at the system level, enabling the execution of real transient condition tests. During transient vehicle operation, the cabin's intake air-flow fluctuates continuously. Currently, this paper has not delved into an in-depth study of how the test stand can provide consistent and precise transient air-flow rates that match the operation of a real vehicle. This will be a focus of future research.

### Acknowledgment

This research is funded by Youth Foundation of National Natural Science Foundation of China (No. 52102458) and Major Science and Technology Projects in Jilin Province and Changchun City (Grant No. 20210301023GX, 20220301010GX).

#### References

- [1] Jia, M. X., *et al.*, Progress and Perspectives of Integrated Thermal Management Systems in PEM Fuel Cell Vehicles, A review, *Renewable and Sustainable Energy Reviews*, *3* (2022), 155
- [2] Feng, L., The Investigation on Match and Design of Commercial Vehicle Cooling Module, Ph. D. thesis, Zhe Jiang University, Zhejiang, China, 2010, pp. 45-56
- [3] Zhao, Y. B., Study on Thermodynamic Model of New Energy Vehicle Thermal Management System Testbed, Science and Technology Information, 19 (2021), 33, pp. 58-61
- [4] Wang, W. M., *et al.*, Research on Development Technologies of Thermal Management System with Heat Pump for Battery Electric Vehicles, *Chinese Journal of Automotive Engineering*, *11* (2021), 6, pp. 434-441
- [5] Xue, S., Development and Research of Test Platform for Electric Vehicle Thermal Systems Cooperative Management, Ph. D. thesis, Zhe Jiang University, Zhejiang, China, 2012, pp. 37-44
- [6] Ma, J., *et al.*, Experimental Study on the Performance of Vehicle Integrated Thermal Management System for Pure Electric Vehicles, *Energy Conversion and Management*, *1* (2022), 2
- [7] Li, W. B., et al., Coupled Thermal Electrochemical Model of 3-D Lithiumion Battery, Chinese Journal of Power Sources, 40 (2016), 7, pp. 1362-1366
- [8] Fu, Y., Research and Optimization on Thermal Management of Engine Compartment of an Agricultural Machinery, Ji Lin University, Jilin, China, 2021, pp. 34-42
- [9] Geng, Y. L., Research on Heat Dissipation of Commercial Vehicle Engine Compartment Based on 1-D/3-D co-Simulation, Ji Lin University, Jilin, China, 2021, pp. 59-61
- [10] Ye, L., et al., Establishment and Analysis of a Simulation Model for an Electric Vehicle's Thermal Load (in Chinese), The Journal of New Industrialization, 9 (2019), 5, pp. 70-79

- [11] Liang, S. C., Study on Effect of Temperatureon Resistance of Li-ion Battery, Guangzhou Chemical Industry, 46 (2018), 15, pp. 66-67
- [12] Liu, X. T., et al., State-of-Power Estimation for Li-ion Battery Considering the Effect of Temperature, Transactions of China Electro Technical Society, 31 (2016), 13, 156
- [13] Tete, P. R., et al., Developments in Battery Thermal Management Systems for Electric Vehicles: A Technical Review, Journal of Energy Storage, 3 (2021), 35
- [14] Cao, S. Y., et al., Reproduction of Wind Velocity History in a Multiple Fan Wind Tunnel, Wind Eng. Ind. Aerodyn, 90 (2002), 12-15, pp. 1719-1729
- [15] Shao, P. L., et al., Active Simulation of Transient Wind Field in a Multiple-Fan Wind Tunnel via Deep Reinforcement Learning, Wind Eng. Ind. Aerodyn, 147 (2021), 9
- [16] Hu, X., et al., Low Wind Drag Numerical Research on Fenders and Wheels of Car Model, Applied Mechanics and Materials, 397-400 (2013), Sept., pp. 599-602
- [17] Zhang, Y., et al., Transient Simulation Research on Automobile Aerodynamic Lift Based on LBM Method, Mechanics of Fluids and Gases, 23 (2017), 6
- [18] Mason, M. S., et al., Numerical Simulation of Downburst Winds, Wind Eng, Ind. Aerodyn., 34 (2009), 12, pp. 523-539
- [19] Teunissen, H. W., Simulation of the Planetary Boundary-Layer in a Multiple-Jet Wind Tunnel, Atmos. Environ, 9 (1975), 2, pp. 145-174
- [20] Bienkiewicz, B., et al., Active Modelling of Large-Scale Turbulence, Wind Eng, Ind. Aerodyn., 13 (1983), 1-3, pp. 465-475
- [21] Kobayashii, H., et al., Active Generation of Wind Gust in a 2-D Wind Tunnel, Wind Eng. Ind. Aerodyn., 42 (1992), 1-3, pp. 959-970

Paper submitted: September 26, 2023 Paper revised: September 30, 2023 Paper accepted: November 8, 2023