

Study of Hot Air Recirculation in Engine Compartment of a Newly Developed Sport Utility Vehicle

Huojie Shi,¹ R.H. Rao,² J. Chen,³ and Z.L. Zheng⁴

¹Fuzhou Polytechnic, Mechatronics Engineering, China

²Fujian Normal University, Computer and Cyber Security, China

³Fujian Polytechnic Normal University, Mechatronics Engineering, China

⁴Southeast (Fujian) Automobile Industry Co., Experimental Testing Department, China

Abstract

During idling tests of a newly developed sport utility vehicle (SUV) under tropical high-temperature conditions, the condenser surface temperature exceeded the allowable range, degrading the air-conditioning system's cooling performance. In this study, a three-dimensional computational fluid dynamics (CFD) model of the engine compartment flow field was established using STAR-CCM+. The results reveal that under idling conditions, the kinetic energy of hot air passing through the cooling module was insufficient to overcome the pressure difference between the front and rear sections, thus inducing hot air recirculation (HAR) and increasing the overall compartment temperature. To address the unfavorable flow field characteristics, four structural improvements were proposed and simulated for both flow and temperature fields. Through comparative analysis, the optimal scheme was determined: installing a flow guide baffle above the engine. Simulation results show that the airflow velocity above and below the engine increased by 54% and 71%, respectively, and HAR was effectively suppressed. The optimal scheme was further validated under real-vehicle idle conditions, and the temperature deviation between simulation and measurement was within 2%, confirming the reliability of the numerical model. In addition, the optimized scheme was verified under three typical harsh driving conditions, including hill climbing, high-speed climbing, and high-speed driving. Both simulation and test results indicate that the scheme significantly enhances airflow velocity in the engine compartment, with temperature errors maintained within 5%. The present study effectively mitigates the compartment temperature rise caused by HAR, and the proposed baffle scheme provides a feasible solution for the thermal management design of new SUVs under both idle and severe driving conditions.

History

Received: 08 Jul 2025
Revised: 05 May 2026
Accepted: 01 Jun 2026
e-Available: 17 Jun 2026

Keywords

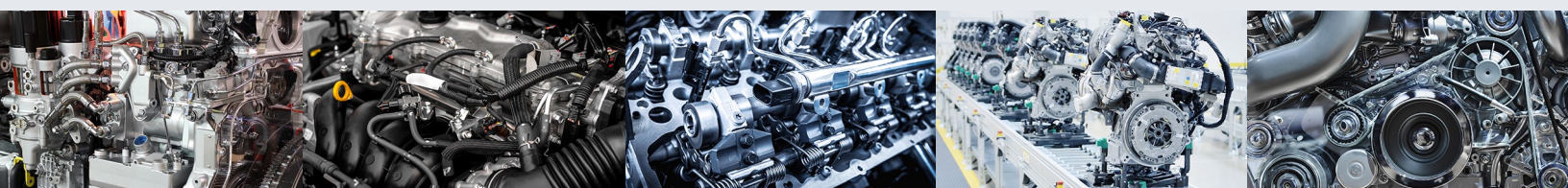
Hot air recirculation, Idle condition, STAR-CCM+, Engine compartment, Driving condition

Citation

Shi, H., Rao, R., Chen, J., and Zheng, Z., "Study of Hot Air Recirculation in Engine Compartment of a Newly Developed Sport Utility Vehicle," *SAE Int. J. Engines* 19(3):241-256, 2026, doi:10.4271/03-19-03-0013.

ISSN: 1946-3936
e-ISSN: 1946-3944

© 2026 Fuzhou Polytechnic. Published by SAE International. This Open Access article is published under the terms of the Creative Commons Attribution Non-Commercial, No Derivatives License (<http://creativecommons.org/licenses/by-nc-nd/4.0/>), which permits use, distribution, and reproduction in any medium, provided that the use is non-commercial, that no modifications or adaptations are made, and that the original author(s) and the source are credited.



Introduction

The engine compartment of an automobile is a semi-enclosed space with a compact structural arrangement, which makes heat dissipation difficult. During the R&D process, unreasonable layout of its components and improper matching of the cooling system often hinder the circulation of cooling air, resulting in hot air recirculation (HAR). Existing studies have clearly pointed out that uneven distribution of airflow is the key inducement of HAR, and the coupling effect of convective and radiative heat transfer will further aggravate abnormal temperature inside the compartment [1]. This phenomenon causes the cooling system to suck in heated stagnant air, which further deteriorates the heat dissipation performance inside the compartment, resulting in local or overall temperature exceeding the standard for the engine and the compartment, and seriously affecting the safe and stable operation of the vehicle. Reference [2] showed that the heat dissipation efficiency of the engine compartment directly determines the dynamic performance, fuel economy, lubricating oil durability, and long-term reliability of the automobile, which is a core technical problem that needs to be focused on during the whole vehicle R&D process.

Relevant studies have addressed three major aspects: structural optimization, parameter control, simulation and test techniques. In terms of structural optimization, Reference [3] adopted a full-wrap design of heat insulation plates, which can effectively block thermal radiation from high-temperature sources such as the exhaust system, significantly reduce the risk of thermal damage to surrounding components, and provide ideas for the design of the improvement scheme in this article. Reference [4] classified thermal management architectures for different powertrain engine compartments and identified that thermal component coupling and chaotic airflow within confined spaces are dominant causes of thermal failures. It also quantified the applicability of numerical simulations in full-field thermal analysis, providing solid theoretical support for this work. In terms of parameter control, Reference [5] focused on the impact of key parameters such as the opening area of the engine compartment and the inclination angle of the intake grille on the thermal performance of the cooling system and found that reasonable parameter matching can effectively improve the circulation efficiency of cooling air and reduce HAR. Reference [6] analyzed the coupling mechanism of convective heat transfer and radiative heat transfer in the engine compartment and clarified the influence law of airflow velocity and engine power on the thermal environment inside the compartment. Studies have shown that the matching characteristics between the fan and radiator exert a considerable influence on engine compartment HAR, and scientific and reasonable matching can effectively improve fan efficiency and reduce the risk of HAR

[7]. In terms of simulation, with the continuous development of computational fluid dynamics (CFD) technology, computer simulation has become the mainstream method for the research of engine compartment thermal management. It has the advantages of low cost, short cycle, and strong repeatability, and can accurately simulate the distribution characteristics of the flow field and temperature field inside the compartment [8–10]. References [11, 12] adopted a combined 1D and 3D simulation method, which balances simulation efficiency and accuracy, and provided an efficient way for thermal management analysis under complex working conditions. The virtual test of automotive accessories and the integrated CFD simulation method of multiple thermal systems proposed in [13] improved the simulation accuracy. The CFD simulation and fan parameter optimization method for the engine compartment of compact off-road vehicles proposed in [14] improved fan efficiency. Reference [15] further proposed a 1D/3D collaborative simulation method for vehicle thermal management systems. The thermodynamic characteristics of the engine cooling and air-conditioning systems were characterized by the 1D model, and the structure and flow field distribution of the engine compartment were described by combining with the 3D model, clarifying the coupling effect of the cooling system and the air-conditioning system and the impact of HAR on thermal management performance. These research methods fully verified the rationality and feasibility of using STAR-CCM+ software to establish the flow field model in this article and provide important method references for the HAR simulation analysis of this study [7, 16, 17]. In the field of experimental research on engine compartment thermal management, relevant studies have focused on test point layout, working condition design, and analysis of thermal behavior laws, providing solid support for simulation model calibration and optimization scheme verification. Reference [18] studied the measurement methods and physical analysis of temperature and heat flux density inside the compartment, proposed a method for separating convective heat transfer and radiative heat flux, and solved the problem that traditional heat flux measurement cannot distinguish between heat transfer types. Reference [19] carried out thermal tests of passenger car engine compartments in a wind tunnel environment, and systematically studied the variation laws of temperature and heat flux density of various components in the compartment under different engine powers and airflow velocities by arranging multiple sets of thermocouples. Reference [20] conducted research on the HAR problem of the engine compartment and analyzed the formation mechanism of HAR under complex working conditions by test method.

Although existing studies have achieved fruitful results, prominent limitations still exist for the new SUV investigated in this article, which are specifically reflected

in three aspects: first, most studies focus on traditional models such as sedans, with limited research on HAR in SUV engine compartments; second, existing studies mostly focus on a single idle or high-speed working condition, and there is insufficient research under harsh and complex working conditions such as idling in tropical high-temperature environments, climbing, and high-speed driving, which is difficult to meet the diverse needs of SUV; third, some studies only propose improvement schemes through simulation analysis, lacking real-vehicle test, and thus difficult to be directly applied to practice [17, 21].

During the idling test of the new SUV in the early development stage, it was found that the surface temperature of the front condenser in the engine compartment exceeded the allowable limit, reaching a maximum of 98°C. This issue significantly degraded the cooling performance of the air-conditioning system and hindered the normal operation of the engine. Preliminary analysis indicated that this phenomenon was caused by HAR. Under idling conditions, the vehicle speed was zero, and the external airflow velocity was extremely low. The cooling system relied mainly on the cooling fan to drive the surrounding air. This induced flow included both the low-temperature fresh air entering through the front intake grille and the heated recirculated air inside the compartment [22]. As a result, the airflow velocity in the rear region of the compartment was insufficient, and the accumulated hot air couldn't be discharged in a timely manner, leading to obvious HAR [23, 24].

To address the above problems, the following investigations were performed:

1. A 3D CFD simulation model of the new SUV was established using STAR-CCM+ software to accurately characterize the internal airflow and thermal environment, and clarify the formation mechanism of HAR under idle conditions.
2. Multiple improvement schemes were designed for the unreasonable flow field distribution inside the engine compartment, and simulation analyses were carried out for each scheme to screen out the optimal improvement scheme with good improvement effect, low cost, and simple technical implementation.
3. Real-vehicle tests were performed to verify the effectiveness of the optimal improvement scheme, the reliability of which was confirmed by comparing the steady-state temperatures at key monitoring points in the engine compartment with the simulation results.
4. The research was extended to typical harsh driving conditions such as hill climbing, high-speed climbing, and high-speed driving. Simulation and test verification under these conditions were carried out to avoid the limitations of research conclusions caused by focusing only on a single working condition.

1. Materials and Methods

1.1. Materials

The experimental setup consisted of an engine flow deflector and acoustic insulation foam for the engine hood.

K-type thermocouples were employed to measure temperatures at key locations, including the condenser surface, behind the fan, and above and below the engine.

1.2. Computational Hardware

The simulation model in this work featured a large computational domain and an ultrahigh total mesh count, so high-performance computing servers were required to meet the computational demands for large-scale mesh preprocessing, numerical solution, and post-processing. The specific configuration was as follows: The FusionServer 2288H V5 was equipped with an Intel Xeon Silver 4210R CPU (main frequency: 2.4 GHz) and 16 GB high-bandwidth DDR4 RAM. Its storage system adopted a 2 TB SATA enterprise-class hard disk.

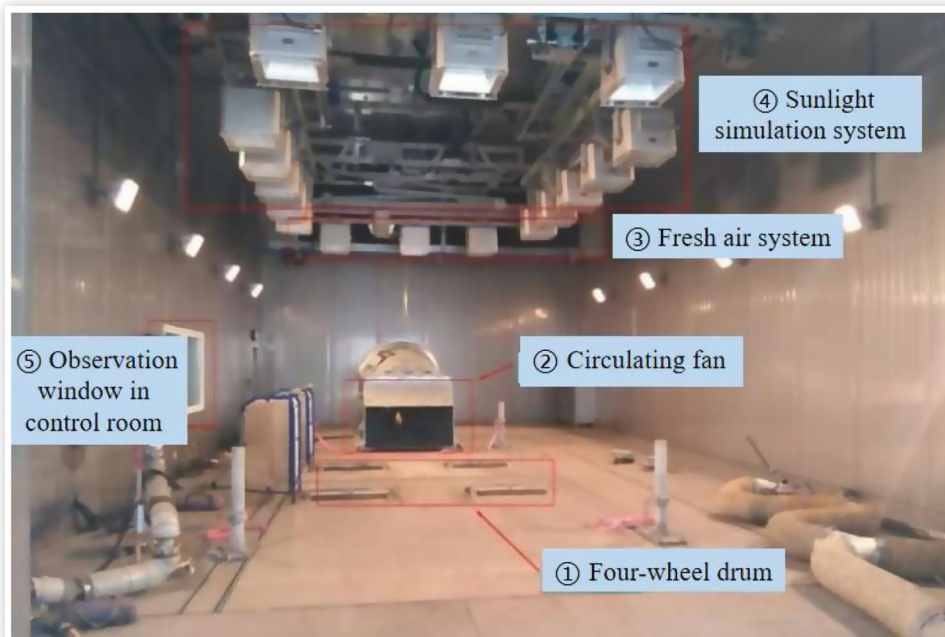
1.3. Test Equipment

The key equipment for the real-vehicle tests was the Climate Environment Laboratory (CEL), which can simulate high/low temperatures, solar radiation, humidity, wind resistance, and road gradient conditions (Figure 1). It provides a standardized test environment for the measurement of the engine compartment temperature field under harsh conditions. During vehicle operation, it is difficult to measure the convective airflow on the windward side and its heat exchange with internal and external environments directly [3]. However, tests conducted in the CEL enable well-controlled conditions and reliable data acquisition, which is a common experimental method in the automotive industry [9].

1.4. Research Method

In the early stage of product design, adopting simulation methods to evaluate vehicle thermal management risks can shorten the development cycle and reduce R&D costs, which makes this approach widely applied by major automakers and research institutions. The use of the finite element method (FEM) to analyze the thermal management of the engine compartment is a low cost and efficient means of automotive design and development. It can intuitively obtain the required flow field and temperature distribution diagrams, thereby quickly locating problems, reasonably optimizing the internal structure, and avoiding excessive temperature caused by HAR.

1.4.1. Simplification and Establishment of Geometric Model To ensure accurate simulation of the heat

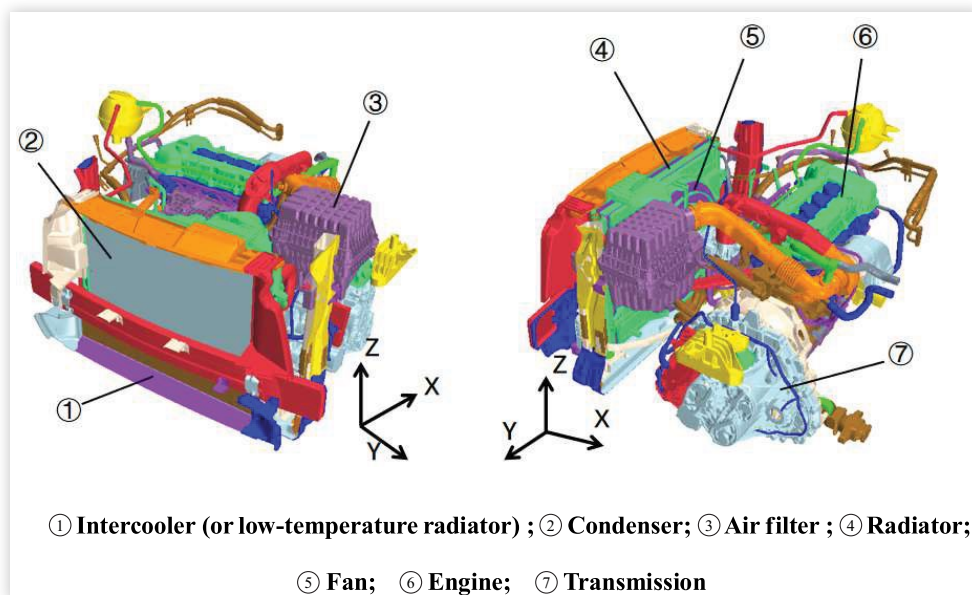
FIGURE 1 Diagram of Climate Environment Laboratory (CEL).

© Fuzhou Polytechnic

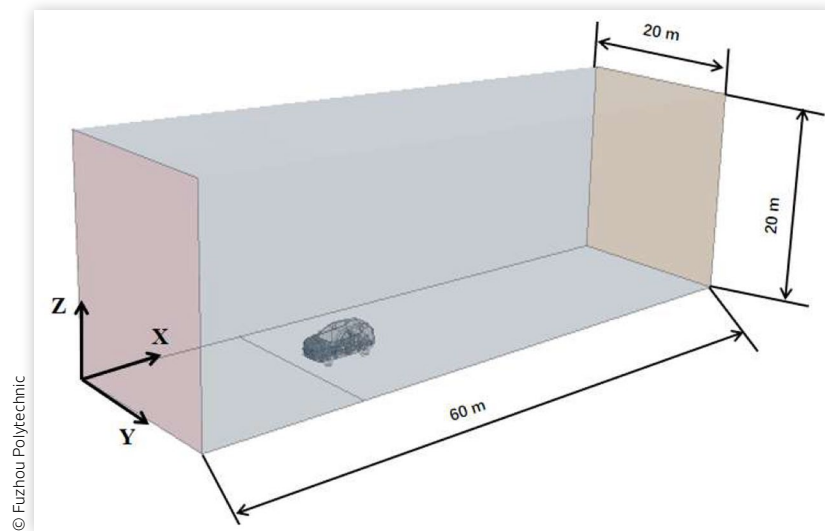
dissipation inside the engine compartment for subsequent performance analysis and evaluation, a 1:1 full-scale model was used for simulation. All components affecting the internal airflow distribution must be retained, so the whole vehicle modeling is indispensable, including the outer surface of the vehicle body, chassis system, intake grille, cooling module, and various pipelines and wiring harness components inside the compartment; the heat exchanger cooling module includes a radiator, condenser, fan, and low-temperature radiator, and the system layout

is shown in Figure 2. The front cooling module of this model adopted a condenser–radiator–fan arrangement; specifically, the three components were placed in sequence, and the low-temperature radiator was located at the bottom of the condenser.

To simulate the real environment, it was assumed that the vehicle model was placed in an open space for calculation, and a sufficiently large hexahedral space was specifically used for simulation, as shown in Figure 3. On the premise of meeting engineering accuracy, noncritical

FIGURE 2 The components in the automotive engine compartment.

© Fuzhou Polytechnic

FIGURE 3 Three-dimensional diagram of the flow field calculation in STAR-CCM+.

components such as small-diameter pipelines, wiring harnesses, and nuts were ignored.

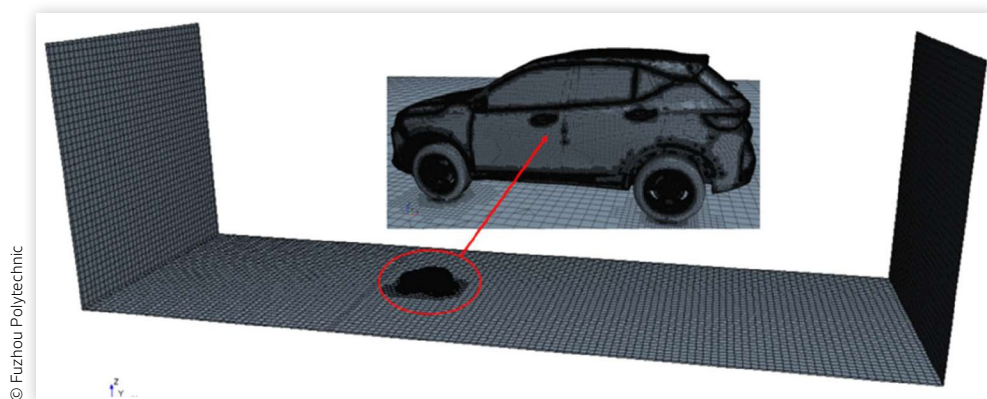
1.4.2. Computational Mesh Division and Mesh Independence Verification The model was face meshed in HyperMesh and then imported into T-Grid, where all Trimmer unstructured body meshes were generated in the computational domain. Mesh refinement was applied to the areas near the body and inside the engine compartment where the flow varies drastically to improve the accuracy of the calculation. For the far areas that had less influence on the calculation results, a larger mesh size was used to save computational resources, as shown in Figure 4. The size of the computational domain was large enough to ensure smooth airflow and reduce the influence on the flow field in the engine compartment to ensure the accuracy of the simulation results [25]. The wind tunnel area was set to be 60 m long, 20 m wide, and 20 m high. Two boundary layers were defined for the air domain, with a growth rate of 1.3 and a total thickness of 2 mm. The final mesh count reached 53

million. The mesh independence verification involved the results of simulation calculation and real-vehicle test, so it was arranged in Section 2.3.

1.4.3. Governing Equations Since the airflow velocity within the engine compartment is far lower than the speed of sound, the density variation remains negligible. Accordingly, the fluid in the computational domain can be defined as a viscous, constant-property and incompressible fluid [15]. The complex structure of the engine compartment is easy to cause airflow separation, so the realizable $k-\epsilon$ model is chosen to simulate the external flow of the car. Due to the adaptation of boundary layer and shear flow, the model is widely used in industry to analyze the turbulent motion, which needs to meet the following basic equations of flow and heat transfer [6, 8].

Mass Conservation Equations:

$$\frac{\partial}{\partial x_i}(\rho u_i) = 0 \quad (1)$$

FIGURE 4 The mesh division diagram of simulation model.

where ρ is the density of air with a value of 1.184 kg/m^3 ; u_i is the component of the air velocity in the i -direction.

Conservation of Momentum Equation:

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(\rho u_i u_j)}{\partial x_j} + \frac{\partial p}{\partial x_i} = \frac{\partial \sigma_{ij}}{\partial x_j} + g_i + S_i \quad (2)$$

where u_j is the component of air velocity along direction j ; p is the air static pressure; σ_{ij} is the stress vector; g_i is the component of gravity in direction i ; and S_i is the generalized source term of the equation for the conservation of momentum in the 3 directions.

Energy Conservation Equation:

$$\frac{\partial(\rho T)}{\partial t} + \frac{\partial(\rho u_i T)}{\partial x_i} = \frac{\partial}{\partial x_i} (k + k_i) \frac{\partial T}{\partial x_i} + S_h \quad (3)$$

where T is the air temperature; k is the molecular conductivity; k_i is the conductivity due to turbulent transfer; and S_h is the defined volume element.

1.4.4. Setting of Simulation Boundary Conditions

1.4.4.1. Boundary Condition Settings at Idle. The front end of the numerical wind tunnel was set as a velocity inlet with the ambient temperature assigned, with the turbulence intensity set to 1% and the dissipation rate to 10; the other end was set as a pressure outlet (0 MPa gauge) and also assigned ambient temperature. A porous medium model was used to model the condenser and radiator, and their coefficients of inertial resistance and viscous resistance were provided by the supporting manufacturer; the Multiple Reference Frame method was used to model the independent area of the fan [8, 21]. The air-conditioning was set to operate at the maximum airflow rate and the minimum temperature, while the heat transfer rates of the condenser and radiator were defined as constant values [26]. The surfaces of heat-generating components in the engine compartment were defined as temperature boundary conditions, whereas the surfaces of other components were set as adiabatic boundaries. The relevant parameters are presented in Table 1, and additional simulation settings can be found in [9].

1.4.4.2. Simulation Boundary Settings for Driving Modes. In the process of designing cooling modules, automakers usually select three typical working conditions—idling, climbing, and high-speed driving—to evaluate the impact of the heat dissipation performance of each heat exchanger component on the engine compartment [5, 9]. In this article, the climbing condition was further divided into hill climbing and high-speed climbing conditions. The specific working conditions and engine parameters are shown in Table 2.

TABLE 1 Simulation working condition settings.

Parameters	Idle working condition
Speed of the vehicle (km/h)	0
Slope (%)	0
Ambient temperature (°C)	50
Ambient humidity (%)	50
Engine speed (r/min)	900
Engine torque (N·m)	15
The inertial resistance coefficient of radiator (kg/m^4)	138.9
The viscous resistance coefficient of radiator ($\text{kg}/(\text{m}^3 \cdot \text{s})$)	1744.5
The inertial resistance coefficient of condenser (kg/m^4)	240
The viscous resistance coefficient of condenser ($\text{kg}/(\text{m}^3 \cdot \text{s})$)	734.5

© Fuzhou Polytechnic

1.4.5. Experimental Procedures

1.4.5.1. Sensor Placement. To verify the effectiveness of the HAR improvement scheme and calibrate the CFD simulation results, K-type thermocouples were arranged at multiple positions on the condenser (Figure 5), as well as behind the fan, above the engine, and below the engine (Figures 6–8) [27]. The condenser was the key monitoring object of the test, because its surface temperature exceeding the standard was the main problem caused by HAR. Arranging thermocouples at multiple positions could comprehensively capture its temperature distribution and avoided single-point measurement deviation; arranging sensors behind the fan was used to reflect the fan heat dissipation efficiency, supporting the calibration of the fan simulation model; arranging sensors above the engine directly monitored the improvement effect of the deflector baffle on the core area of HAR; arranging sensors below the engine could assist in judging the heat discharge efficiency and the HAR at the bottom of engine compartment.

TABLE 2 Parameter settings for three typical driving conditions.

Parameters	Hill climbing	High-speed climbing	High-speed driving
Speed of the vehicle (km/h)	60	110	150
Slope (%)	9	3	0
Ambient temperature (°C)	38	50	50
Ambient humidity (%)	50	50	50
Engine speed (r/min)	2399	3300	4138
Engine torque (N·m)	137	195	141

© Fuzhou Polytechnic

FIGURE 5 Temperature sensors were arranged on the surface of the condenser.

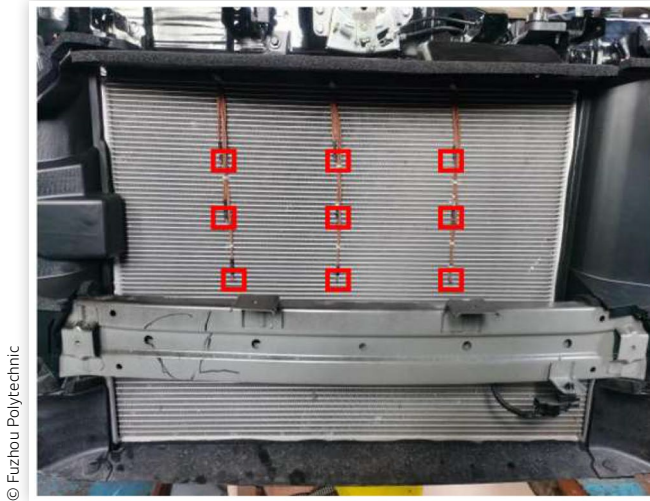


FIGURE 6 Temperature sensors were arranged behind the fan.

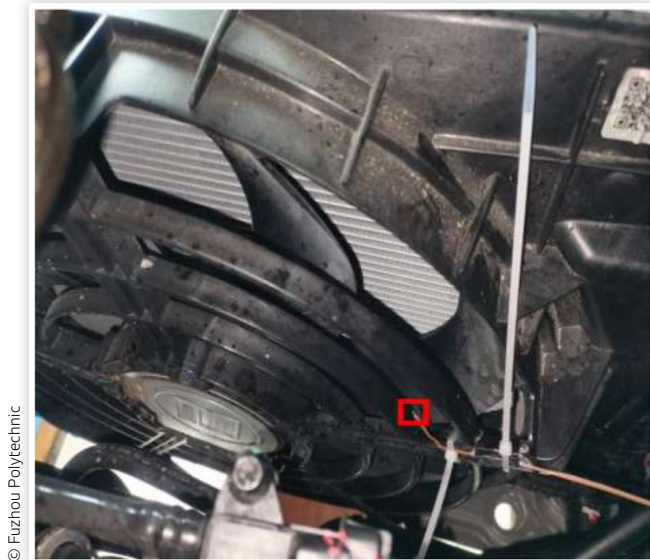


FIGURE 7 Temperature sensors were arranged above the engine.

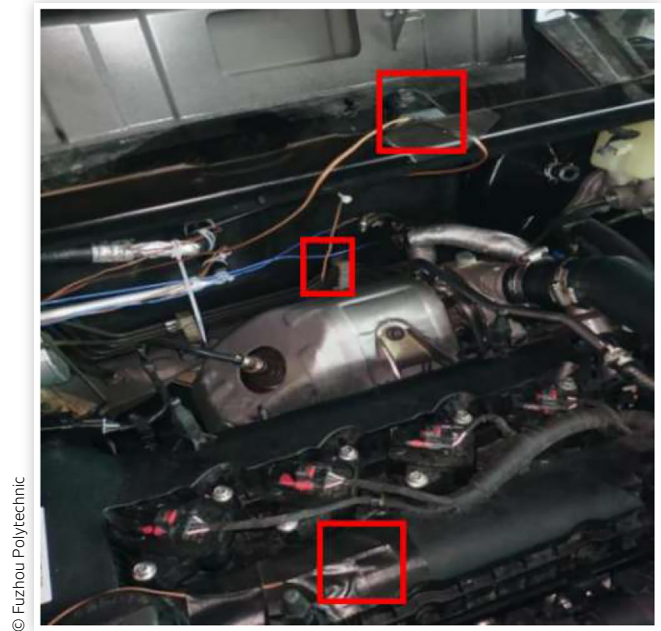
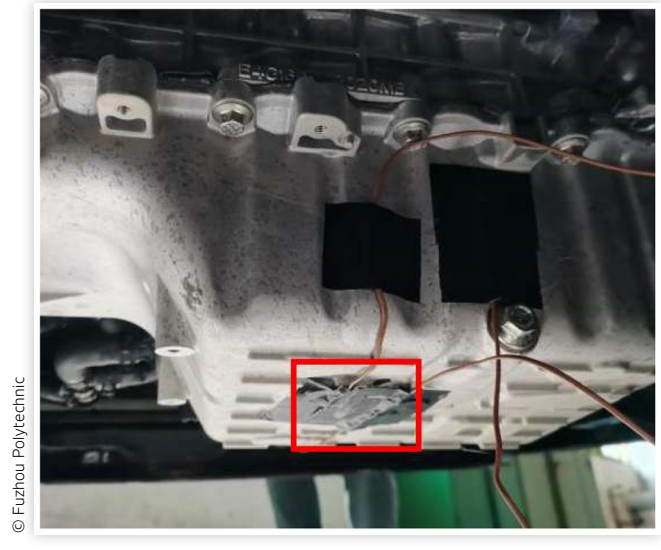


FIGURE 8 Temperature sensors were arranged below the engine.



1.4.5.2. Test Steps for Idle Conditions

1. Test vehicle was fixed on the four-wheel dynamometer drum of the CEL (Figure 9), and the engine, cooling system, and air-conditioning system were checked for no leakage; the oil and coolant levels met the standards.
2. K-type thermocouples were calibrated with a standard constant temperature bath, then pasted on the condenser surface, behind the fan, above and below the engine; the data acquisition instrument was connected to test that the signal was normal.
3. The CEL was set to an environment of $50 \pm 2^\circ\text{C}$, relative humidity $50 \pm 5\%$, and wind speed $\leq 1 \text{ m/s}$; the engine was started when the vehicle was cold, and idled at 900 r/min for preheating 30 min.
4. Air-conditioning was turned on to the maximum cooling mode; the sampling frequency of the data acquisition instrument was set to 1 Hz, and steady-state data were continuously collected for 60 min.
5. Upon completion of the test, all equipment was shut down. The sensors were arranged and recovered, and the measured data were processed and analyzed.

1.4.5.3. Test Steps for Driving Conditions

1. The test vehicle was slowly driven into the CEL and fixed on the four-wheel dynamometer drum; the vehicle body was fastened with special fixtures to prevent displacement; and the drum parameters were adjusted to match the vehicle wheelbase and track width.
2. As described earlier, K-type thermocouples were installed, the data acquisition instrument was

connected, and repeated debugging was performed to ensure stable signal transmission and no electromagnetic interference. The laboratory door was closed.

3. According to the three preset typical working conditions, the CEL ambient temperature, relative humidity, wind speed, and drum gradient were set sequentially. The wind tunnel and drum were then activated. After all environmental and equipment parameters had stabilized for 30 min, the subsequent procedure was performed [25].
4. The engine was started, and its speed and torque were adjusted to the preset values corresponding to the working conditions. The air-conditioning system was set to the maximum cooling mode and operated for 30 min until the vehicle thermal management system reached a stable state. Subsequently, the sampling frequency of the data acquisition instrument was set to 1 Hz, and the steady-state data were collected continuously for 60 min, during which the equipment operation status was monitored in real time.
5. After completing the test under a single operating condition, the engine, wind tunnel, and drum were shut down, while the CEL environmental parameters were kept unchanged. The vehicle was then cooled to the current ambient temperature before the test under the next operating condition was performed. The equipment operation status and data recording were continuously monitored throughout the entire test process.

2. Results and Discussion

2.1. Analysis of Simulation Results

After model building, boundary setting, and meshing were completed in accordance with the above instructions, calculations were carried out to obtain the flow field diagram at the Y-direction cross-section inside the engine compartment under idling conditions, as shown in Figure 10(a). The airflow inside the engine compartment was driven solely by the rotation of the cooling fan. This airflow included both the outside air drawn in through the front air inlet grille and the original air inside the engine compartment. Due to the low airflow rate in the engine compartment at idle and the irrational arrangement of its internal parts, HAR occurred [27]—particularly above the engine. Consequently, heat accumulated continuously in the compartment, which led to the overall temperature being maintained at a relatively high level [28]. By the time the fresh air entering the front grille reached the condenser surface, its temperature had risen to a higher level under the influence of the surrounding high-temperature airflow, as shown in Figure 10(b).

FIGURE 9 Diagram of vehicle thermal balance test in CEL.



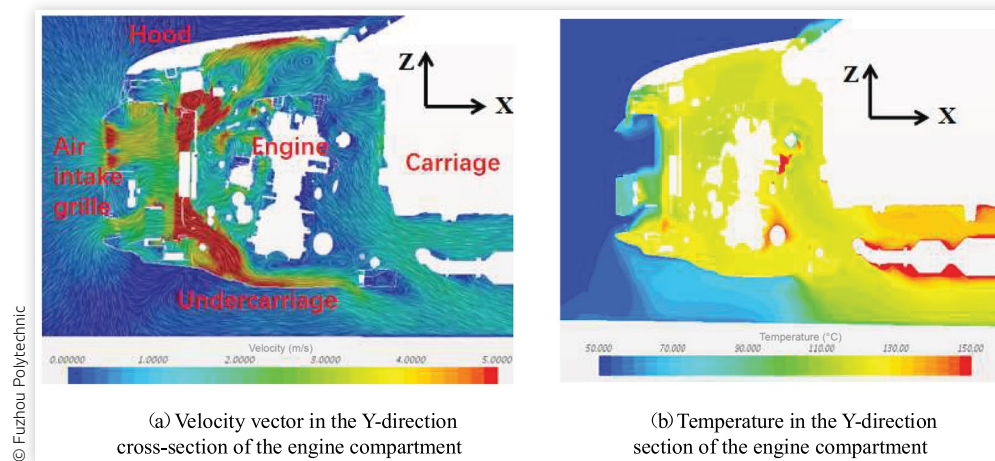
FIGURE 10 Internal flow field and temperature field of the engine compartment.

Figure 10(a) presents the velocity vector in the Y-direction cross-section of the engine compartment, while Figure 10(b) illustrates the corresponding temperature distribution in the same Y-direction cross-section. These two figures exhibit a negative correlation between the velocity and temperature distributions, and relevant studies on this negative correlation between temperature and flow velocity have been reported in [6, 9]. Therefore, due to space limitations, the temperature distribution plots for each analysis are omitted in the following sections.

2.2. Determination of the Optimal Scheme

2.2.1. Design Principles of Improvement Schemes The optimization scheme was targeted at regions with unreasonable airflow distribution, as reflected in the velocity vector cloud diagram in Figure 10(a). Since the newly developed SUV had entered the verification stage of the prototype vehicle, the layout of the power assembly and cooling module inside the engine compartment had been finalized. Considering the cost, technical complexity, space size, and improvement effect, the improvement plan mainly focused on the upper and lower spaces inside the compartment to make minor structural modifications, aimed at solving the HAR problems, and enhanced the airflow rate to reduce the compartment temperature [10].

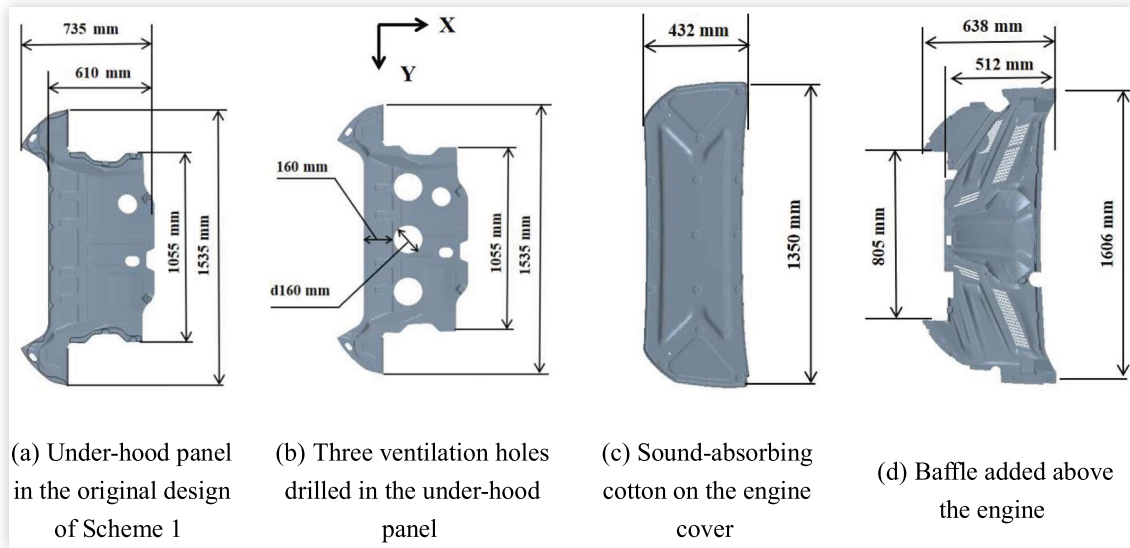
2.2.2. Specific Design of Four Improvement Schemes

- Scheme 1: Without adding, reducing, or changing any parts, only increased the fan speed from 2300 r/min to 3000 r/min, as shown in Figure 11(a).
- Scheme 2: Drilled three ventilation holes with a diameter of 160 mm in the under-hood panel in Figure 11(b).

- Scheme 3: Eliminated the under-hood panel and stretched the hood acoustic foam downward by 50 mm, as shown in Figure 11(c).
- Scheme 4: Added a baffle to the space above the engine as shown in Figure 11(d).

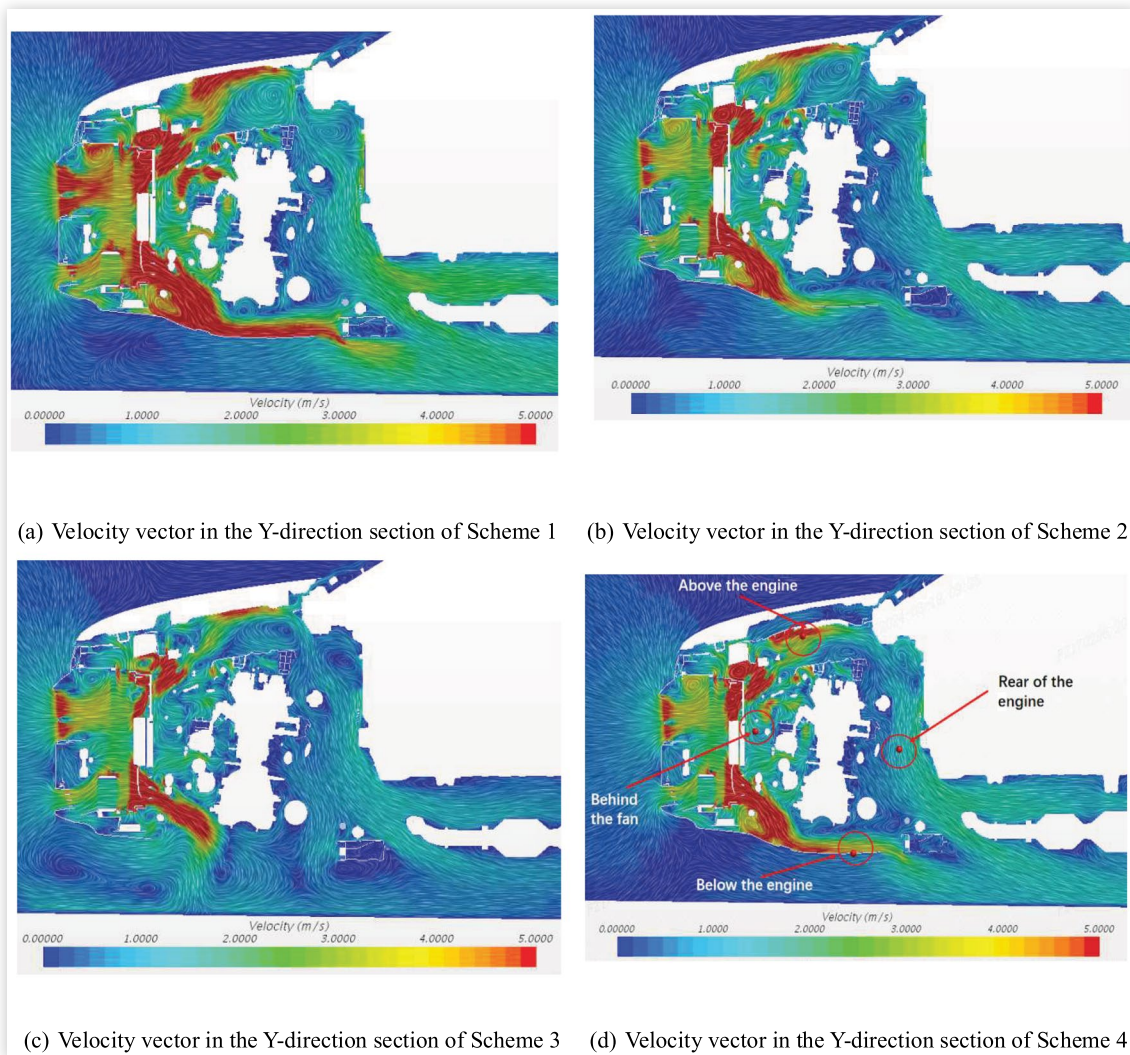
2.2.3. Comparative Analysis of Each Scheme The results of the above four schemes were obtained through simulation, and the airflow inside the engine compartment for each scheme is shown in Figure 12. Flow rate monitoring points were added behind the fan, above, below, and at the rear of the engine, respectively. The airflow rate of each monitoring point is displayed in Table 3. The results showed that the overall airflow rate inside the compartment increased significantly after increasing the fan speed in Scheme 1. However, the HAR in the upper part of the compartment still existed, and the phenomenon of poor airflow was not effectively solved, as shown in Figure 12(a). In Scheme 2, with the ventilation holes in the under-hood panel, not only did it fail to increase the airflow rate inside the compartment, but it also caused the fast airflow from the fan to be broken down at the holes and partially flow outside the compartment. The airflow rate inside the compartment deteriorated and was not as good as the original condition, as shown in Figure 12(b). Scheme 3 eliminated the under-hood panel, which resulted in a serious deterioration of the flow field in the compartment (almost stagnant). The upper hood acoustic foam, due to its small area, could not effectively improve the local flow field even when stretched downward, as shown in Figure 12(c). The addition of a baffle above the engine in Scheme 4 effectively improved the localized HAR phenomenon. The flow rates above and at the rear of the engine compartment were increased by 54% and 71%, respectively. The phenomenon of poor airflow discharged out of the compartment was significantly improved, and the HAR could be effectively eliminated, as shown in Figure 12(d).

FIGURE 11 Components used in the improvement schemes.



© Fuzhou Polytechnic

FIGURE 12 Velocity vector contour clouds of the Y-direction section for each scheme.



© Fuzhou Polytechnic

TABLE 3 Airflow velocity values at monitoring points (m/s).

Monitoring points	Original scheme	Scheme 1	Scheme 2	Scheme 3	Scheme 4
Behind the fan	1.25	1.46	1.26	0.86	1.27
Above the engine	2.95	3.69	2.18	2.88	4.53
Below the engine	3.57	6.75	2.45	0.35	4.07
Rear of the engine	0.66	1.21	0.67	0.73	1.13

Therefore, based on the simulation results, Scheme 4 demonstrates the best improvement performance.

2.3. Real-Vehicle Validation of the Optimal Scheme under Idle Conditions

To verify the actual vehicle improvement performance of the optimal scheme (Scheme 4), real-vehicle idle tests were conducted following the test methods and procedures presented in [Section 1.4.5](#), and the temperature values of monitoring points were compared with the simulation results, as shown in [Table 4](#) (for the case of 53 million meshes). The results showed that the errors between the two were within 2%, which validated the simulation accuracy and verified Scheme 4 as the optimal solution.

The analysis results of other mesh numbers were also listed in the table, which were the mesh independence verification analyses carried out during the modeling period. According to the theory of FEM, the mesh count correlates with the analysis results, but is not the core determinant of simulation accuracy. In fact, FEM is a discretization-based numerical method, and its solutions are approximate. An insufficient mesh count leads to large computational errors, while an excessive mesh count may reduce accuracy due to the accumulated errors from discretization and numerical calculation. Accordingly, the FEM requires multiple trial calculations to determine the appropriate mesh division, which is a basic step in using the method [3]. Mesh independence verification was conducted in this study, with the volume mesh number set to 30 million, 40 million, 50 million, and 53 million, respectively. Relevant test specifications are presented in [Section 1.4.5](#). The simulation results at three monitoring points are listed in [Table 4](#). These results were compared with the experimental data. It was observed that the simulation results with 53 million cells agreed best with the test data. Therefore, the mesh count of 53 million cells was adopted in this study.

2.4. Analysis of the Optimal Scheme under Driving Conditions

2.4.1. Simulation Results and Analysis of Driving Conditions

According to the above settings for climbing and high-speed conditions, STAR-CCM+ was used to simulate and obtain the flow field diagrams of the engine compartment under driving conditions for the original scheme and Scheme 4, as shown in [Figure 13](#). The airflow velocity values at each monitoring point are shown in [Table 5](#). The results show that under Scheme 4, the airflow above the engine was smoother, and the airflow velocity behind the engine was also increased simultaneously. Compared with the original scheme, the flow velocity above the engine was increased by 65% and that below the engine by 3% under hill climbing conditions; by 69% and 3.5%, respectively, under high-speed climbing conditions; and by 41.6% and 4.5%, respectively, under high-speed driving conditions.

2.4.2. Real-Vehicle Test Verification under Driving Conditions

Following the aforementioned test methods, experiments were carried out and relevant parameters were collected under harsh conditions in the CEL. Temperature data at each monitoring point were obtained, as presented in [Figures 14–16](#), and the corresponding average values are listed in [Table 6](#). A comparative analysis with numerical results shows great consistency between simulation and measurement, with overall errors within 5%. The simulation data agree well with the test data, further verifying the effectiveness of Scheme 4 under severe driving conditions. This scheme not only eliminates idle-state HAR, but also improves the engine compartment heat dissipation performance under driving conditions.

2.5. Discussion

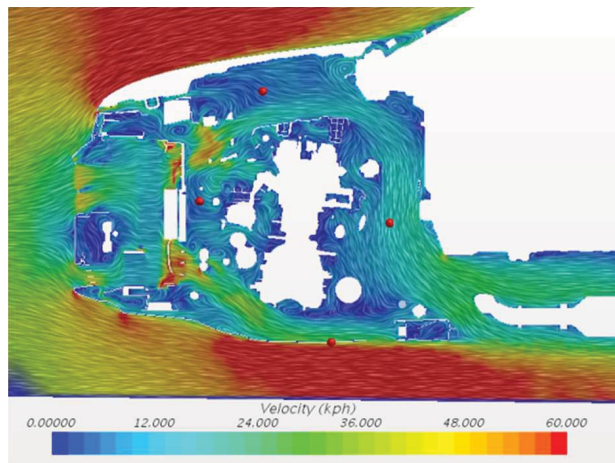
2.5.1. Refined Detailed Modeling of the Simulation Model and In-depth Verification

References [8, 9] have emphasized the impact of the accuracy of the heat

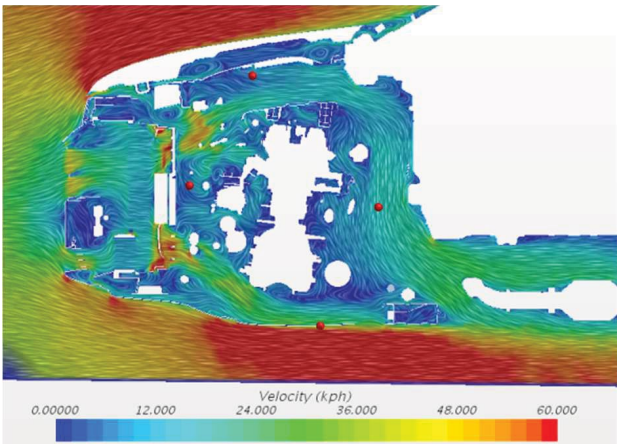
TABLE 4 Comparison of simulation results versus test values for monitor points using different numbers of grid divisions (°C).

Monitor points	Results of numerical calculation with different meshing strategies				Test value
	30 million	40 million	50 million	53 million	
Temperature below the engine	91.30	90.89	92.35	91.30	90.85
Temperature above the engine	89.48	89.95	91.25	88.30	89.30
Temperature behind the fan	90.30	89.67	91.13	87.90	86.45

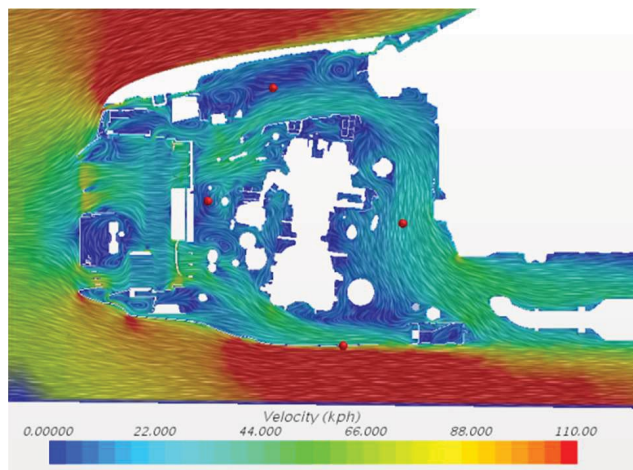
FIGURE 13 Y-direction cross-sectional velocity vectors of the engine compartment under different driving conditions: Original scheme vs. Scheme 4.



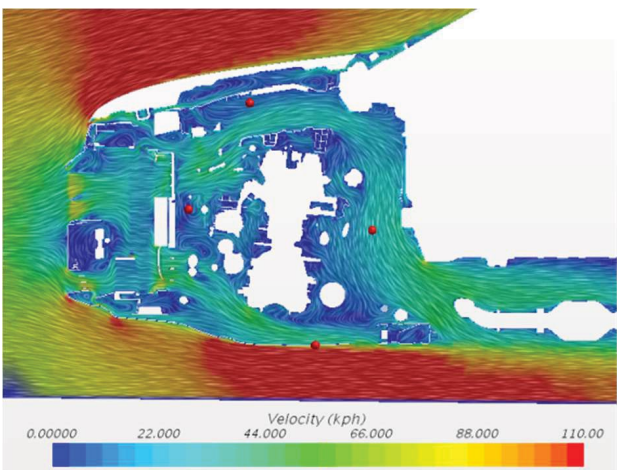
(a) Hill climbing condition in Original scheme



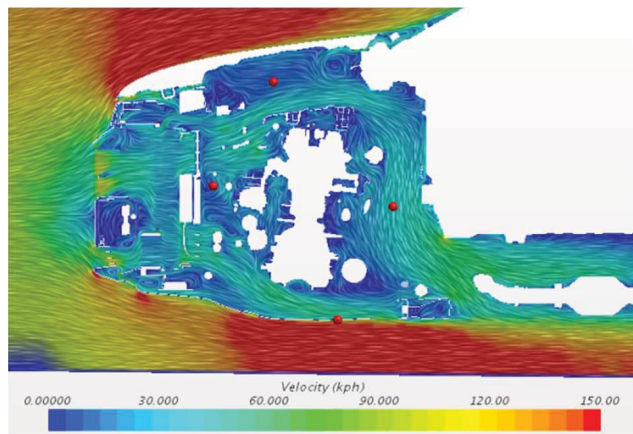
(b) Hill climbing condition in scheme 4



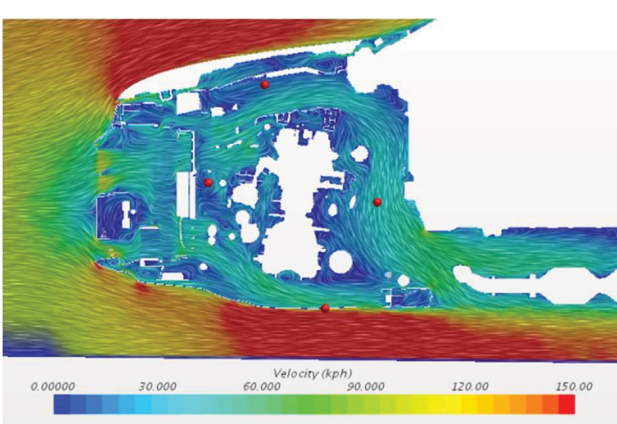
(c) High-speed climbing condition in Original scheme



(d) High-speed climbing condition in scheme 4



(e) High-speed driving condition in Original scheme



(f) High-speed driving condition in scheme 4

TABLE 5 Airflow velocity values at monitoring points in three typical driving conditions (m/s).

Monitoring point	Hill climbing		High-speed climbing		High-speed driving	
	Original scheme	Scheme 4	Original scheme	Scheme 4	Original scheme	Scheme 4
Behind the fan	1.71	1.73	1.19	1.14	3.65	4.07
Above the engine	1.01	1.67	1.22	2.06	2.26	3.2
Below the engine	7.99	8.01	11.05	9.97	14.1	14.82
Rear of the engine	4.56	4.71	8.16	8.45	10.86	11.35

FIGURE 14 Temperature at the monitoring points of hill climbing test (°C).

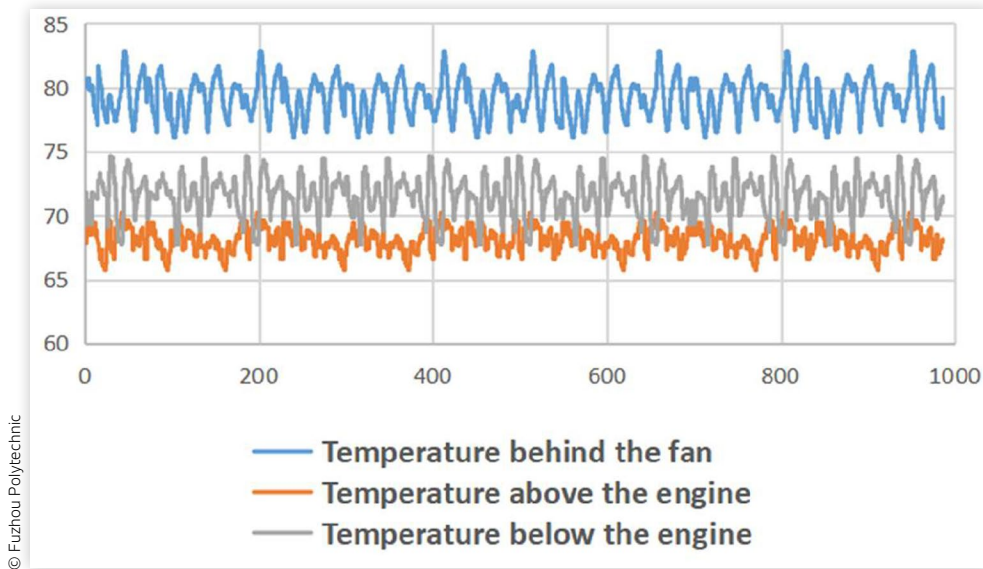


FIGURE 15 Temperature at the monitoring points of high-speed climbing test (°C).

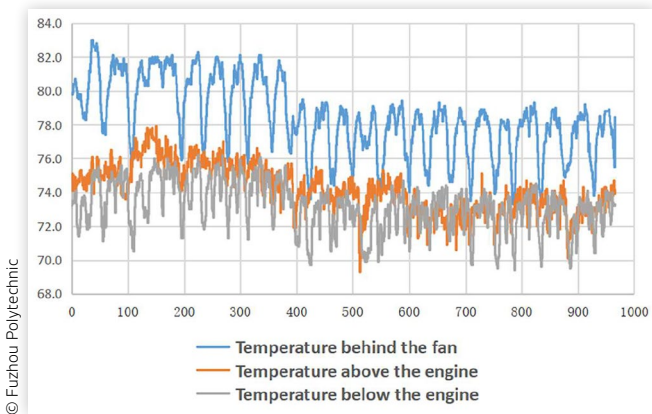


FIGURE 16 Temperature at monitoring points of high-speed driving test (°C).

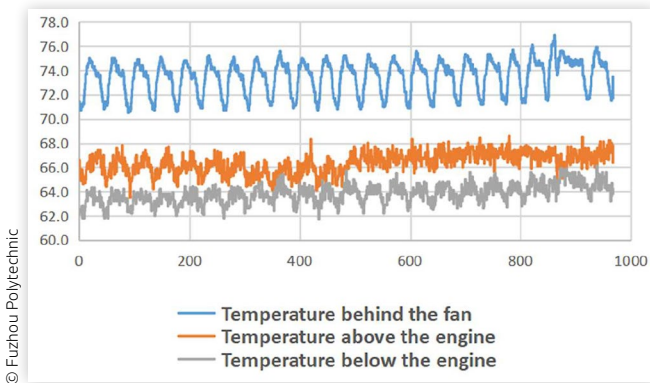


TABLE 6 Comparison of the results of vehicle walking test in Scheme 4 with the results of simulation.

Monitoring point	Hill climbing (60 kph)			High-speed climbing (110 kph)			High-speed driving (150 kph)		
	Test	Simulation	Percentage of temperature difference/simulation value	Test	Simulation	Percentage of temperature difference/simulation value	Test	Simulation	Percentage of temperature difference/simulation value
Behind the fan (°C)	79.2	80	1%	78.4	77	1.8%	73.5	71	3.5%
Above the engine (°C)	68.1	70	2.7%	74.2	73.7	0.6%	66.4	64.8	2.4%
Below the engine (°C)	71.5	74	3.4%	73.2	72.3	1.2%	63.8	62	2.9%

© Fuzhou Polytechnic

exchanger porous medium model and the fan multi-reference frame model on the simulation results. Therefore, it is recommended to check the modeling details of key components in heat dissipation simulation, such as turbocharger, exhaust manifold, and cooling fan, to ensure that the settings of convective heat transfer coefficient and radiative heat transfer parameters are consistent with the actual situation [11]; at the same time, high-precision test methods such as Particle Image Velocimetry (PIV) flow field measurement and infrared temperature measurement should be used to verify the model's ability to capture local backflow at the rear of the radiator and around the engine, instead of only evaluating based on the overall temperature [27].

2.5.2. Targeted Analysis Considering SUV Structural Characteristics Compared with conventional passenger cars, SUVs feature a more compact layout of the engine compartment, and often operate under severe conditions such as trailer towing and off-road driving. These characteristics lead to more complex heat dissipation mechanisms. It is suggested to focus on the unique influencing factors of SUVs, such as the effects of larger grille openings on airflow distribution and flow blockage, and then propose targeted optimization strategies.

2.5.3. Carry Out Multi-Objective Collaborative Design of Heat Dissipation Schemes Multi-objective optimization has exhibited promising performance in relevant studies [5]. In future research, in addition to focusing on the mitigation of heat accumulation, the thermal efficiency of heat dissipation components in the engine compartment, such as radiators and intercoolers, can be further evaluated. By performing multi-parameter optimization—including adjusting the deflector angle and grille opening ratio—collaborative improvements in both thermal management and energy efficiency can be realized [29].

These three research directions complement each other and compensate for the deficiencies of existing studies from three perspectives: model accuracy, application relevance, and integrated optimization. In-depth investigations in these fields can enhance the comprehensiveness of heat dissipation mechanism analysis, the

practicability of improvement strategies, and the adaptability of numerical models, thereby providing more reliable theoretical support for the thermal management design of SUV engine compartments.

3. Conclusion

In this article, STAR-CCM+ software was used to conduct 3D CFD simulations of the engine compartment for a newly developed SUV model. The HAR areas inside the compartment and the causes of poor airflow were analyzed. Based on the compartment spatial layout, four schemes were proposed to improve its airflow. The optimal improvement scheme was then selected for experimental verification. The results showed that after optimizing the compartment flow field, the air could flow smoothly to the rear and carry away heat, ensuring the compartment air temperature was maintained within the permissible range.

The following conclusions were drawn from this study:

1. Under idle conditions, the airflow velocity inside the engine compartment was insufficient, resulting in a high overall compartment temperature and degraded cooling performance of the air-conditioning system.
2. In the original scheme, intake air entered the engine compartment through the front cooling module, but the airflow above the engine stagnated and induced HAR due to the lack of flow guidance.
3. Following the principle of minimal modification, a flow baffle was installed above the engine. This structure prevented hot air accumulation in the compartment, increased the intake of cool air, and effectively eliminated HAR.
4. After validating the effectiveness of the scheme under idle conditions, simulations and real-vehicle tests were further performed under severe driving conditions. The results confirmed that the optimized scheme maintained stable thermal balance performance.

5. This study provided a feasible technical solution for the thermal management of new SUVs. The results verified the effectiveness of the baffle in optimizing the flow field and provided references for improving the computational efficiency of automotive CFD simulations and shortening the vehicle development cycle.

Declaration of AI-Assisted Technologies

The authors declare that no generative AI technologies were used in the preparation and writing of this article. In doing so, the authors take full responsibility for the authenticity and scientific integrity of this article.

Contact Information

Huojie Shi, corresponding author
College of Mechatronics Engineering, Fuzhou Polytechnic,
Fuzhou, 350108, China
shj@fvti.edu.cn

References

1. Corrigan, D., Breda, S., Fontanesi, S., Cavina, N. et al., "Knocking Cylinder Pressure Data Interpretation for Modern High-Performance Engines—A Computational Fluid Dynamics Informed Approach," *SAE Int. J. Engines* 16, no. 2 (2023): 229-247, doi:<https://doi.org/10.4271/03-16-02-0014>.
2. Qiu, S., Xu, Z., and Huang, H., "Multi-Objective Optimization Study on the Power Cooling Performance and the Cooling Drag of a Full-Scale Vehicle," *Struct. Multidisc. Optim.* 64 (2021): 4129-4145, doi:[10.1007/s00158-021-03035-6](https://doi.org/10.1007/s00158-021-03035-6).
3. Lam, S.Y., Shuaib, N.H., and Hasini, H., "Computational Fluid Dynamics Investigation on the use of Heat Shields for Thermal Management in a Car Underhood," *Int. J. Automot. Mech. Eng.* 6 (2012): 785-796, doi:[10.15282/ijame.6.2012.10.0064](https://doi.org/10.15282/ijame.6.2012.10.0064).
4. Wang, Y., Gao, Q., Zhang, T. et al., "Advances in Integrated Vehicle Thermal Management and Numerical Simulation," *Energies* 10, no. 10 (2017): 1636, doi:[10.3390/en10101636](https://doi.org/10.3390/en10101636).
5. Saab, S., Hetet, J., Maiboom, A., and Charbonnelle, F., "Impact of the Underhood Opening Area on the Drag Coefficient and the Thermal Performance of a Vehicle," SAE Technical Paper [2013-01-0869](https://doi.org/10.4271/2013-01-0869) (2013), doi:<https://doi.org/10.4271/2013-01-0869>.
6. Vasudevan, S. and Bovo, M., "Subcooled Flow Boiling in High Power Density Internal Combustion Engines II: Numerical Modeling," *SAE Int. J. Engines* 16, no. 1 (2023): 97-120, doi:<https://doi.org/10.4271/03-16-01-0003>.
7. Nain, A., "Optimizing Cooling Fan Power Consumption for Improving Diesel Engine Fuel Efficiency Using CFD Technique," *SAE Int. J. Engines* 12, no. 4 (2019): 385-395, doi:<https://doi.org/10.4271/03-12-04-0024>.
8. Patidar, A., Gupta, U., and Marathe, N., "Optimization of Front-End Cooling Module for a Commercial Vehicle Using a CFD Approach," SAE Technical Paper [2013-26-0044](https://doi.org/10.4271/2013-26-0044) (2013), doi:<https://doi.org/10.4271/2013-26-0044>.
9. Xu, Q., Song, M., Zhang, J., Chen, J. et al., "Thermal Management Analysis of Engine Compartment Based on 1D and 3D Coupling Simulation," SAE Technical Paper [2019-01-0896](https://doi.org/10.4271/2019-01-0896) (2019), doi:<https://doi.org/10.4271/2019-01-0896>.
10. Lu, P., Gao, Q., Lv, L. et al., "Numerical Calculation Method of Model Predictive Control for Integrated Vehicle Thermal Management Based on Underhood Coupling Thermal Transmission," *Energies* 12, no. 2 (2019): 259, doi:[10.3390/en12020259](https://doi.org/10.3390/en12020259).
11. Kumar, V., Kapoor, S., Arora, G., Saha, S. et al., "A Combined CFD and Flow Network Modeling Approach for Vehicle Underhood Air Flow and Thermal Analysis," SAE Technical Paper [2009-01-1150](https://doi.org/10.4271/2009-01-1150) (2009), doi:<https://doi.org/10.4271/2009-01-1150>.
12. Kumar, V., Shendge, S., and Baskar, S., "Underhood Thermal Simulation of a Small Passenger Vehicle with Rear Engine Compartment to Evaluate and Enhance Radiator Performance," SAE Technical Paper [2010-01-0801](https://doi.org/10.4271/2010-01-0801) (2010), doi:<https://doi.org/10.4271/2010-01-0801>.
13. Seider, G., "Flow and Thermal Performance Prediction for Automotive Accessory Units and Their Integration into Underhood CFD Flow Analysis with Multi Thermal Systems," in Seider, G. and Bet, F. (eds.), *Vehicle Thermal Management Systems Conference and Exhibition (VTMS10)* (Frankfurt, Germany: Woodhead Publishing, 2013), 209-218, doi:[10.1533/9780857094735.5.209](https://doi.org/10.1533/9780857094735.5.209).
14. Ferrari, C., Beccati, N., and Pedrielli, F., "CFD Methodology for an Underhood Analysis towards the Optimum Fan Position in a Compact Off-Road Machine," *Energies* 16, no. 11 (2023): 4369, doi:[10.3390/en16114369](https://doi.org/10.3390/en16114369).
15. Gao, P., Gao, Q., and Zhu, Y., "The Simulation Methods Based on 1D/3D Collaborative Computing for the Vehicle Integrated Thermal Management," *Appl. Therm. Eng.* 104 (2016): 42-53, doi:[10.1016/j.applthermaleng.2016.05.047](https://doi.org/10.1016/j.applthermaleng.2016.05.047).
16. Xu, G., Wang, H., and Chen, L., "Predicting Unsteady Heat Transfer Effect of Vehicle Thermal Management System Using Steady Velocity Equivalent Method," *Sci. Prog.* 104, no. 2 (2021): 1-21, doi:[10.1177/00368504211025933](https://doi.org/10.1177/00368504211025933).
17. Della Torre, A., Montenegro, G., Cerri, T., and Onorati, A., "A 1D/Quasi-3D Coupled Model for the Simulation of I.C. Engines: Development and Application of an Automatic Cell-Network Generator," *SAE Int. J. Engines* 10, no. 2 (2017): 471-482, doi:<https://doi.org/10.4271/2017-01-0514>.

18. Khaled, M., Harambat, F., and Peerhossaini, H., "Underhood Thermal Management: Temperature and Heat Flux Measurements and Physical Analysis," *Appl. Therm. Eng.* 30, no. 6-7 (2010): 590-598, doi:[10.1016/j.applthermaleng.2009.11.003](https://doi.org/10.1016/j.applthermaleng.2009.11.003).
19. Khaled, M., El Hage, R., Harambat, R., and Peerhossaini, R., "Temperature and Heat Flux Behavior of Complex Flows in Car Underhood Compartment," *Heat Transf. Eng.* 31, no. 13 (2010): 1057-1067, doi:[10.1080/01457631003640321](https://doi.org/10.1080/01457631003640321).
20. Karamanos, K., Afrasiabian, E., Kim, S.I. et al., "Multifidelity Simulation of Underhood Thermal System for a Bus Engine," *Appl. Therm. Eng.* 201 (2022): 117707, doi:[10.1016/j.applthermaleng.2021.117707](https://doi.org/10.1016/j.applthermaleng.2021.117707).
21. Wang, P., Karpouzas, N., and Fujii, Y., "Two-Phase MRF Model for Wet Clutch Drag Simulation," *SAE Int. J. Engines* 10, no. 3 (2017): 1039-1048, doi:<https://doi.org/10.4271/2017-01-1127>.
22. Saini, P., Defoe, J., and Farbar, E., "Suitability Assessment of an Uncalibrated Body Force Based Fan Modeling Approach to Predict Automotive Underhood Airflows," *SAE Int. J. Adv. & Curr. Prac. in Mobility* 4, no. 2 (2022): 576-587, doi:<https://doi.org/10.4271/2021-01-0820>.
23. Florea, R., Taraza, D., Henein, N., Bryzik, W. et al., "Transient Fluid Flow and Heat Transfer in the EGR Cooler," *SAE Int. J. Engines* 1, no. 1 (2009): 558-570, doi:<https://doi.org/10.4271/2008-01-0956>.
24. Khaled, M., Mangi, F., El Hage, H., Harambat, F. et al., "Fan Air Flow Analysis and Heat Transfer Enhancement of Vehicle Underhood Cooling System – Towards a New Control Approach for Fuel Consumption Reduction," *Appl. Energy* 91, no. 1 (2012): 439-450, doi:[10.1016/j.apenergy.2011.10.017](https://doi.org/10.1016/j.apenergy.2011.10.017).
25. Khaled, M., El Hage, H., Harambat, F., and Peerhossaini, H., "Energy Management in Car Underhood Compartment—Temperature and Heat Flux Analysis of Car Inclination Effects," *Heat Transf. Eng.* 36, no. 1 (2015): 68-80, doi:[10.1080/01457632.2014.906283](https://doi.org/10.1080/01457632.2014.906283).
26. Itoh, A., Matsumoto, M., Hamamoto, N. et al., "Study on Large-Scale Many-Objective Optimization Method for Aerodynamics and Underhood Cooling Airflow Using Detailed Vehicle Model," *Trans. Soc. Automot. Eng. Jpn.* 52, no. 6 (2021): 1247-1253, doi:[10.1135/jsaeronbun.52.1247](https://doi.org/10.1135/jsaeronbun.52.1247).
27. Vdovin, A., Sathiyar, T.K., Duwig, C. et al., "A Test Rig for the Validation of CFD Simulations of a Passenger Vehicle Under Hood Environment," *Case Stud. Therm. Eng.* 59 (2024): 104423, doi:[10.1016/j.csite.2024.104423](https://doi.org/10.1016/j.csite.2024.104423).
28. Mondal, K., Kethavath, N.N., and Ghaisas, N.S., "Large Eddy Simulation Study of Boundary-Layer Flow behind a Smooth-to-Rough Surface Roughness Transition: LES of Flow behind Smooth-to-Rough Transition," *Boundary-Layer Meteorol.* 191 (2025): 16, doi:[10.1007/s10546-025-00906-z](https://doi.org/10.1007/s10546-025-00906-z).
29. Zhang, J., Wang, Q., Shu, C.M. et al., "A Multi-Objective Optimization Method for Under-the-Hood Thermal Management of Vehicles," *Appl. Therm. Eng.* 192 (2021): 116818, doi:[10.1016/j.applthermaleng.2021.116818](https://doi.org/10.1016/j.applthermaleng.2021.116818).