Abstract: High temperature of the radiator group is harmful to the power system and hydraulic system. In order to improve the heat dissipation performance of the loader, the flow field characteristics of the cooling chamber are analyzed by simulation and heat balance test. Firstly, the mathematical model of heat flow is established. Secondly, the flow field in the cooling chamber under different speeds is simulated based on CFX. And then the influence of fan position and internal flow field distribution on radiator performance is studied. Through the simulation of four different distances, it is concluded that the optimal distance between cooling fan and radiator is 76mm. Finally, a testing system is built for the temperature acquisition of engine water radiator, torque converter oil radiator, hydraulic oil radiator and air-to-air cooler of the hood structure, and then the simulation and optimization results are verified.

Keywords: Loader-digger, CFX, Flow performance, Heat dissipation performance, Temperature testing system

1 Introduction

The loader-digger is a multi-functional construction machinery product with both loading and excavation functions. It is widely used in industrial and agricultural production and defense industries. The heat load of the loader-digger heat removal system refers to the heat of the engine system, transmission system, and hydraulic system radiator. Most of this heat is dissipated through the convection heat exchanger of the radiator. The heat of the engine, transmission and hydraulic system is respectively mainly distributed by the engine water radiator, air-air cooler, torque converter oil radiator and hydraulic oil radiator. Because the working environment of the loader-digger is complex and diverse, the heat balance performance of the whole machine is required to be high. The engine, loading system, excavation system, and traveling system all need to be below the rated temperature in order to carry out various operations safely and efficiently.

There are three methods for evaluating the heat dissipation performance of construction machinery: First, thermal balance vehicle test. This method is the most direct and simple, but it has long cycle, high cost and poor repeatability. Second, it is a bench simulation test. This method is more accurate, with a higher degree of quantification, but it also costs a lot. The test equipment is extremely demanding. The third is the simulation technology. Through the establishment of the target product theory and simulation model, the use of simulation software for calculation and analysis, combined with physical experiments for comprehensive evaluation, this method can save a lot of time and manpower and resources, has become the most effective means of evaluating the thermal performance of complex machinery products one. Wang Jing [1] studied the changes of the flow field and temperature field in the cabin, using a combination of numerical simulation and experimentation. Fon-Chieh Chang [2] used a heavy-duty diesel engine for loaders. The cooling
system performed one-dimensional simulation analysis. The error between the simulation results and the test data was less than 10%. Sort M [3] simulated the cooling module fan of an off-road vehicle and compared the results of two different turbulence CFD (Computational Fluid Dynamics) models. The difference, in the test duct simulation model, studied the cooling module mass flow, fan energy consumption and fan efficiency characteristics. Zhang Dejun [4] based on Hypermesh and Fluent software simulation of an MPV (Multi-Purpose Vehicle) engine compartment thermal management. The simulation model of the gas flow in the engine compartment analyzed the thermal balance of the engine compartment of the car and solved the problem that the engine water temperature is too high. Khaled M [5] combined the FNM (Flow Network Modeling) fluid grid model with the CFD computational fluid dynamics method. The relationship was studied between the position layout of the electric and mechanical components in the power cabin and the influence of the temperature field. The automatic gas and heat management evaluation theory were put forward. The flow field and the temperature field were analyzed. Mahmoud Khaled [6] optimized the hood cooling module and proposed a method to control the positioning of the cooling module using the engine's energy demand, which reduced the heat transferred from the engine to the pump and the compression chamber, achieving the goal of high efficiency and energy saving. Vijay Datnodaran [7] was based on the FLUENT software platform to study the internal flow field and temperature field of the automobile power cabin. The simulation results were verified by wind tunnel test. The feasibility of solving the heat problem of the power cabin through the CFD method was verified. Jahani Kambiz [8] analyzed the impact of the different shape of the inlet grid on the flow field in the power cabin. A special front grille was constructed with the guide plate and the baffle plate. The inlet flow rate of the power cabin was increased by 10%, and the heat dissipation efficiency of the radiator was improved. Saha Rohit [9] studied the heat dissipation system of heavy load truck. The heat dissipation process of its power cabin was analyzed. A comprehensive evaluation of the heat dissipation effect of the power cabin was established, which was helpful to the early design of the heat dissipation system of heavy load truck. Zhang Chunhui [10] designed the wind pipe between the radiator and the inlet grille, which was beneficial to the introduction of the air inlet to the radiator. The uniformity of the gas flow was improved through the radiator. The grille was closed on both sides of the power cabin, reducing the resistance in the course of the vehicle and improving the economy of the vehicle's fuel. Jiajie Ou [11] studied the heat dissipation system of LPG (Liquefied Petroleum Gas) vehicle through field coordination theory. The air flow field inside the power cabin was analyzed. It was found that the air flow velocity and the thermal fluid range had a certain influence on the heat dissipation effect of the heat dissipation system. The angle of the velocity field and the temperature field were also obviously related. Liu Jiaxin et al. [12] studied the relationship between the cooling fan speed and the heat dissipation performance of the radiator in order to find out the performance law of the engine cooling system of the dual-fan engineering vehicle. Dong Xin et al. [13] proposed a method of heat flux coupling topology optimization based on density, which transformed the optimization design problem into a multi-objective optimization problem including heat resistance, energy dissipation and pressure drop. Xu Wang et al. [14] combined a full automatic multi-objective optimization with coupled heat transfer calculation to obtain the optimal cooling arrangement of transonic high-pressure guide vanes.

In order to solve the problem of overheating and residual heat accumulation in the loader-digger, the CFX is used to simulate and analyze the air flow of the cooling system under
the running condition. The distribution of the internal flow field under different speeds conditions is analyzed. The influence of the relative position of the fan and the radiator on the internal flow field is studied. And, the hood structure is optimized. Then, the cooling efficiency of the system under the running condition is improved.

The remainder of this paper is organized as follows. In Section 2, we establish the mathematical model of turbulent fields. Section 3 describes the numerical method. In Section 4, we analyze the simulation of the heat dissipation system of the excavator loader. We test the thermal balance of the vehicle in Section 5 and the performance of the excavator loader system is evaluated. Finally Sect. 6 concludes this paper and points out future research directions.

2 Mathematical models

The flow field in the bilge chamber of the loader-digger contains both flow transfer problems. Due to the low gas flow velocity in the chiller cabin, it can be considered as a non-compressible fluid calculation. The continuous equations of incompressible fluids and the Reynolds Navier-Stokes equations can be expressed as follows:

\[
\frac{\partial \rho}{\partial t} + \frac{\partial \rho U_i}{\partial x_i} = 0
\]  

(1)

\[
\frac{\partial \rho U_i}{\partial t} + \frac{\partial \rho U_j U_i}{\partial x_j} = -\frac{\partial \tau_{ij}}{\partial x_i} + \left(\mu + \mu_t\right)\frac{\partial U_i}{\partial x_j} + \frac{\partial}{\partial x_j}\left(\frac{\partial U_i}{\partial x_j}\right)
\]  

(2)

where \( U_i \) represents the \( i \) component of velocity, \( t \) represents time, \( x_i \) represents cartesian coordinate, \( \rho \) represents fluid density, \( p \) represents dynamic pressure, \( \mu \) represents viscosity, \( \mu_t \) represents eddy viscosity.

In fluid mechanics, the k-\( \varepsilon \) double equation model is often used to describe the turbulent flow field. The commonly used k-\( \varepsilon \) double equation models include Standard k-\( \varepsilon \) turbulence model, Realizable k-\( \varepsilon \) turbulence model and RNG k-\( \varepsilon \) turbulence model. Due to the complexity of the flow field in the bilge chamber of the loader-digger, the Realizable k-\( \varepsilon \) turbulence model adds a new equation to calculate the dissipation rate, which is even more significant for swirling flow, boundary laminar flow, flow separation and complex two-phase flow. Accurate, so the Realizable k-\( \varepsilon \) turbulence model is used to describe the turbulent flow field in the cooling chamber. The turbulence model is as follows:

\[
\frac{\partial (\rho \mu_k)}{\partial t} + \frac{\partial (\rho \mu_k u_i)}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\nu_{ei} \frac{\partial \mu_k}{\partial x_i}\right] + G_k + G_b - \rho \varepsilon - Y_{\mu} + S_{\mu}
\]  

(3)

\[
\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho \varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\nu_{ei} \frac{\partial \varepsilon}{\partial x_i}\right] + Y_{\mu} - \rho C_1 Y_{\mu} - \rho Y_{\mu} - C_3 \frac{\varepsilon^2}{k} + C_4 \frac{\varepsilon}{k} G_k G_b + S_{\varepsilon}
\]  

(4)

where \( \rho \) represents fluid density, \( x_i \), \( x_j \) represents each coordinate component, \( \mu \) represents molecular viscosity coefficient, \( \mu_t \) represents turbulent viscosity coefficient, \( \sigma_{\mu} \), \( \sigma_{\varepsilon} \) represent turbulent Prandtl number for turbulent kinetic \( k \) energy and dissipation rate \( \varepsilon \), \( G_k \) represents turbulent kinetic energy produced by mean velocity gradient, \( G_b \) represents the turbulent kinetic energy produced by the impact of buoyancy, \( Y_{\mu} \) represents the influence of the dynamic expansion of the compressible iliac artery on the total dissipation rate, \( S_{\mu} \), \( S_{\varepsilon} \) represent the user-defined source terms for the model.

3 Description of the numerical method

3.1 Mesh generation
In order to fully analyze the cooling capacity of the loader-digger, the physical model of the loader-digger is properly simplified, retaining the main heat source (engine, turbocharger and muffler) and components which can significantly affect the direction of air flow (air cleaner, air hood, intake and exhaust pipe, frame, hood, counterweight and cab). The simulation condition is the running condition of the sports car, and the loading end is at the lowest position. At this time, the loading end has little influence on the flow field in the heat-dissipating cabin, so the interference of the loading end is not considered. The radiators are all air-cooled, including transmission oil radiators, hydraulic oil radiators, engine cooling radiators, and air-air coolers. Their structure is a plate-fin structure. The dimensions and temperature requirements of each radiator are shown in Table 1. As shown. The fan is an eight-blade suction type with a maximum blade speed of 2200 rad/s.

| Name              | Length * Width * Height (mm) | Temperature requirements |
|------------------|-------------------------------|--------------------------|
| Transmission oil radiator | 635*70*225                  | TOC≤100°C                |
| Hydraulic oil radiator    | 635*70*490                   | HOC≤100°C                |
| Cooling radiator         | 635*110*650                  | RAD≤105°C                |
| Air-air cooler          | 710*110*130                  | CAC≤ring temperature +30°C |

According to the relevant technical data, the simulation model of the radiating cabin of a certain type of loader-digger is shown in Fig.1. Main components include radiators, fans, engines, turbochargers, mufflers, air cleaners, wind deflectors, intake and exhaust pipes, racks, hoods, counterweights, and cabs.

![Diagram of radiating cabin simplified model](image)

1. Rack 2. Turbo charger 3. Engine 4. Counter weight 5. Radiator Set 6. Air Hood 7. Air Filter 8. Muffler 9. Hood 10. Cab

Fig.1 Radiating cabin simplified model

In order to analyze the internal characteristics of the radiating cabin in the working conditions of the excavating loader, a virtual wind tunnel simulation model is established. Since the front air intake of the vehicle is affected by the external flow field, a calculation domain suitable for the external flow field is adopted. The size of the external flow field is 10 times the length of the vehicle, 5 times the width of the vehicle and 3 times the height of the vehicle. The size of the calculation domain is: the distance between the entrance and the front of the loader-digger is 10 meters. The distance between the exit and the back of the loader-digger is 20 meters. The distance between the upper wall and the top of the cab is 10 meters. And, the distance between the two side walls and the sides of the excavator loader is 5 meters.

Due to the complex structure of the heat-dissipating cabin, a tetrahedral grid with good
adaptability is selected for the heat-dissipation cabin and the periphery of the car body. The structure of the radiator group is regular and a hexahedral grid is adopted. The flow field on the surface of the heat-dissipating cabin changes drastically. And, the surface of the heat-dissipating cabin is refined. The results of meshing are shown in Fig.2. The total number of meshes in all computational domains is approximately 7.05 million.

![Fig.2 Compute domain meshing](image)

### 3.2 Boundary conditions

For the change of the flow field and temperature field inside the radiating cabin, the following simplifications and assumptions are made in the numerical simulation. A three-dimensional steady-state flow field is adopted, and the inlet wind speed is uniform. The airflow rate in the heat-dissipating cabin is low and can be considered as incompressible Newtonian fluid, turbulence model using Realizable k-Epsilon model. The effects of thermal radiation and gravity field are ignored. The convection with the outside air is ignored. The loader-digger is full load, and the environment is the worst case.

The boundary conditions are set as follows: The entrance is set to speed entrance, the speeds are respectively 0, 6.8km/h, 11.1km/h, 23.8km/h, 42.5km/h. The exit is set to free flow exit, the pressure is 0Pa. The fan speed is 2200rpm. The ambient temperature is 45°C. The radiator is set to a porous medium and the material is aluminum.

### 4 Simulation of heat dissipation system based on ANSYS CFX

#### 4.1 Prediction of flow performance at different speeds

Under the five operating conditions of 0, 6.8, 11.1, 23.8, and 42.5 km/h respectively, the airflow in the radiating cabin and the pressure cloud diagram are compared and analyzed in the same vehicle environment.

Fig.3(a) is a vertical cross-sectional pressure cloud diagram at the center of the bilge chamber when the loader-digger is idling. Due to the suction effect of the fan, a large low-pressure zone is generated near the radiator group at the front of the fan. A high-pressure zone is formed near the engine at the rear of the fan, and at the entrance of the radiator compartment and the air intake at the bottom of the engine. The pressure is also higher, so the difference in pressure causes the outside air to flow into the radiating cabin. Fig.3(b) shows the speed vector diagram. The air entering from the air inlet is directly sucked by the fan, flows through the radiator group, passes through the air hood and the fan, and then from the engine, the air intake system and the muffler. The surface of the components and parts flows through them, and finally they are discharged from the bottom of the heat-dissipating chamber and the air outlets on both sides of the hood to the heat-dissipating chamber. There is no serious backflow near the engine, and the overall flow of the cooling chamber in the cross section is smooth.
According to Fig.3 and Fig.4, when the vehicle speed is low, the fan pressure rise and the air hood influence the intake air volume very much. As the vehicle speed increases, the intake air volume increases, the flow into the radiator group increases, and the fan pressure rises. The influence of the air hood on the cooling air volume is gradually reduced. Due to the compact structure of the heat-dissipating cabin and numerous components, gas flows through the heat-dissipating cabin toward the rear and the underside of the vehicle body, and is hindered by a plurality of components, resulting in separation and generation of complicated eddy currents, which also leads to an increase in internal flow resistance. In high-speed operating conditions, the vehicle speed is higher, the air velocity entering the air inlet is higher, the overall flow rate in the heat-dissipating cabin is higher, and the flow condition is better. It is more important to note that the fan pressure rise and air deflector are at idle speed and low speed in the loader-digger. Therefore, in order to increase the air intake flow rate under idle conditions and improve the fan efficiency, the influence of the position of the fan of the loader-digger on the flow performance in the radiating cabin is studied.

4.2 Prediction of flow performance on the relative position of the fan and the radiator

When the vehicle speed is low, the air flow in the cooling chamber is mainly generated by the suction action of the fan. Therefore, the relative position of the fan and the radiator set at idle is studied. Select the fan and radiator group relative positions are respectively 56, 76, 96, 116 mm to establish four kinds of programs, and radiator group and engine relative distance, inlet speed is 0, the fan speed is 2200rpm.
When the fan speed is 2200rpm, the speed vector diagram of the four programs under idle conditions is shown in Fig.5. It can be seen from the figure that the flow rate behind the engine of option 4 is low, and it is easy to cause waste heat to accumulate. The flow field in option 2 and option 3 for the radiating cabin is denser, which is conducive to the heat discharge from the radiating cabin.

![Fig.5 x=0 cross-sectional speed vector at idle](image)

The horizontal cross-section velocity vector at the center of the fan is shown in Fig.6. It can be seen from the figure that the air flow path becomes smaller due to the smaller gap between the engine and the cab, thus making the airflow in the cab. There is certain stagnation and backflow between the engine and the engine, and the flow rate behind the engine is small, which is unfavorable to the heat dissipation performance of the engine and the entire radiating cabin. The vortex phenomenon occurs behind the mufflers of option 3 and option 4 and the vortex phenomenon behind the mufflers of option 1 and option 2 are weak.

![Fig.6 z=1.5 section speed vector at idle](image)

### 4.3 Optimization of the hood

When the vehicle speed is high, the air flow in the radiator compartment is mainly driven by
the air entering the air inlet. At this time, the overall flow rate in the overall radiator compartment is high. However, due to the compact structure of the cooling chamber and numerous components, the flow velocity above the engine is relatively low. Therefore, the hood is optimized and improved, and the exhaust port shown in Fig.7 is added. The entrance speed is 42.5km/h, and the fan speed is 2200rpm.

![Fig.7 Hood to increase the exhaust port](image)

Fig.7 Hood to increase the exhaust port

Fig.8 shows the vertical section speed vector at the center of the chiller cabin when the speed of the loader-digger is 42.5 km/h. Comparing the calculation results, it shows that after the optimization, the speed tends to increase significantly above the engine, and the flow velocity distribution behind the engine is more uniform.

![Fig.8 x=0 cross-section speed vector before and after optimization](image)

(a) After optimization (b) Before optimization

Fig.8 x=0 cross-section speed vector before and after optimization

Fig.9 shows the horizontal cross-section velocity vector at the center of the fan and at the center of the muffler. From the figure, it can be compared and found that the optimized flow field on the right side and rear side of the engine is more uniform than before the optimization. The phenomenon of eddy currents and stagnation becomes weaker. After optimization, the flow velocity above and behind the engine becomes faster, and the eddy current behind the muffler weakens, but a vortex is generated in front of the silencer due to air collision.
5 Heat balance test

In order to understand the working performance of the excavator loader, check whether its heat balance performance meets the requirements under various load conditions, and analyze the thermal balance state of the whole machine. The vehicle heat balance of the 766A excavator loader under different working conditions was tested to evaluate the performance of the cooling system.

5.1 Heat balance test environment and machine configuration

The test environment is fog free, rain free and snow free. The ambient temperature is more than 24°C, the air pressure is 95~102kPa, and the wind speed is no more than 5km/h. The test site shall be the site closest to the actual working condition (can be on the actual site or special test site).

The test prototype is the 766A excavating loader produced by a certain manufacturer. The main parameters are shown in table 2.

| parameter                      | value          | parameter                      | value          |
|--------------------------------|----------------|--------------------------------|----------------|
| Operating hydraulic system pressure | 22MPa          | Full load lift time on the loading side | 4.0s          |
| Engine power                   | 75kW           | Load down time at full load     | 2.6s          |
| Maximum no-load engine speed   | 2307r/min      | Load loading load unloading time | 1.6s          |
| No load machine quality        | 890kg          | Action time three items and     | 8.2s          |
| Full load machine quality      | 10570kg        | I gear speed                    | 9.24km/h      |
| Bucket rated capacity          | 1m³            | II gear speed                   | 16.21km/h     |
| The bucket dug hard            | 53.54kN        | III gear speed                  | 27.4km/h      |
| The moving arm lifts the force | 41.00kN        | IV gear speed                   | 39.18km/h     |
5.2 Test instrument and data acquisition point arrangement

In order to analyze the heat balance of the excavator loader under various load conditions, the heat balance of the whole vehicle is tested, and the performance of the excavator loader system is evaluated. The test equipment are arranged as follow: temperature measuring device (thermocouple wire) accuracy is 1 °C, timer accuracy is 0.1s, data acquisition recorder, sampling frequency is 1Hz, ncode instrument and tce software. The test points are set to: engine cooling water inlet and outlet temperatures, hydraulic cooling oil inlet and outlet temperatures, converter cooling oil inlet and outlet temperatures, and air-to-air cooler inlet and outlet air temperatures, shown in Fig.10.

![Test data collection points](image)

5.3 Analysis of test results

The test condition is a sports car condition, that is, the highest-grade full-speed running condition. The test results are shown in Fig.11.  
![Radiator group temperature curve at full speed driving conditions](image)
As can be seen from the figure, the maximum temperature of the cooling system of the loader-digger is shown in Table 3. Since the loader-digger is often working under harsh conditions, the heat dissipation system temperature must be converted to the harshest environment, i.e., the ambient temperature is 45°C. Referring to Table 1, it can be seen that the temperature of each radiator of the optimization scheme is lower than the temperature limit. The heat dissipation performance of the entire machine after optimization is up to standard. The optimized scheme can make the loader-digger operate in an efficient and energy-saving manner.

From the comparison between the actual measured temperature before and after optimization in Table 3, it can be seen that the overall temperature of the improved radiator group slightly decreases. Among them, the cooling effect of the air-air cooler and the torque converter oil radiator in the upper half of the radiator assembly is more significant than that of the engine radiator and the hydraulic oil radiator in the lower half, and the torque is converted after the temperature is converted. The oil cooler temperature is reduced by 6.3%.

Table 3 Test results of full-speed driving conditions

| Measured point                          | Measured temperature (°C) | Converted temperature (°C) | Measured temperature before improvement (°C) |
|----------------------------------------|---------------------------|-----------------------------|---------------------------------------------|
| Ambient temperature                    | 32                        | 45                          | 32                                          |
| Engine cooling water inlet temperature | 77                        | 90                          | 80                                          |
| Engine cooling water outlet temperature| 73                        | 86                          | 75                                          |
| Torque converter cooling oil inlet temperature | 76                    | 89                          | 82                                          |
| Torque converter cooling oil outlet temperature | 72                    | 85                          | 78                                          |
| Hydraulic cooling oil inlet temperature | 62                        | 75                          | 63                                          |
| Hydraulic cooling oil outlet temperature | 35                       | 48                          | 35                                          |
| Air-air cooler inlet air temperature    | 35                        | /                           | 36                                          |
| Air-air cooler outlet air temperature   | 153                       | /                           | 160                                         |

6 Conclusions and outlooks

According to this study, the air intake of the radiator group is greatly affected by the fan pressure rise and the air guide hood when the speed of the excavator loader is low. With the increase of vehicle speed, the intake air of the front end increases, and the flow into the radiator group increases. The fan pressure rise and the air hood will gradually reduce the influence on the air intake of the radiator group.

For the fourth part of the studies, when the loader-digger is idling, the air intake of the radiator group increases first and then decreases as the distance between the radiator group and the fan increases. When the distance between the two is 76mm, the air intake of the radiator group is the largest.
The last set of the studies is the improvement which is the addition of exhaust ports on the hood. Under the running conditions of the sports car, the overall flow rate inside the cooling chamber has been improved, the flow field velocity distribution has become more uniform, the vortex phenomenon at the rear of the engine has been eased, and the backflow phenomenon at the top of the engine has weakened, which is conducive to an increase in the air intake of the radiator group. With the heat dissipation performance of the radiating cabin, the temperature of the torque converter oil cooler is reduced by 6.3%. The actual vehicle test verified the feasibility of the optimized solution.

Due to the limitation of some conditions, the research work in this paper is still limited. The contents to be improved and the further work to be carried out in this paper are as follows:

1. due to the complicated working conditions of stone shoveling and transporting by the excavator loader, it is necessary to analyze each movement in detail.

2. as the cooling system is a dynamic cooling process, it needs to be combined with electromechanical fluid integration and dynamic simulation.

Abbreviations

| Abbreviation | Description |
|--------------|-------------|
| CFX          | a practical fluid engineering analysis tool |
| CFD          | Computational Fluid Dynamics |
| MPV          | Multi-Purpose Vehicle |
| FNM          | Flow Network Modeling |
| LPG          | Liquefied Petroleum Gas |

Acknowledgements

The authors acknowledged the anonymous reviewers and editors for their efforts in valuable comments and suggestions.

Funding

NO

Availability of data and materials

Data sharing is not applicable to this article as no datasets were generated or analyzed during the current study.

Authors’ contributions

LI LEI proposes the innovation ideas and theoretical analysis. CHEN XI conceived of the study, and participated in its design and coordination and helped to draft the manuscript. All authors read and approved the final manuscript.

Competing interests

The authors declare that they have no competing interests.

Publisher’s Note

Springer Nature remains neutral with regard to jurisdictional claims in published maps and institutional affiliations.

Author details

1Mechanical Engineering College, Zhenjiang, Jiangsu University of Science and Technology, 212003, China
2Jiangsu Liu-gong Machinery Co., Ltd., Zhenjiang, 212000, China
References

1. Wang Jing, Zhang Chengchun, Zhang Chunyan, et al., Influence of Opening of Passenger Car Rear Door on Heat Dissipation of Engine Compartment. Journal of Agricultural Machinery. 43(9), 31-36 (2012)

2. Chang F C, Malipeddi S, Uppuluri S, et al., Underhood Thermal Management of Off-Highway Machines Using 1D-Network Simulations. International Truck & Bus Meeting & Exhibition (2003)

3. Sortor M, On-System Engine Fan Measurement as a Tool for Optimizing Cooling System Airflow Performance and Noise. Sae International Journal of Materials & Manufacturing. 4(1):1221-1230 (2011)

4. Zhang Dejun, Wang Wenyong, Application of CFD Technology for Thermal Management of Vehicle Engine Room. Development and Innovation of Mechanical and Electrical Products. 25(6), 113-115 (2012)

5. Khaled M, Ramadan M, El-Hage H, et al., Review of underhood aerothermal management: Towards vehicle simplified models. Applied Thermal Engineering. 73(1), 842-858 (2014)

6. Mahmoud Khaled, Fan air flow analysis and heat transfer enhancement of vehicle underhood cooling system–Towards a new control approach for fuel consumption reduction. Applied Energy. 91(1), 439-450 (2012)

7. Damodaran V, Rahman M, Front-end cooling airflow performance prediction using vehicle system resistance. SAE International-Thermal Management SP1751, SAE Paper (2003)

8. Jahani K, Beigmoradi S, Under-hood air flow evaluation of pedestrian-friendly front-end style using CFD simulation. SAE International Journal of Passenger Cars-Mechanical Systems. 7(2), 787-792 (2014)

9. Saha A K, Acharya S, Parametric study of unsteady flow and heat transfer in a pin-fin heat exchanger. International journal of heat and mass transfer. 46(20), 3815-3830 (2003)

10. Zhang C, Uddin M, Song X, et al., Simultaneous improvement of vehicle under-hood airflow and cooling drag using 3D CFD simulation. SAE Technical Paper (2016)

11. Ou J J, Li L F, Cui T, et al., Application of field synergy principle to analysis of flow field underhood of LPG bus. Computers & Fluids. 103, 186-192 (2014)

12. Liu Jiaxin, Wang Baozhong, Qin Sicheng et al., Numerical simulation of heat dissipation performance of cooling module for construction vehicles with dual fan. Journal of Huazhong University of Science and Technology. 46(4), 127-132 (2018)

13. Dong Xin, Liu Xiaomin, Multi-objective optimal design of microchannel cooling heat sink using topology optimization method. Numerical Heat Transfer; Part A: Applications, 77(1) 90-104 (2020)

14. Xu Wang, Huazhao Xu, Jianhua Wang et al., Multi-objective optimization of discrete film hole arrangement on a high pressure turbine end-wall with conjugate heat transfer simulations. International Journal of Heat and Fluid Flow. 78(2019)
Figures

1. Rack 2. Turbo charger 3. Engine 4. Counter weight 5. Radiator Set 6. Air Hood 7. Air Filter 8. Muffler 9. Hood 10. Cab

Fig. 1 Radiating cabin simplified model

Fig. 2 Compute domain meshing

(a) Section pressure map  (b) Speed vector

Fig.3 x=0 section pressure and speed diagram at idle
(a) 11.1km/h                             (b) 42.5km/h
Fig. 4 x=0 cross-section speed vector at different speeds

(a) Option 1                                     (b) Option 2

(c) Option 3                               (d) Option 4
Fig. 5 x=0 cross-sectional speed vector at idle

(a) Option 1                                     (b) Option 2

(c) Option 3                               (d) Option 4
Fig. 6 z=1.5 section speed vector at idle
Fig. 7 Flood to increase the exhaust port

Fig. 8 $x=0$ cross-section speed vector before and after optimization

Fig. 9 $z=1.7$ cross-section speed vector before and after optimization
Test data collection points

Torque converter oil radiator
Hydraulic oil radiator
Air-air cooler
Engine water radiator

Torque Converter Oil Inlet
Hydraulic Oil Outlet
Cold Inlet
Water Inlet

Torque Converter Oil Outlet
Hydraulic Oil Inlet
Cold Export
Water Outlet

Fig. 10 Distribution of test data collection points

(a)
Water Inlet Water Outlet Torque Converter Oil Inlet Torque Converter Oil Outlet

(b)
Hydraulic Oil Inlet Hydraulic Oil Outlet Cold Inlet Cold Export

Fig.11 Radiator group temperature curve at full speed driving conditions