CHEMICAL ENGINEERING TRANSACTIONS

VOL. 66, 2018

A publication of AIDIC
The Italian Association of Chemical Engineering
Online at www.aidic.it/cet

Guest Editors: Songying Zhao, Yougang Sun, Ye Zhou

ISBN 978-88-95608-63-1; **ISSN** 2283-9216

DOI: 10.3303/CET1866066

Research on Features of Heat Spatial Distribution for Heat Exchanger of Automobile Engine Bay and Improvement Model

Maosheng Tian[a, b]

[a]School of Automobile, Chang'an University, Xi'an 710064, China;
[b]School of Traffic and Transportation, Chongqing Jiaotong University, Chongqing 400074, China
maoshengtian5439@126.com

This paper sets to analyse the features of heat spatial distribution and improvement model for the heat exchanger used in the automobile engine bay. We adopted the method focused on research subjects of bus and truck and based the research on the three-dimensional model and numerical simulation of the heat exchanger. The results showed that the external cool air can enter directly into the cooling module of the automobiles and be emitted back to the outside after the cooling refrigerant, thus forming an airflow loop. To calculate the pressure loss, it should be counted as system resistance. It helps to satisfy the heat radiation needs of automobiles by improving the model of heat spatial distribution of heat exchanger in the automobile engine bay. In conclusion, we used the DUAL heat dissipation module in the calculation of heat spatial distribution of automobile engine heat exchanger by focusing on analysing the features of spatial distribution with the help of the three-dimensional model and relevant calculations, enabling the reflection of the process where cool airflow is heated and then cooled internally by the cooling refrigerant.

1. Introduction

Engine is a key component of automobiles. The internal heat flowing and transferring varies among different types of engine bays, therefore the controls from the engine system have an effect on the surrounding environment and utilization of energy. Faced with tightened resource and pressing environmental problems, it is vital that we understand the heat spatial distribution features of the engine heat exchanger and improve the exiting models.

This paper conducted research on subjects including bus and truck, and analyzed the models via one-dimensional numerical simulation, heat exchange process of the heat exchangers, and three-dimensional modelling. We used the DUAL heat dissipation module and three-dimensional model in the calculation of heat spatial distribution of automobile engine heat exchanger and discussed the approach to have precise calculation of heat distribution of automobile heat exchangers so as to provide support to the research on the heat dissipation needs of automobiles.

2. Literature review

Based on the calculation theory and demand, the model of engine cabin heat management is assumed and simplified in combination with the actual conditions of computing resources. In accordance with hypothesis and simplification, a three-dimensional numerical calculation model of bus and truck is set up, and a three-dimensional numerical model is calculated based on the standard turbulence model. According to the simulation results, the problems in vehicle design are improved, and one-dimensional numerical calculation is corrected. Combined with the 3D calculation model of the two vehicle types, Dominik and others simulated the standard model, realization model, shear pressure transmission model and other four turbulence models. In addition, they analyzed the effective viscosity correction and eddy viscosity correction two modified models and compared their performance and applicability in the calculation (Dominik et al., 2017). Frate and so on, through the comprehensive comparison, obtained the numerical calculation of the realization model which is more suitable for the engine cabin heat management, and the model needs to adopt its enhanced model to improve the simulation precision of the engine cabin simulation (Frate et al., 2017). Kakavand and Molana analyzed the characteristics of heat space distribution in heat exchanger. In view of the problems in the influence of the average volume heat source in the space position of heat exchanger to the spatial heat distribution, two improved methods are proposed: non-uniform heat source method and element model method (Kakavand and Molana, 2018). The calculation results show that the two methods can improve the spatial distribution of heat in the heat exchanger and provide a reliable improvement on studying the influence of the space position of the heat exchanger on the heat dissipation.

Auto heat management starts from the point of view of system and integration. It integrates the main system into a whole, and then considers the heat relation of the entire system as a whole. Combining the control theory and the method of system management, the heat transfer of automobile is controlled and optimized, so as to ensure the efficient operation of the car and reduce the size and power consumption of the system. At the same time, the vehicle economy is improved and the emissions are reduced. Lee and Kim studied the thermal management of vehicles, including the thermal characteristics of heat management objects, the integration of thermal management systems as well as the comprehensive utilization of thermal energy (Lee and Kim, 2017). In broad sense, it contains management and optimization of all vehicular heat sources. The heat balance test of automobile engine is mainly to test the energy balance of the engine. Heat balance experiments were carried out under the conditions of engine acceleration, maximum torque and calibration conditions. The experimental results show the proportion of the heat released by the combustion engine in each system of the engine. At the same time, the heat dissipation requirement of the engine can be obtained in various working conditions, which provides the basis for the matching and selection of the radiator and the medium cooler. The heat balance test can provide the boundary conditions for one-dimension calculation: the engine speed, the cooling capacity of the water tank, the cooling capacity of the oil cooler, the inlet temperature of the cooler, the flow of the cooler, the inlet pressure of the medium cooler and so on.

The technology of vehicle thermal management is put forward on the premise of energy crisis, environmental pollution and people's requirements for riding comfort. It is a comprehensive management of the relationship between the various systems and the whole vehicle from the point of view of the system and the whole. It adopts the methods of modularization of components, the integration of systems and accessories, and the methods of controlling and managing. Under the premise of ensuring efficient and reliable operation of components, energy-saving, emission reduction and ride comfort are achieved. Thermal management technology has become an important measure to improve fuel economy, ensure safety and reduce emissions. Righetti and others, in the traditional automobile design process, evaluated the heat dissipation of engine ship by experience or engineering scheme. After the sample is produced, experimental means are used for later verification. In this way, more accurate and visual data can be obtained, but the design cycle is long and the cost is high (Righetti et al., 2017). Shiga and Fujii stated that, simulation calculation becomes an important means of vehicle thermal management through the progress and development of computing technology. Compared with the experiment, the simulation calculation has some remarkable characteristics, such as pre-research, unconditional restriction, rich information, low cost, short cycle, and so on. The experimental results can be used to calibrate the numerical model, and numerical simulation can use similarity criterion to simulate the experiment (Shiga and Fujii, 2017).

The research of automobile thermal management is developing rapidly in China. The research of automobile energy management is mainly focused on two aspects: the study of the overall balance and the details of the key parts. In the study of key components, the researchers have done a lot of researches and obtained many reliable results, which almost cover all the components related to the heat source of the vehicles. Sukkar et al. applied genetic algorithm to the optimization and design of automobile radiator, proposed the optimization scheme of the structure parameters of the radiator core, and verified the scheme by experiment (Sukkar et al., 2017). Thaker and so on, through the numerical simulation analysis of the radiator core with blinds, obtained the influence characteristics of the main structural parameters of the blinds on the key factors related to the flow and heat transfer. In addition, the flow and heat transfer characteristics of the best geometric structure parameters of the blinds are obtained. A reliable analysis and evaluation method is formed in domestic, and many domestic radiator manufacturers cooperated with colleges and universities to carry out experiments and simulation calculations, making the performance of the domestic radiator greatly improved (Thaker et al., 2017).

To sum up, the above research work mainly analyzes the characteristics of the heat space distribution of the engine compartment heat exchanger and the improved model. From the point of view of the computer, the distribution space factors of the automobile heat management system are analyzed, and the design of the genetic algorithm is applied to the system analysis of the automobile heat dissipation device. Therefore, based on the above research status, the characteristics of the heat space distribution of the engine compartment heat exchanger and the related analysis of the improved model are focused on, and the data explanation and help are made from the experimental point of view and the simulation model.

3. Methods

3.1 One-dimensional Numerical Simulation of Automobile Engine Bay

The energy flow loop of automobile engine bay mainly consists of the following, including engine, intercooler, water box, oil cooler, thermostat, as well as pipeline connecting various parts, air grid and other envelop structures of the engine bay. To conduct the one-dimensional calculation, simulative calculation often serves two purposes: one to determine the size of the cooler and intercooler to ensure that the targeted heat dissipation demand can be met; the other is to double check that the prescribed cooler and intercooler can meet the demands of heat dissipation. In order to conduct the one-dimensional calculation, a number of boundary conditions were set. All of the parameters can be obtained following the approaches in the first three sections of the paper, but the restriction loss caused by the air grid and pipeline that needed to be provided via extra experiments or three-dimensional calculations. Software FLOWMASTER or KULI were used to conduct the calculations following the method of heat dissipation module.

3.2 DUAL Module Modeling

The DUAL module modeling can be used to simulate the process of heat exchange. It consists of two grid regions, representing certain work medium respectively that completely overlaps with each other spatially. In the calculation, the flow pressure loss can be calculated using the porous medium, but the inter-regional heat transfer was computed using the heat dissipation module method. In the DUAL model, the two flows are directed orthogonally. When conducting the modeling, it was better to make sure that not only the two regions overlapped spatially, the modules of the regions also overlapped. This was because when calculating the inter-region heat transfer, it was based on the heat transfer between the modules. In line with the requirement on treating the grid using DUAL module method, taking into considerations of the truck model and its boundary value conditions, we established calculation models for automobile engine bay heat management, and set to improve the heat distribution around the heat exchanger using three-dimensional calculation.

3.3 Three-dimensional Modeling for Bus and Truck

The bus and track have differences in terms of the body structure, dynamic system and cooling system. The differences results in subtle variances on the flow characteristics of the two types of automobiles. In spite of this, the key parts determining the heat balance are the same for bus and truck, such as fan, intercooler, and heat exchanger. As a result, the three-dimensional modeling for the two automobiles was basically the same. In this paper, we based our modeling on bus, and provided details of modeling for bus. The three-dimensional model for truck was developed based on the modeling process for bus. Following the modeling method of three-dimensional calculation for bus, the model for truck was established with an overall grid of over 14 million.

The fan area is an important component to the cool air distribution and flow in automobiles. Accurate simulation of fans plays a vital role in automobile heat management calculations. The fan parts must be prepared with great attention to the details. But if the fan is too detailed, it would pose much pressure on computing resources since the number of grid required would spike due to the details. The extent of dissipation that can serve the purpose of calculation should be discussed before overall calculation was made. This paper adopted the grid independence detection and three-dimensional simulative calculation, under the same boundary conditions, to analyze and compare the models for different blade mesh division schemes. Our analysis focused on the trend of flow variance along with the grid changes, and selected a proper division scheme. The calculation of fan was based on the MRF model with a speed of 1800 r/min. The pressure inlet and outlet were both set at the atmospheric pressure. The duct passageway was set as the boundary condition of the slip wall, and the duct resistance was realized through the evenly distributed holes. The calculation was to facilitate the selection of a proper dissipation plan between the fan and grid. In the process of modeling, only the grid scale for the fan area and its adjacent was adjusted. For the holes, since the mesh is high, it was kept unchanged during model adjustment. Table 1 shows the grid division scheme of fan grid independent monitoring.

Based on the analysis, with the increase of grid number population, the fan has higher traffic. But to a certain level, the traffic will not change even when the grid number population is further enhanced. By this point, we believe that the calculation result no longer varies along with the population of grid number. Figure 1 shows Air flow values under different grid values.

Table 1: The grid division scheme of fan grid independent monitoring

plan	Number of blade profile grid	Population grid number(m)
one	2	83
two	3	126
three	4	263

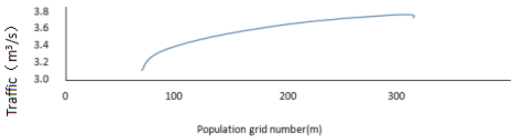

Figure 1 : Air flow values under different grid values

The water tank and intercooler region were simplified by the porous medium, while the porous medium interchanges with the external using the Interface method to realize transfer of substance and heat at the joints. In terms of heat exchanger calculation, we discarded the influence of gravity and radiation, and assumed that the property of air is only relevant to temperature. Integrated with Dacy's law of porous medium, the relationship between pressure and speed was established with the speed and pressure data. In this way, we determined the main parameters of porous medium fluidity: viscosity resistance coefficient and inertia resistance coefficient. Table 2 shows Flow performance data of bus tank.

Table 2: Flow performance data of bus tank

Cold side volume flow (m^3/s)	Wind resistance（kPa)
7.2	0.531
6.4	0.441
5.6	0.361
4.8	0.281

The shape of the engine exerts certain influence over the dynamics inside the engine bay of automobiles. Taking into consideration of existing boundary conditions and their imposing methods, we segmented the engine into various regions and left out the parts that have negligible influences over the heat flow and transfer overall. The boundary condition for the engine shell was set to be a temperature condition. And the segmentation was based on temperature. Hence, this paper divided the engine into the following parts: intake pipe, exhaust pipe, red cover area, trouble wheel shell, compressor shell, oil pan, block, pulley, oil filter, high-pressure oil pump and oil pipeline and others.

4. Results and Analysis

4.1 Improvements by Three-dimensional Calculation over One-dimensional Calculation

In one-dimensional calculation, the air resistance of the air duct was assumed, making the result of calculation a primary conclusion. However, based on the analysis of the three-dimensional calculations, we improved the boundaries of the one-dimensional calculation, and achieved accurate results. When the automobile is in motion, the speed of driving amounts to enhancing the inlet pressure before the intercooler, hence the increased overall traffic of cool air. As a result, the speed of automobile mainly influences the inlet pressure into the cooling module. The truck and bus has different layout of cooling modules. For the truck, the inlet grid is set vertical to the driving direction, while the inlet grid is set parallel to the driving direction for the bus. The cool air has a turn of direction before entering the air grid in the bus and has another turn of direction after entering the air grid and before the intercooler.

The cool air completes a loop of flow from the outside to the cooling module of the automobile, through the cooling refrigerant, and be remitted back to the outside. All the pressure loss but the pressure change over the exchanger module and fan area were counted as system resistance. The assumption of system resistance is a primary estimate, and after three-dimensional calculations, we arrived at precise calibrations of the system resistance.

Table 3: results of one-dimensional calculation of freight truck

Working condition	The outlet temperature of the cooler（°C）	Heat sink inlet temperature （°C）	The environment temperature（°C）	Wind resistance（Pa）
Calibration mode	45.1	88.1	38	125.8
Maximum torque	44.8	91.0	38	89.5

Table 4: calculation results of one-dimensional bus

Working condition	The outlet temperature of the cooler（°C）	Heat sink inlet temperature（°C）	The environment temperature（°C）	Wind resistance（Pa）
Calibration mode	57.3	93.9	42	420.2
Maximum torque	53.7	93.3	42	290.2

With the results of the three-dimensional calculations (As figure 3 and figure 4), the one-dimensional calculation was improved. We found out that the truck's heat exchanger met the design requirements, while the intercooler and water tank of the bus did not. Adjustments needed to be made to the sizes of the intercooler and water tank of bus. Combined with the results of three-dimensional calculations and given that the model and height of the bus intercooler and water tank stayed unchanged, the width should be increased to 680 mm and 790 mm in order to meet to requirements of heat dissipation. By that time, with the declared work conditions and maximum torque condition, the outlet temperature at the intercooler was witnessed at 54.1°C and 51.8°C respectively; while the inlet temperature at the water tank was 91.2°C and 89.1°C respectively.

4.2 One-dimensional Calculation Results

The outlet temperature at the intercooler was below room temperature by 13t: and the inlet temperature at the water radiator by 92°C. The initial calculations indicated that the truck's heat exchanger was up to the design requirements, while the water tank of the bus was not. Though the intercooler of the bus can met the design requirements, it required special attention since it mostly met the boundary requirements. Without coupling with the three-dimensional calculation results, the result of air duct resistance varied greatly from the actual conditions. This variance would result in deviation of the cool air traffic, hence affecting the temperature calculations of the intercooler and radiator, as table 5 shown.

Table 5: size structure parameters of truck and bus heat exchanger

Type car	Heat exchanger type	High(mm)	wide(mm)	thick(mm)
truck	The radiator	700	700	55
truck	Cold machine	700	700	50
The bus	The radiator	840	700	78
The bus	Cold machine	840	600	63

4.3 DUAL Module Method Result Analysis

Using the DUAL module calculation method, we calculated the distribution of refrigerant temperatures of the intercooler and water tank. The pressurized air in the intercooler flowed in the negative direction, while the internal refrigerant in the water tank also flowed in the negative direction. The DUAL module calculation method had a comparatively accurate simulation of internal refrigerant temperature distribution. The cool air underwent temperature change when it flew through the intercooler. The air was significantly heated in the intercooler, and the closer it was to the inlet of pressurized air into the intercooler, the more heated it got. A high temperature point on the right section was noticed and caused by some back-flow at the top of the intercooler in the direction of automobile height. In terms of the sectional distribution of temperature of the water tank, the cool air was significantly heated at the inlet of cooling water. The cool air temperature, before the water tank, increased following the negative direction. This was due to that the internal refrigerant of the front-end intercooler flowed in the negative direction, while the outlet cool air temperature decreased gradually in this direction as well. The right section witnessed higher temperature because of the backflows at the top of the intercooler in the direction of the automobile height. Based on the above discussion, we arrived at the conclusion that DUAL dissipation module method not only enables an accurate distribution of the temperature of a simulative heat exchanger, but also provides the primary calculations of outlet temperature of the internal refrigerant. This method directly integrates the results of one-dimension calculations with the three-dimension calculations.

5. Conclusion

This paper simulated one-dimensional heat management calculations and obtained data such as size of intercooler and water tank and heat dissipation power so as to provide initial boundary conditions of three-dimensional simulation. The results of the one-dimensional calculation indicated that the design of truck heat exchanger was proper while the design for bus heat exchanger only met the boundary requirements of design. We analyzed the heat distribution features of automobile heat exchangers and provided improvement on heat radiation using the DUAL model, which can effectively simulate the spatial distribution of heat of the automobile heat exchanger in three-dimensional calculations. The research lays good foundation for interactive influence of the spatial location of heat exchanger, and at the same time, the DUAL module simulation can directly output the outlet temperature of the internal refrigerant of the heat exchanger.
In actual flows, many elements can cause the vortex viscosity of the flows. As a result, in order to have accurate simulation of the automobile flow fields, experiment is required and improvements on the turbulence model should be made based on the experiments.

Reference

Dominik, S., Leif, K., 2017, Building a large database of facial movements for deformation model-based 3d face tracking, Computer Graphics Forum, 36(2), DOI: 10.1111/cgf.13080

Frate G.F., Antonelli M., Desideri U., 2017, A novel pumped thermal electricity storage (ptes) system with thermal integration. Applied Thermal Engineering, 121, DOI: 10.1016/j.applthermaleng.2017.04.127

Kakavand H., Molana M., 2018, A numerical study of heat transfer charcteristics of a car radiator involved nanofluids, Heat Transfer—asian Research (1), DOI: 10.1002/htj.21292

Lee J., Kim H., 2017, Correlation analysis among engine room components for improving the cooling performance of gasoline vehicles, Transaction of the Korean Society of Automotive Engineers, 25, DOI: 10.7467/ksae.2017.25.4.516

Righetti G., Zilio C., Mancin S., Longo G.A., 2017, Heat pipe finned heat exchanger for heat recovery: experimental results and modelling, Heat Transfer Engineering (13), DOI: 10.1080/01457632.2017.1358483

Shiga T., Fujii M., 2017, Power reduction of micro-electro-mechanical system shutter display by temporal smoothing of image signal, Journal of the Society for Information Display, DOI: 10.1002/jsid.567

Sukkar G.A., Al-Sharif L., Mansour M., Gharbieh M., Farraj E., Jarrah R, 2017, Reconciling the value of the elevator round trip time between calculation and simulation, Simulation Transactions of the Society for Modeling & Simulation International, DOI: 10.1177/0037549717698014

Thaker S., Oni A. O., Kumar A., 2017, Techno-economic evaluation of solar-based thermal energy storage systems, Energy Conversion & Management, 153, 423-434, DOI: 10.1016/j.enconman.2017.10.004