A Study on the Capacity of a Cooling Medium to Dissipate Heat in Armored Vehicles in a Plateau Environment

Lijun HAN*, Jianmin LIU, Pukai WANG,Yi DONG, Xinyong QIAO, Xiaoming ZHANG

Department of Mechanical Engineering, Academy of Armored Forces Engineering, Beijing, 100072, China

Abstract **— In this paper a Computational Fluid Dynamics (CFD) model of the engine compartment and a 1-D model of the radiator are established and verified experimentally for a plateau of 3700m. The results agree well with results of experiments. Furthermore, the effects of environmental pressure and temperature on the capacity for dissipating heat of the cooling medium are obtained as follows: i) the capacity for dissipating heat decreases by nearly 46% as the altitude increases from 0m to 5000m, while ii) it decreases by nearly 70% as the temperature increases from 0℃ to 40℃. Moreover, the approximation model is designed to comprehensively reflect the capacity for dissipating heat influenced by the environment.**

Keywords - Capacity for dissipating heat; plateau test; model verification ; coupling calculation; regularity analysis

I. INTRODUCTION

The cooling system is one of the most important compartments for the vehicles, which is the important assurance of the vehicular reliability. The typical cooling system of vehicles mainly consists of two parts: the fan providing motive force for the air flow around the radiator and the radiator providing cooling fluid around the engine [1].

Many researchers have conducted an analytical study on the capacity for dissipating heat of vehicles. QU Xiao hua, et al[2], established a CFD model for a car heating radiator and test on the radiator was conducted for verification. They claimed that the performance of the automotive heating radiator, influenced by the air mass flow and water flow, was predicted. WANG Qin, et al [3], established a model of single water-cooling flow channel and had a research on the effects on fluid outlet temperature between different inlet velocity and air heat transfer coefficient, and thermal deformation. QIN Si cheng, et al[4-5], studied the performance law of a construction vehicle radiator under different working conditions and ensured its working stability. C. Oliet, et al[6], designed a heat exchanger model and had a set of parametric studies performed on radiators: mass flows, inlet temperatures and geometrical parameters (fin pitch and louver angle). Akhilnandh Ramesh, et al[7], had modified radiators and improved its efficiency. The flow characteristics and thermal performance by means of numerical investigation and experiments were carried out.

However, in plateau region, over-heating problems for the radiator and surfaces of other parts become more profound, even with other faults. The dissipating capacity of cooling system deteriorates as the altitude increases due to the reduction of the ambient pressure and density, especially for armored equipments, which are equipped with high power-density diesel engines. In China, the mountain area accounts for 60% of the total land of area: about 58% over 1000m above sea level, 33% over 2000m above sea level, and 26% over 3000m above sea level[8].

Therefore they are bound in plateau region and it is necessary to research on the regulars of the dissipating heat capacity for the armored vehicle diesel. In this paper, a CFD model of engine compartment and a 1D model of the radiator are established. The established models are validated by experiments. The results agree well with experimental results. Furthermore, the effects of environment on the capacity for dissipating heat of cooling medium in plateau environment are analyzed. The technical approach for studying in details is shown in Fig.1.

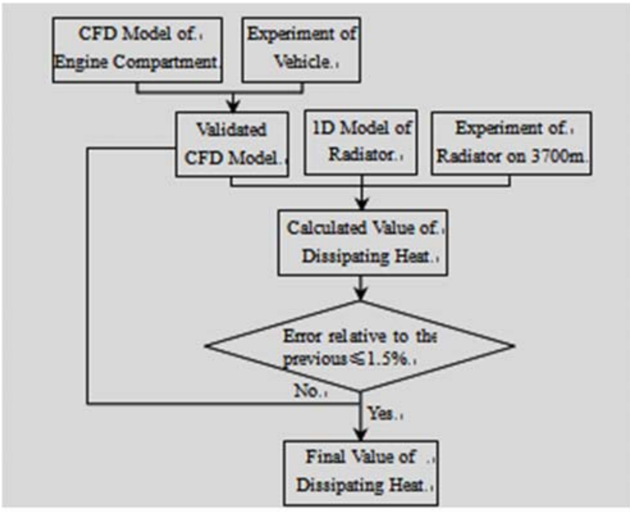

Figure 1.The technical approach for the research.

II. ESTABLISHMENT AND VALIDATION OF CFD MODEL

A. Geometry Model

The accuracy geometry model of engine compartment is the foundation for fluid calculation. In this paper, the assemble model of engine compartment is established on the base of the geometry size and assembly size. As shown in Fig.2, the assembly model consists of engine, transmission device, box of engine oil, shell of engine compartment, intake-louver and exhaust-louver, et al.

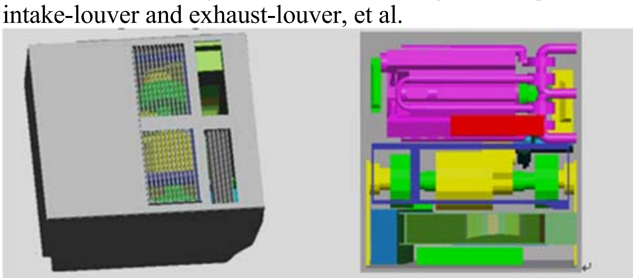

Figure 2.Geometry model of engine compartment

B. Calculating Regions

The external flow area, on one hand, can effect on the flow and temperature distributions of coolant medium, on the other hand, can mainly effect on the calculating time and resources. In this paper, the mass flow of fan is taken as the target to obtain the external height, width and length, whose results are shown in Fig.3.

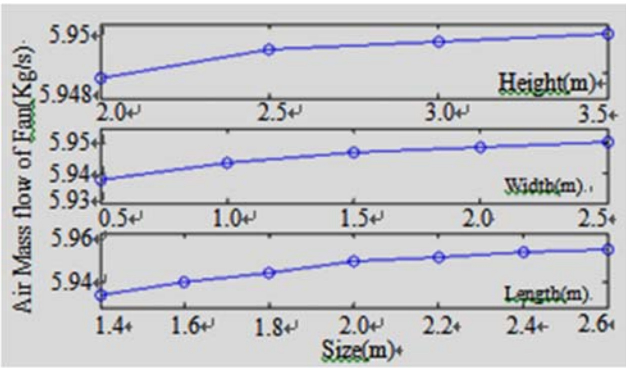

Figure 3.Air mass flow of fan with different calculating regions

The result indicates that the air mass flow of fan has a little change with the external calculating region (height 2.5 m, width 2.0m and length 2.4m). Structured and unstructured grids are employed in the CFD model, which is divided into 11 regions with 5080679 elements and 1156278 nodes.

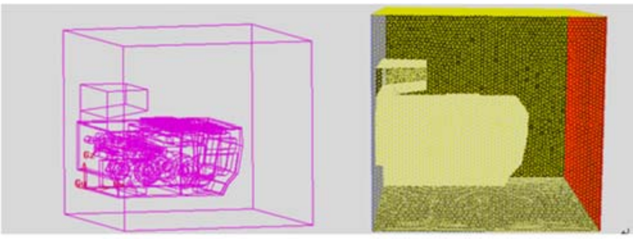

Figure 4.CFD model of the engine compartment

C. Boundary

Boundaries are the most important for the CFD model to obtain accurate results. As the length limit for this paper, two most important boundaries are given below in details: radiator and fan[9].

1) Boundary of Radiator

In general, a CFD model of the radiator takes too much time and too many calculating resources, which is unnecessary especially in term of the flow of the engine compartment. In this paper, the radiator boundary is simplified into a surface， whose boundary could be described by the heat flux and the polynomial function of the loss coefficient of the radiator.

$$k_{\mathrm{L}} = \sum_{n=1}^{N} r_{\mathrm{n}} v^{n-1} \tag{1}$$

where k_{L} is the loss coefficient, v is the fluid velocity perpendicular to the radiator, r_{n} is polynomial coefficients.

The function between the loss coefficient of the radiator and the air velocity, based on the data from the designing parameters and experiments, is described as Eq.(2).

$$k_{\mathrm{L}} = 32.825\text{-}3.3709\, v + 0.1204\, v^2 \tag{2}$$

2) Boundary of Fan

In general, the accurately internal flow of the fan is not mainly concerned for the engine compartment. In this paper, in order to reduce the computational resource, the fan boundary is simplified into a surface，considering the affect on the pressure distributions in the flowing field from the rotating effect of the fan. The fan model could be described by the function between the pressure jump from the rotating blades and the air normal-velocity：

$$\Delta P = \sum_{n=1}^{N} f_{\mathrm{n}} v^{n-1} \tag{3}$$

where ΔP is pressure jump, f_{n} is multinomial coefficient, v is normal-velocity.

The function between the pressure jump and the normal-velocity, based on the data from the experiments when the velocity of engine is 2 000 r/min, is described as Eq.(4).

$$\Delta P = -2.7741 v^2 - 23.7175 v + 3427.889 \tag{4}$$

Furthermore, in order to describe the affect on the flow distributions from the fan, the tangential-velocity and radial-velocity is given by the equations blow.

① Equation for tangential-velocity U_θ of coolant air is given by

$$U_\theta = \frac{\pi D n}{60} = \frac{\pi n}{30} r \tag{5}$$

where D is the fan hub dimension, n is the fan rotational speed.

② Equation for radial-velocity U_r of coolant air is given by

$$U_r = q_V / S \tag{6}$$

where S is the exhaust gas area of fan.

D. Control Equations

The flow and thermal transmission can be described by the 3-D turbulent control equation of viscous compressible flow, which consists of mass conservation equation, momentum conservation equation, and energy conservation equation.

The mass conservation equation is given as

$$\frac{\partial}{\partial x_\mathrm{i}}(\rho u_\mathrm{i}) = 0 \tag{7}$$

where ρ is the air density, u_i is the mean velocity in the direction of i, x_i is the rectangle coordinate, i is the subscript representing three directions of rectangular coordinates system, of which x is the opposite moving orientation of vehicles, y is the width orientation of vehicles, and z is the vertical orientation of vehicles.

The momentum conservation equation is given as

$$\frac{\partial}{\partial x_\mathrm{j}}\left(\rho u_\mathrm{i} u_\mathrm{j}\right) = \frac{\partial}{\partial x_\mathrm{j}}\left(\mu + \mu_\mathrm{t}\right)\left(\frac{\partial u_\mathrm{i}}{\partial x_\mathrm{j}} + \frac{\partial u_\mathrm{j}}{\partial x_\mathrm{i}} - \frac{2}{3}\frac{\partial u_\mathrm{i}}{\partial x_\mathrm{i}}\delta_\mathrm{ij}\right) - \frac{\partial}{\partial x_\mathrm{j}}\frac{2}{3}\rho k \delta_\mathrm{ij} - \frac{\partial p}{\partial x_\mathrm{i}} + S_\mathrm{Fi} \tag{8}$$

where p is pressure, δ_ij is a factor(if i=j, δ_ij is 1; while i≠j, δ_ij is 0), μ is dynamic viscosity, S_Fi the energy source term in the direction of i, μ_t is the turbulent dynamic viscosity. The standard $k-\varepsilon$ model is given as

$$\mu_\mathrm{t} = \rho c_\mu \frac{k^2}{\varepsilon} \tag{9}$$

where c_μ is a constant.

The energy conservation equation is given as

$$\frac{\partial}{\partial x_\mathrm{i}}\left(\rho H u_\mathrm{i}\right) = \frac{\partial}{\partial x_\mathrm{j}}\left\{\left(\lambda + \frac{c_\mathrm{p}\mu_\mathrm{t}}{\sigma_\mathrm{t}}\right)\frac{\partial T}{\partial x_\mathrm{j}} + u_\mathrm{i}\left(\frac{\mu}{Pr} + \frac{\mu_\mathrm{t}}{\sigma_\mathrm{t}}\right)\left[\left(\frac{\partial u_\mathrm{i}}{\partial x_\mathrm{j}} + \frac{\partial u_\mathrm{j}}{\partial x_\mathrm{i}}\right) - \frac{2}{3}\frac{\partial u_\mathrm{i}}{\partial x_\mathrm{i}}\delta_\mathrm{ij}\right]\right\} + S_\mathrm{h} \tag{10}$$

where H is the enthalpy, λ is the thermal conductivity number of air, T is temperature, c_p is heat capacity at constant pressure， σ_t is the energy turbulent Parandtl number, Pr is the Parandtl number, and S_h is the energy source term.

The turbulence kinetic energy of fluid per unit mass is given as:

$$k = \frac{1}{2}\overline{u_\mathrm{i}'u_\mathrm{i}'} \tag{11}$$

where u_i' is the root-mean-square value of fluctuation velocity.

E. Validation of CFD Model

The accuracy of CFD model is validated by the comparison of wind speed results in 10 test points, which are shown in Fig.5.

Experiment place: Beijing, China (0m); Temperature: 3℃; Speed of engine: 2000r/min; Vehicle condition: in-place and no-load.

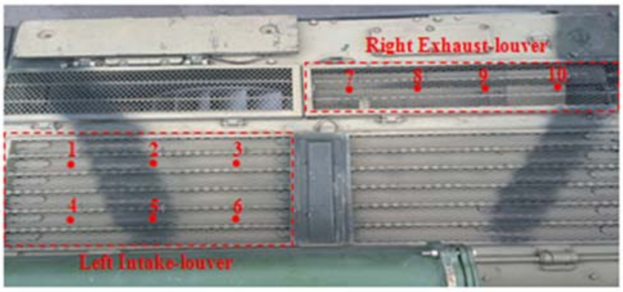

Figure 5.Verification experiment of CFD model

As shown in Fig.6, the results of simulation and experiments match very well with each other. The mean relative errors of intake-louver and exhaust-louver are 6.15% and 3.72%, while the maximal relative errors are 10.48% and 4.43%, which are in the requirements of engineering analysis. Thus, the CFD model could be used for further relevant fluid calculations.

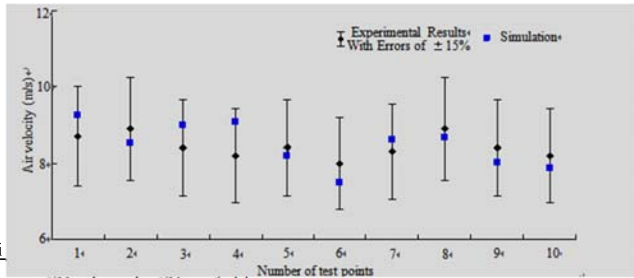

Figure 6.Verification experiment of CFD model

III. ESTABLISHMENT AND VALIDATION OF THE RADIATOR MODEL

A. Establishment of 1D Radiator model

The radiator consists of 4 flow circuits, totally 714 water pipes and 104 air-cooling fins. And For the air-cooling, the cross-section of fins is similar to the ladder trapezoid with surface ripples, which contribute to 1.364 times of surface areas compared with the smooth surface.

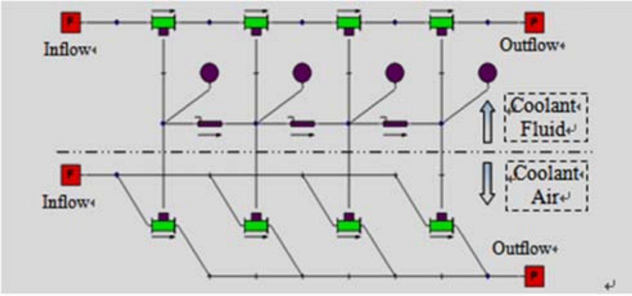

Figure 7.1D model of radiator

where is a thermal bridge, is a solid bar taken as the thermal resistance, is a flow source, is a pressure source, is a point mass.

B. Validation of 1D Radiator Model

Experiment of the capacity for dissipating heat of cooling medium is carried out in Tibet plateau(3700m), and the testing data consist of volume flux of inflow, the temperatures of inflow and outflow, as shown in Fig.8.

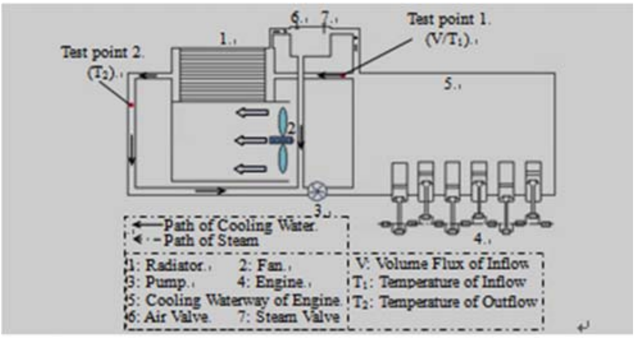

Figure 8.Experimental plan of the capacity for dissipating heat

The capacity for dissipating heat of the radiator can be obtained by the Eq.(12) below.

$$Q = V * \left(\rho_1 * Cp_1 * T_1 - \rho_2 * Cp_2 * T_2\right) \quad (12)$$

where Q is the capacity for dissipating heat, V is the volume flux of inflow, ρ_1 is the fluid density of outflow, ρ_2 is the fluid density of inflow, Cp_1 is the fluid specific heat of outflow, Cp_2 is the fluid specific heat of inflow, T_1 is fluid temperature of outflow, and T_2 is fluid temperature of outflow.

The model of radiator is established by the software FLOWMASTER based on the turbulent fluid flow and heat transfer empirical in tubes, which can be given as

$$n_u = a \mathrm{Re}^b Pr^c$$

$$n_u = \frac{HD}{\lambda}, \quad Pr = \frac{\mu C_\mathrm{p}}{\lambda}, \quad \mathrm{Re} = \frac{uD}{\upsilon}$$

(13)

where a, b and c are the constants, D is characteristic dimension, H is convective heat transfer coefficient, μ is dynamic viscosity, C_p is the specific heat at constant pressure, λ is thermal conductivity coefficient of coolant fluid, and υ is kinematic viscosity.

Eq.(13) is based on the turbulent fluid flow and heat transfer empirical correlation in tubes, which consists of three constants and physical equations with physical properties. In general, the constants are obtained from a large number of experiments. However, it is unapplied for the coolant air of the radiator, because of the air-cooling fins with surface ripples. Thus the established model should be updated by experiments.

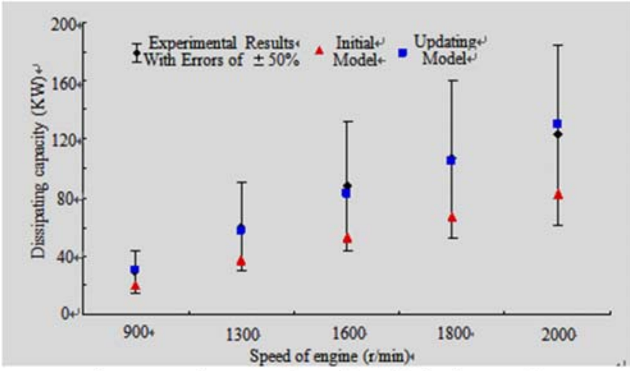

Figure 9.Results comparison of heat dissipating capacity

As shown in Fig.9, there is a big difference between the results from the initial model and experiments, with the maximal relative error 39.68%, the minimal relative error 32.41% and the mean relative error 35.95%. Obviously, the initial model could not be used for further research. According to the model updating method mentioned above, better results could be obtained from the updating model. The maximal relative error, the minimal relative error and the mean relative error are 6.33%, 1.53% and 4.182%, which could match very well with the results of experiment in consideration of many factors causing errors.

IV. PREDICT OF THE CAPACITY FOR DISSIPATING HEAT

The environmental pressure and temperature are the most important of all the factors influencing on the capacity for dissipating heat, which is more obvious in high plateau. In this paper, two main factors are taken into consideration. Fig.10 and Fig.11 illustrate the capacity for dissipating heat with the speed of 1600r/min and 2000r/min on 3700m.

As shown in Fig.10, the results indicate that: the capacity for dissipating heat decreases by 45.89% from 124KW to 67.1 KW at the engine speed of 1600r/min as the altitude increases from 0m to 5000m, while the capacity for dissipating heat decreases by 45.47% at the engine speed of 2000r/min.

Fig.11 indicates that: the capacity for dissipating heat decreases by 70.74% from 135KW to 39.5KW at the engine speed of 1600r/min as the temperature increases from 0℃ to 40℃, while the capacity for dissipating heat decreases by 63.5% at the engine speed of 2000r/min.

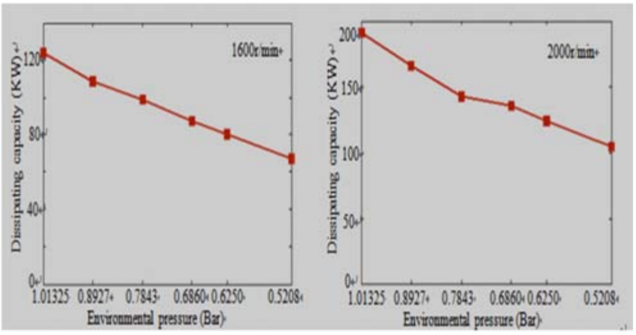

Figure 10.Results of heat dissipating capacity with pressure

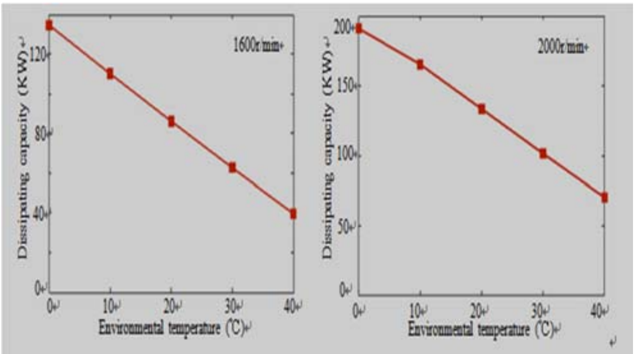

Figure 11.Results of heat dissipating capacity with temperature

In the study, the approximation model is designed to comprehensively describe the capacity for dissipating heat influenced by the environment. Thus polynomial function and radial basis function[10] are employed to build equivalent model. The model of the Speed 2000r/min is taken as an example.

The polynomial function can be given as

$$y = \alpha_1 f(x_1) + \alpha_2 f(x_2) + \cdots + \alpha_n f(x_n) \tag{14}$$

where α_i is the coefficient.

The radial basis function can be given as

$$y = \sum_{i=1}^{n} \beta_i f(x_i) + \sum_{i=1}^{m} \lambda_i \varphi(\| x - x_i \|) \tag{15}$$

where β_i is the coefficient, $\varphi(\|\cdot\|)$ is the basis function, and λ_i is the weight number.

In this paper, two appraisal indices are introduced to validate the approximation model: the maximum relative error($MaxErr$) and the root mean square error($RMSE$).

Fig.12 illustrates the approximation models of the capacity for dissipating heat influenced by the environment with the appraisal indices in Tab1.

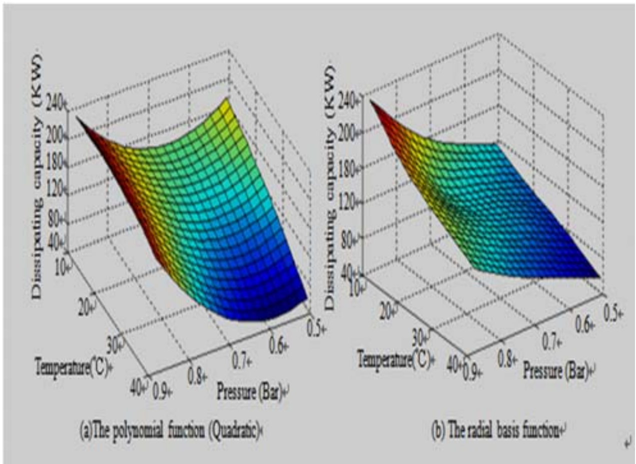

Figure 12.The approximation models for the speed of 2000/min

Tab.1 The appraisal indices of approximation model

Speed of engine(r/min)	Approximation Model	*MaxErr*	*RMSE*
2000	The polynomial function(Quadratic)	0.46147	0.24761
	The radial basis function	**0.12486**	**0.1563**

Two appraisal indices indicate that the maximum relative error and the root mean square error of the radial basis function are smaller than the polynomial function (Quadratic) ,which means that the radial basis function method has a better capacity to describe the capacity for dissipating heat influenced by the environment.

V. CONCLUSIONS

(1) A CFD model of the engine compartment and a 1D model of the radiator are both established in this paper. And the capacity for dissipating heat of cooling medium is obtained via simulation. Both models are validated by experiments. The simulation results match well with the experimental results.

(2) The capacity for dissipating heat decreases by approx 46% as the altitude increases from 0m to 5000m, while the capacity for dissipating heat decreases by approx 70% as the temperature increases from 0℃ to 40℃. The approximation model is designed and the radial basis function method has the better capacity to describe the capacity for dissipating heat influenced by the environment.

(3) Actually the capacity for dissipating heat is affected by many other factors such as coolant pumping power, coolant flow, coolant inlet temperature, fin pitch and louver angle, et al. Many further researches should be made for comprehensive results.

REFERENCES

[1] YAO Zhongpeng, WANG Xinguo. Vehicles Cooling Heat Transfer [M].Beijing: Press of Beijing Institute of Technology,2001

[2] Qu Xiaohua, SHI Junye, CHEN Jiangping, et al. Numerical Model for Automotive Heating Radiator [J]. Automotive engineering, 2009, 31(4):345-352

[3] WANG Qin, YANG Lian fa, ZHANG Zhen. Analysis of Thermal Field and Thermal Deformation of Water-cooling Radiator by Finite Element Simulation [J]. FLUID MACHINERY, 2011, 39(1):83-86

[4] LIU Jiaxin, QIN Sicheng, XU Zhenyuan, et al. Numerical Simulation of Heat Exchange Performance of Radiator Module in Construction Vehicles [J]. JOURNAL OF SOUTHWEST JIAOTONG UNIVERSITY, 2012, 47(4):623-628

[5] LIU Jiaxin, QIN Sicheng, XU Zhenyuan, et al. Improvement and Analysis of Heat Exchange Performance of Vehicle Radiator Module in Virtual Tunnel [J]. Journal of Jilin University (Engineering and Technology Edition), 2014, 44(2):330-334

[6] C. Oliet, A. Oliva, J. Castro, et al. Parametric Studies on Automotive Radiators [J]. Applied Thermal Engineering, 2007, 27:2033–2043

[7] Akhilnandh Ramesh, M. Jaya Arun Prasanth, A.Kirthivasan, et al. Heat Transfer Studies on Air Cooled Spiral Radiator with Circumferential Fins [J]. Procedia Engineering, 2015, 127 :333 -339

[8] ZHANG Weihua, HAN Zhiqiang, DOU Shoujian. Research on Zoning of Natural Environment for Military Vehicle, 2015, 10(5):26-30

[9] BI Xiao ping, LU Liangdong,Sicheng, ZHOU Guoyin. Evaluation of Cooling Air Duct Comprehensive Performance in Vehicle Based on CFD Analysis [J]. Chinese Internal Combustion Engine Engineering , 2012, 33(3):49-53

[10] KIRKPATRICK S, GELATT C D. Optimization by simulated annealing [J]. Science, 1983, 220:671-680.