Research Journal of Applied Sciences, Engineering and Technology 7(17): 3425-3432, 2014
DOI:10.19026/rjaset.7.693
ISSN: 2040-7459; e-ISSN: 2040-7467

Submitted: January 22, 2013 Accepted: March 02, 2013 Published: May 05, 2014

Research Article

Heat Transfer Research of Gas-solid-liquid Three Phase Coupling of EGR Cooler

[1]Fu-Wu Yan, [1]Yan-Lei Liu, [2]Zhi-En Liu and [2]Ding-Yian Fu
[1]School of Automobile Engineering, Wuhan University of Technology, Wuhan, China
[2]Hubei Key Laboratory of Advanced Technology of Automobile Parts, Wuhan, China

Abstract: The main aim of the study is to get the temperature and backpressure of a car engine exhaust gas which goes through the EGR-cooler. So the internal fluid flow and heat transfer process of the EGR cooler must be studied more clearly, numerical simulations are applied. Based on the strong coupling method, gas-solid-liquid three phases coupling model of the typical heat transfer unit is established. According to the coupling result, the heat flux of the tube's outside surface is gained and then mapped to the inner surface of the cooler's water. The water model is set up based on the separation coupling method. According to the analysis of the calculation, the detailed pressure and temperature distribution of the gas, water and solid are obtained. From the distribution cloud, we know the changes of the parameters along the fluid flows streamline.

Keywords: CFD, EGR-cooler, gas-the solid-liquid three phases coupling, heat transfer process

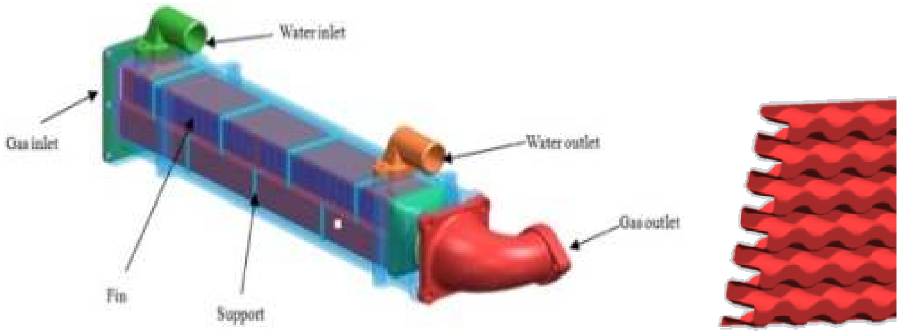
INTRODUCTION

Exhaust Gas Recirculation (EGR) technology is the most effective measure reducing diesel engine NO_X emissions. The engine exhaust in EGR cooler is cooled to a certain temperature and then is leaded back to cylinders to reduce the cylinders' maximum combustion temperature and pressure, thus the formation of the harmful gas NO_X is inhibited and the goal of reducing emissions is achieved. Therefore cooling effect of the cooler is very important to an EGR system.

At present, many domestic and overseas scholars have been researching the numerical simulation of the EGR cooler system. Gu (2007) and Huang *et al*. (2008) simulated gas-liquid two phases coupling in the cooler using boussinesq assumption. Gas pressure changes very little which has little influence on the density. So the process of the heat transfer was natural convection. Ignoring the thickness of the tubes's wall or the helical baffle, shell conduction model was carried as the relative wall thickness to calculate the heat transfer. Three different models of EGR cooler were investigated by Yu-qi Huang. The models were a traditional and two others which were improved by adding a helical baffle in the cooling area. The results showed that the improved structures not only lengthened the flow path of the cooling water, but also enhanced the heat exchange rate between the cool and hot media. The wall was ignored, but absorbed some heat, so the result had some difference with the truth.

Deng and Tao (2004) and Xie and Gao (2002) researched the fluid's flow and heat transfer in the cooler's shell side. They used the computing method of three dimensional, staggered structure that was based on porous media and distribution resistance. Volume penetration rate, surface penetration rate and distribution resistance etc., were used to describe the solid in the heat exchanger. The fluid and solid were simplified as the same control volume and then the influence on the solid was reflected by the conservation equation and the modified finite difference method. But this method had many deficiencies which are: Tube side flow can only be simplified to a one dimensional model because of the simplified solids, the result was not detailed; The calculation of the resistance and convection heat transfer coefficient need appropriate test correlations, the accuracy of correlations had large influence on the accuracy of the simulation results; For a complicated model, the values of the Volume penetration rate, surface penetration rate were different to get. Sun *et al*. (2006) applied the multi-component transmission model to analyze the thermal-fluid coupling of the cooler. Heat transfer characteristics, air flow characteristics were researched comprehensively and then some improvements were put forward to improve the performance of the heat exchanger. Fluid in this method can only be gas or liquid. If both the gas and liquid were existed, the method was not be used. So it was some limitations. Nasiruddin and Kamran (2007) studied the heat transfer augmentation in a heat exchanger's tube using a baffle. The effect of baffle

Corresponding Author: Yan-Lei Liu, School of Automobile Engineering, Wuhan University of Technology, Wuhan, China, Tel.: 13429814235



(a) EGR cooler integral structure (b) Partial enlarged detail of the fin

Fig. 1: EGR cooler's structure chart

size and orientation on the heat transfer enhancement was studied in detail. The results show: the vertical baffle, an increase in the baffle height causes a substantial increase in the pressure loss is also very significant; a baffle inclined towards the downstream side will have the minimum pressure loss. In this study, the numerical simulations were conducted in a two-dimensional domain, which can't describe the flow in detail.

On the basis of predecessors' studies, this study establishes three phases heat coupling model of gas-solid-liquid based on the strong coupling method. It simulates the fluid flow and the heat transfer of the typical heat transfer unit (High temperature gas in the two tubes and surrounding cooling water) of a cooler in EGR system. Because the gas is the high temperature compressed gas that density changes large with the pressure, so the idea gas law is applied to reflect the changing process of the gas density. According to the compute of the three phases coupling, the heat flux of the tube wall's outside surface is gained. The other tubes' heat flux is equal to the typical heat transfer unit's which is mapped to the water inner surface of the water. The water flow and heat transfer process of the cooling water are calculated. Analysis of the whole calculation: Detailed of the internal flow field and temperature field distribution are gained; The heat every domain absorbing or absorbed are all got, the heat of the solid is very little which can't be ignored. It provides theoretical basis for the EGR cooler's evaluation of the performance, design and optimization.

METHODOLOGY

Physical model:

EGR cooler's physical model: The EGR system cooler's structure studied in this study is shown in Fig. 1. This cooler is shell tube heat exchanger (Yu, 2006). Exhaust gas flows through tube space (called tube side) from the inlet end to the outlet of twenty parallel tubes (called bundle). Cooling water flows into the outer space of the tubes from water inlet and then outflows from the water outlet. Heat is transferred from the exhaust to cooling water through the tubes and fins. It is fair current. There are two corrugated fins which are "bow" font. At the same time, there are seven supports in the water area. All of these will make sure the water's flow path like the letter "S". So the areas where water contacts with the tubes are increased, which strengthens cooling effect. Cooled gas flows into a bend where it mixes and heat transfers and its temperature is basically uniform at the outlet. In order to obtain more accurate heat transfer relationship, this study adopts the strong coupling method of numerical simulation.

Basic theories: In the actual calculation and analysis, the main hypotheses used are listed as follows.

Ideal gas law: When it comes to the ideal gas, it needs to introduce two parameters which are total pressure and total temperature. The relation between total pressure and static pressure is:

$$\frac{p_0}{p_s} = \left[1 + \frac{\gamma-1}{2} M^2\right]^{\frac{\gamma}{\gamma-1}} \quad (1)$$

The relation between total temperature and static temperature is:

$$\frac{T_0}{T_s} = 1 + \frac{\gamma-1}{2} M^2 \quad (2)$$

Density formula of the ideal gas law (Yang and Jia, 2003) is:

$$\rho = {(p_{op} + p)}/{RT_s} \quad (3)$$

where, R is the universal gas constant, p_{op} is the operating pressure (Li *et al*., 2009) and p is the local static pressure. The density is only related to the operating pressure and the local static pressure. So for the compressible gas, the operating pressure influences its density according to the ideal gas law. The values could not be simply set to 0.

Continuity and momentum equation:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \quad (4)$$

Energy equation:

$$\frac{\partial}{\partial t}(\rho E) + \frac{\partial y}{\partial x_i}\left(u_i(\rho E + p)\right) = \frac{\partial}{\partial x_i}\left(k_{eff}\frac{\partial T}{\partial x_i} + u_j(\tau_{ij}|)_{eff}\right) \quad (5)$$

$$k_{eff} = k + kt \quad (6)$$

where, k_{eff} is the effective heat transfer co-efficient and k_t is the turbulent heat conduction coefficient. The right side of the formula (5) describes the transmission of energy which is brought by the heat conduction and viscous dissipation.

The energy equation of the solid area:

$$\frac{\partial}{\partial t}(\rho h) + \frac{\partial}{\partial x_i}(\rho u_i h) = \frac{\partial}{\partial x_i}\left(k\frac{\partial T}{\partial x_i}\right) + \dot{q}''' \quad (7)$$

where,

ρ = The solid's density
h = The sensible enthalpy
k = The coefficient of heat transfer
$\dot{q}'''$ = The volumetric heat source

Analyzing process of gas-solid-liquid three phases coupling: In the three phases coupling calculation, the wall function is adopted because $k-\varepsilon$ model is not applicable in near wall region. This method can directly get the variable value (Cell Value) of node which is adjacent to controlled volume, so it need not analyze flow in near wall area. Dimensionless parameters u^+ and y^+ represent the speed and distance in the function of near wall region as follow:

$$u^+ = \frac{u}{u_\tau} \quad (8)$$

$$y^+ = \frac{\Delta y \rho u_\tau}{\mu} = \frac{\Delta y}{\upsilon}\sqrt{\frac{\tau_w}{\rho}} \quad (9)$$

where,

u = The fluid flow time-averaged velocity
u_τ = The wall friction velocity
τ_w = The wall shear stress
Δy = The physical distance between the node and the wall

The temperature T^+ connects with the temperature T_w by the formula (10):

$$T^+ = \frac{(T_w - T_p)\rho C_p C_\mu^{1/4} k_p^{1/2}}{q_w} \quad (10)$$

where,

T^+ = The temperature of the computing grid node
T_p = The temperature of the controlled volume's node p which is next to the wall
T_w = The wall temperature
ρ = The fluid's density
C_p = The fluid's specific heat at constant pressure
q_w = The wall's heat flux

In the iterative calculation, every step will exchange the data and compare the residual value of the monitoring point with the set value. If the result meets the requirements, it will converge. If it converged as the date between gas and solid meet the converged condition, but not converged as the date between solid and liquid isn't meet the converged condition, iteration will go on until they both converged.

GAS-SOLID-LIQUID THREE PHASE HEAT COUPLING MODEL

The model is relatively complicated and it involves gas-solid-liquid three-phase heat coupling, which will cause a problem that the mesh quantity of EGR cooler will be very large. The existing computer is unable to complete. Upper tubes are supported by four supports while bottom tubes are supported by three supports and two fore-and-aft tubes don't distribute symmetrically. In consequence, the air, two fore-and-aft tubes and surrounding water are considered as typical heat transfer unit. Gas-solid-liquid three phase coupling about typical heat transfer unit is calculated and analyzed.

Geometric model: Heat coupling computational geometric field model is established by using 3D modeling software. The fin's structure is complicated and its thickness is only 0.1 mm. In order to mesh conveniently, the structure is simplified as follows: ignoring the fins' thickness which contacts with the wall; ignoring the fins' fillet. The established computational geometric field is shown in Fig. 2.

Discrete grid: Simplified model will be imported into CFD calculation software and then will be meshed. In

Table 1: Detailed boundary conditions

Boundary	Parameters setting
Inlet	Air, mass flow rate is 7.236 g/sec, temperature is 903 K
	Water, mass flow rate is 230 g/sec, temperature is 358 K
Outlet	Air, pressure is 100 kpa
	Water, pressure is100 kPa
Sym	Symmetry
Operating pressure	100 kPa
Outer wall	Natural convection, temperature is 300 K, convective heat transfer coefficient is 30

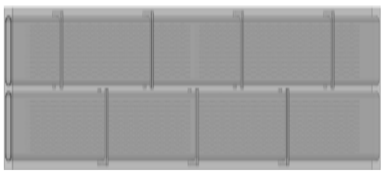
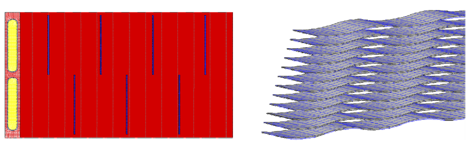


Fig. 2: Computational geometric field of the typical heat transfer unit



(a) The whole grid (b) Fins' grid

Fig. 3: Grid model

order to get ideal result and reduce computing time, the number of grid should be tried to reduce and the quality should be tried to improve. The computational domain is divided in hexahedron element. However the fillet area is divided in tetrahedron element. Due to involving the three phase heat coupling, the distance from the adjacent wall cells in nearly wall area to the wall should be in the scope of the rules of logarithmic wall, which meansy$^+$should be between 30 and 60. The junction of logarithmic area and fully turbulent vary with the pressure gradient and Reynolds. If the Reynolds number increases, the point will be away from the wall. But the boundary layer must have grid points. Because of the irregular structure, the model needs to be divided into relatively inerratic geometries then meshed respectively. Finally, the whole model will be divided into about 1.42 million grid elements by the artificial control and the automatic generation of function of the software. So the number of water grid is about 0. 415 million; the number of gas grid is about 0. 91 million; the number of wall grid is about 52 thousand; the number of fin grid is about 36 thousand; the number of support grid is about 7 thousand. The grid model is shown in Fig. 3. The grid in different areas is distinguished by different colors. The contact walls of the different regions are set as interface that will make the adjacent areas exchange data.

Boundary conditions and computation model: The gas in the cooler is high temperature compressed gas. Because the gas's temperature changes greatly, ideal gas law must be used to reflect the relationship between the density and the temperature in Fluent software. Establishing the model, the key in the process of heat transfer is gas-solid-liquid three phase coupling. The influence of the expandable structure could be ignored when it is heated and the steam evaporated by the water.

In order to simplify the mathematical model, gas (except density) and water physical properties can be seen as the same with normal physical properties. The density of the gas uses the ideal gas law. The material of tubes, fins and supports is all $022Cr_{1}7Ni_{12}MO_{2}$, its physical properties can be got by the National standard (Law of the People's Republic of the State Administration of Quality Supervision, 2007).

In order to use CFD software to calculate the flow field, the implicit solver based on pressure Realizable $k-\varepsilon$ model and standard wall functions are used. The interface of the two phases should be set as coupling that makes the surface result of junction heat couples. Detailed boundary conditions are shown in Table 1.

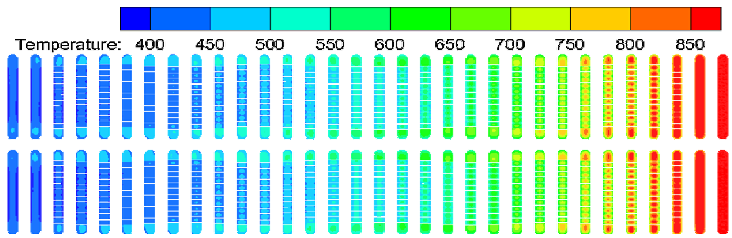
RESULTS

When iteration converged, this study gains the calculation results of the cooler internal flow field about pressure and temperature distribution and so on. There are air, water and solid three phases in the model. We need to analyze their pressure and temperature distribution cloud.

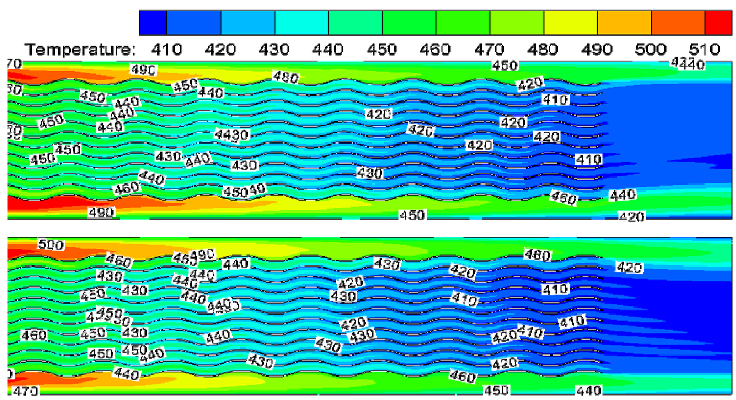
Flow field analysis of gas: According to the calculation result, the backpressure of the gas is 6.5 kPa; average temperature difference is 488.67 K; the average temperature of the gas outlet is about 412.17 K; the gas inlet and outlet density are 0.78 and 1.70 kg/m^3, respectively. According to Fig. 4, pressure reduces gradually along the gas's flow direction. The pressure distribution is definitely symmetrical. In the direction of vertical, the pressure near the middle is lower. Near the upper and lower of the tube's wall, the heat of the gas transfers heat through the wall. In the place of the relatively middle, temperature is higher, that is, the cooling effect is poorer. Around the fins, gas's heat transfers to the fin and then to the wall. Gas's temperature is lower obviously. According to the ideal gas law, the density of the gas is got by the formula (3). The range of the temperature is 450°C. So the temperature has larger influence on the density: the higher the temperature, the smaller the density.

According to Fig. 4d, the density increases gradually along the flow direction and the middle gas's density is relatively higher in the direction of vertical.

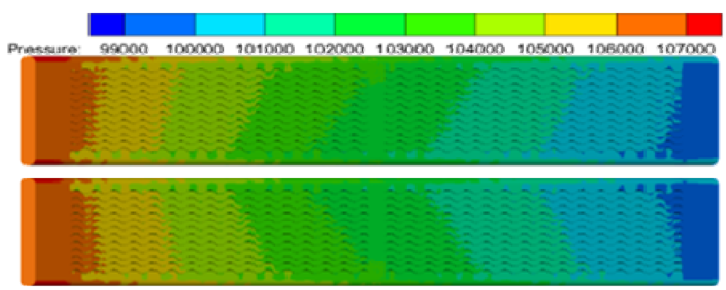
Flow field analysis of water: The average temperature difference between the inlet and outlet is 8.1 K; the backpressure is 6.6 kPa. According to Fig. 5a and b, the pressure decreases gradually and the temperature rises along with the flow direction. The temperature is much higher in some areas which concentrate on the side of the supports. The high temperature area appears behind the first support in the inlet, the temperature is as high as 373 K. The heat gathers behind the first support so that the water is heated to vapor. The main reason is the backflow. If the first support's thickness decreases, the high temperature will be avoided. On the one side of the other supports, the backflow appears and the temperature is higher. According to Fig. 5c, water's



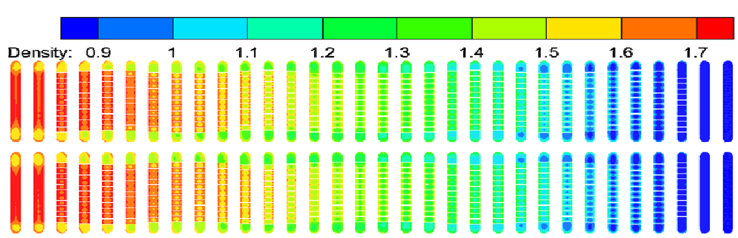
(a) Temperature slices diagram



(b)Temperature enlarged diagram in the outlet

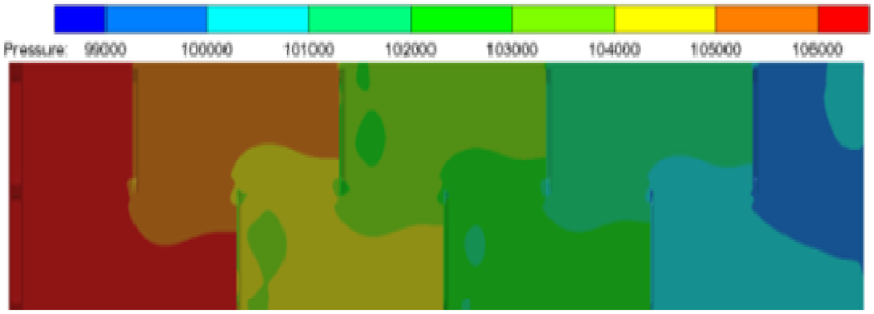


(c) Pressure distribution cloud

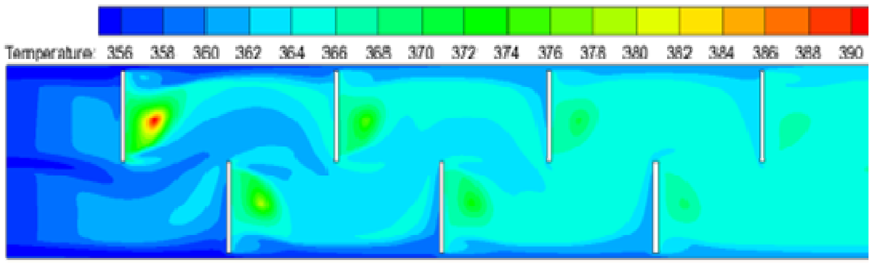


(d) Density slices diagram

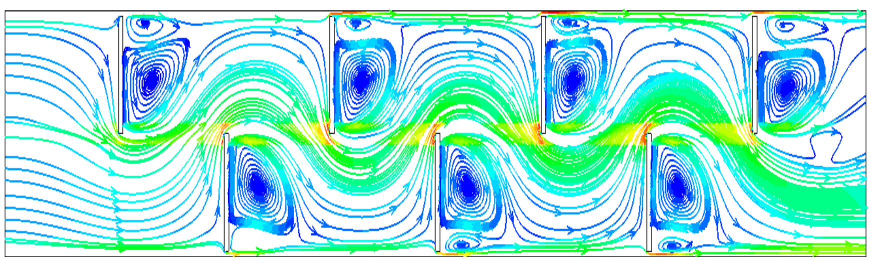
Fig. 4: Distribution cloud of the gas region



(a) Pressure distribution cloud



(b) Temperature distribution cloud of the section



(c) Flow diagram of the section

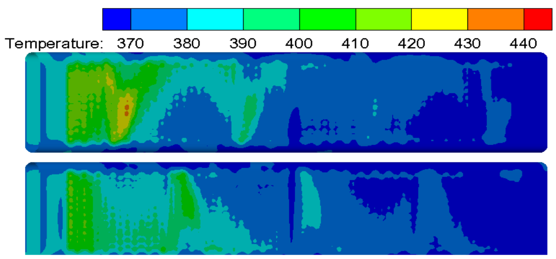
Fig. 5: Distribution cloud of the water regional

streamline is "S" distribution. Most of the water flows along ‘S’. But a small part of the water flows through the space which is between the supports and outer wall, Backset appears on the side of the support and gas’s temperature changes a little.

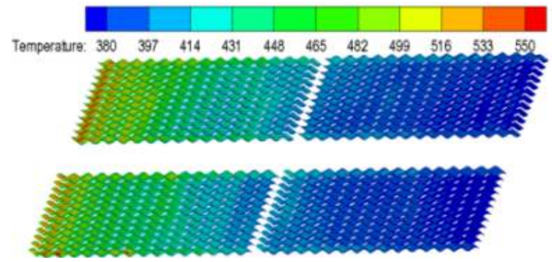
Flow field analysis of solid: According to Fig. 6a, the difference between the highest temperature and the lowest temperature of the tube’s wall is nearly 70 K. The temperature is higher in the area contacting with the fins. The temperature of place which doesn’t contact with fins is significantly lower. According to Fig. 6b, fin’s temperature decreases gradually nearly 170 K along the fluid flow direction. The temperature of fin in the center of the vertical direction is higher and distributes asymmetrically. It suggests that the heat gas sent out is partly absorbed by the tubes and fins. Supports also have obvious temperature difference. The temperature of the place which contacts with the tubes is as high as 420 K. The temperature of the wall that contacts with the tube reduces gradually along the flow direction. But water’s temperature rises gradually. This shows that the heat is transferred from high temperature gas to the tubes and supports (Fig. 6c).

Flow field analysis of the whole water area: According to the overall structure of EGR cooler, flow field model of the water’s area is set up. When meshing the model, tetrahedron element is used because of the irregular structure. According to the result of the coupling, the heat flux of water’s inner surface is 75751 w/m^2. This value will be mapped to the inner surface of the water. When calculating the flow field, the implicit solver based on the pressure, standard $k-\varepsilon$ model are used. The parameters are: The inlet velocity is 2.53 m/sec; Heat flux of the water inner surface is 75751 w/m^2; the outlet pressure is10 kPa.

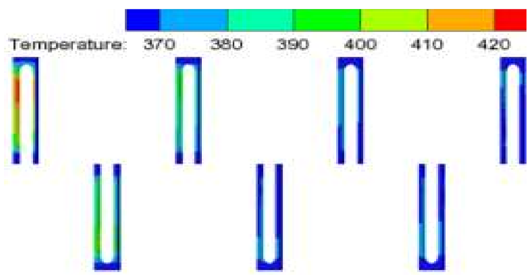
According to the calculation, backpressure is 10.1 kPa; temperature difference is 8.2 K. In the preceding calculation of three phases coupling, backpressure



(a) Temperature distribution of tube's wall

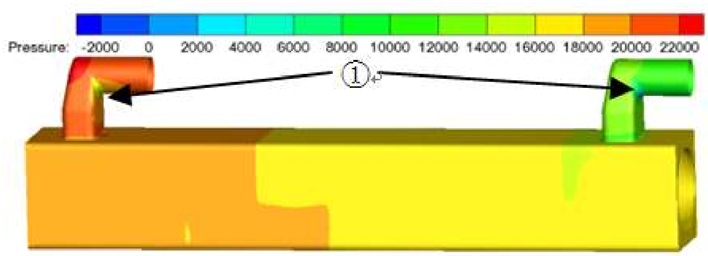


(b) Temperature distribution of fins

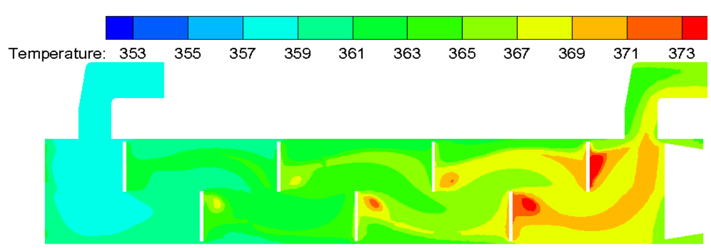


(c) Temperature distribution of supports

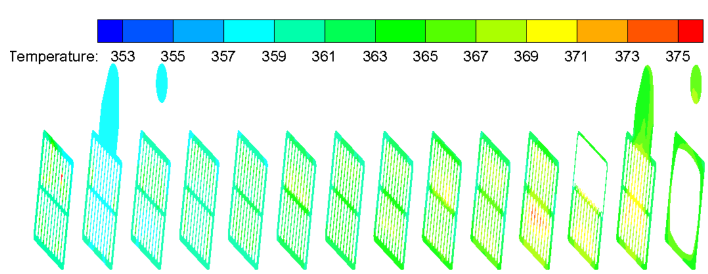
Fig. 6: Temperature distribution of the solid



(a) The overall pressure distribution diagram



(b) Temperature distribution diagram of center section



(c) The whole temperature slices diagram

Fig. 7: Water's distribution diagram

is 6.6 kPa; temperature difference is 8.1 K. Comparing the two situations, the difference in temperature difference is 0.1 K; but the backpressure raises 3.5 kPa. The main reason that raises the backpressure is that the model adds the pipes of water inlet and outlet which are shown in Fig. 1a. According to Fig. 7a, the pressure is lower in the inner corner of the inlet and outlet which is shown as ① in Fig. 7a. The pressure changes little in the area of inlet, outlet and convection heat transfer, but it changes big in the area where the sectional area changes. According to Fig. 7b and c, in the high temperature area of the inlet, the gas is difficult to contact with water, so heat convection is weaker, which causes water's temperature higher.

CONCLUSION

In this study, gas-solid-liquid three phase coupling method is used to calculate an EGR system cooler. This study analyzes the pressure field temperature field and density distribution of the gas solid and water's area. Besides, flow field the pressure field and temperature field of the whole water are analyzed. We get the conclusions as follows.

Based on the strong coupling method, this study establishes the typical unit model of gas-solid-liquid three-phase heat coupling. The influence of tube's wall fins and supports on the flow and heat transfer are taken into account. We get not only the flow field and temperature distribution of the fluid, but also the solid's temperature distribution. It reflects accurately the temperature distribution of the whole EGR cooler and it provides theoretical foundation for the structural design and optimization.

The ideal gas law embodies the density of the gas in the whole flow process. Pressure and temperature distribution diagrams reflect the influence which pressure and heat transfer process impact on density fluctuation.

From the strong coupling calculation, heat transfer relation (heat flux) is got. The separation coupling method is used to numerical simulate the water area of EGR cooler. So the result of numerical simulation is more accurate.

REFERENCES

Deng, B. and W. Tao, 2004. Three dimensional numerical simulation of turbulent flow and heat transfer characteristics in shell side of shell-and-tube heat exchangers. J. Chem. Ind. Eng., 55(7): 1053-1059.

Gu, G., 2007. CFD simulation and designing of heat exchanger in the EGR system [D]. M.A. Thesis, Huazhong University of Science and Technology, Wuhan, China.

Huang, Y., X. Yu and G. Lu, 2008. Numerical simulation and optimization design of the EGR cooler in vehicle. J. Zhejiang Univ., Sci., 9(9): 1270-1276.

Law of the People's Republic of the State Administration of Quality Supervision, 2007. Stainless and Heat-resisting Steels Designation and Chemical Composition. Inspection and Quarantine, China National Standardization Management Committee, China, GB/T 20878-2007.

Li, J., C. Li and R. Hu, 2009. FLUENT 6.3 Flow Field Analysis [M]. Chemical Industry Press, Beijing.

Nasiruddin, M.H. and S. Kamran, 2007. Heat transfer augmentation in a heat exchanger tube using a baffle [J]. Int. J. Heat and Fluid Fl., 28: 318-328.

Sun, D., Z. Deng, P. Wang and H. Luo, 2006. Numerical simulation and optimization on heat exchanger in small motor [J]. J. Chongqing Univ., Nat. Sci. Edn., 29(4): 50-53.

Xie, H. and Z. Gao, 2002. Three dimensional numerical simulation offlow in shell-and-tube heat exchangers [J]. Chinese J. Nucl. Sci. Eng., 22(3): 240-243.

Yang, J. and B. Jia, 2003. Variable property model for natural convection in a square cavity with temperature differences of 20-720K. J. J. Tsinghua Univ., Sci. Technol. Edn., 12(43): 1597-1660.

Yu, J., 2006. Principle and Application of the Heat Exchanger [M]. The Beijing University of Aeronautics and Astronautics Press, Beijing.