HEFAT2012
9th International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics
16 – 18 July 2012
Malta

SIMULATION, FABRICATION AND OPTIMIZATION OF THE VEHICULAR INTERCOOLER BASED ON FIELD SYNERGY PRINCIPLE

Yuqi Huang*, Xiaoli Yu, Zhentao Liu and Guodong Lu
*Author for correspondence
Power Machinery and Vehicular Engineering Institute,
Zhejiang University,
Hangzhou, 310027
P.R.China
E-mail: huangyuqi@zju.edu.cn

ABSTRACT

Heat exchangers are the core components of cooling system in vehicle. The performance of heat exchanger determines the effect of whole cooling system. The core in heat exchanger is a key section where undertake the most of heat exchange. Yet the structure of inlet pipe and tanks play a decisive role in the distribution of inside flow, which not only affect the internal flow resistance, but also impacts the overall efficiency.

The flow and heat transfer of a typical vehicular intercooler are simulated in this paper, and the calculation are validated by wind tunnel experiments. Two different tank models are fabricated and compared with the original one. Considering the practicability and feasibility, an improved model is designed to optimize the flow uniformity. It is found that the improved model significantly reduces the internal resistance while also maintain the proper heat exchange capability. In conclusion we suggest that the improved structures are more powerful than the traditional one.

Keywords: intercooler, tank, heat transfer, numerical simulation, wind tunnel

INTRODUCTION

The cooling system is an important assistance system to ensure the stability of the vehicle, and it has a direct effect to energy-saving and emission reduction. Heat exchangers are the core components of cooling system. The performance of heat exchanger determines the effect of whole cooling system. With the development of turbo-charging technique, intercooler is widely used in more and more vehicles at present, its cooling efficiency directly influence the working state of engine. Thus many scholars at home and abroad have made considerably detailed study on the flow and heat transfer characters of the intercooler.

The core in heat exchanger is a key section where undertake the most of heat exchange. Yet the structure of inlet pipe and tanks play a decisive role in the distribution of inside flow, which not only affect the internal flow resistance, but also impact the overall efficiency. Lalot etc.[1] have studied the influence of non-uniform flowing by experimental and numerical analysis in 1999. It’s found that there could be 25 percent lower of heat transfer efficiency with the fluid flow mal-distribution in a cross flow heat sink. Based on FLUENT program, Zhang etc. [2] have simulated the fluid flow distribution in plate-fin heat exchangers, compared with the experimental results [3]. The results show the flow mal-distribution is very serious in the conventional header. Subsequently, two modified headers are recommended and the flow distributions are definitely improved. The PIV experimental methods are adopted to investigate the flow characteristics in the header of a plate–fin heat exchanger in 2007[4], the results indicate that performance of fluid mal-distribution in a conventional header is very serious, while the improved header configuration with a punched baffle can enhance the uniformity of flow distribution, then increase the heat transfer efficiency accordingly.

On the basis of field synergy principle brought by Guo Zeng-yuan[5], the heat convection is liken to the process of heat conduction with internal heat source. It is considered that the performance of heat convection depend not just on the flow velocity, physical property and the temperature difference between fluid and solid, but also on the synergy of flow field and thermal flow field. The more synergetic, the more intense in heat transfer under the same velocity and temperature conditions. To the heat exchanger, the more uniform of the temperature difference field between hot medium and cold medium, the more efficiency. This theory explains the influence of internal flow uniformity to the heat exchange performance basically.

Based on field synergy principle, together with the application of CFD and the combination of wind tunnel experiments, the present study simulate the flow and heat transfer of a typical vehicular intercooler, fabricate different tanks model, present an effect optimization to improve the structure of header tank. The modified structure allowing a uniform distribution of hot medium in channels, thereby enhance the heat transfer efficiency with lower internal resistance.

NOMENCLATURE

f	[-]	Friction factor
V_c	[m/s]	Average velocity of the flow field
D	[m]	Hydrodynamic diameter of the fin
L_d	[m]	Entire flow length
ΔP_i	[Pa]	Pressure differential produced by the circulation area abruptly narrow
ΔP_e	[Pa]	Pressure differential produced by the circulation area abruptly widening
T	[K]	Temperature
F_h	[m]	Fin height
δ	[m]	Fin thickness
F_p	[m]	Fin pitch
b	[m]	Amplitude of the wavy-fin
L	[m]	Wavelength of the wavy-fin
$\overline{q}$	[W]	The heat exchange power
q_c	[W]	The heat absorbed by the cold air
q_h	[W]	The heat supplied by the hot gas
$\dot{m}_c$	[kg/s]	The mass flow rate of cold air
$\dot{m}_h$	[kg/s]	The mass flow rate of hot gas
$C_{p,c}$	[J/kg·℃]	The specific heat of cold air
$C_{p,h}$	[J/kg·℃]	The specific heat of hot gas
$q_{\Delta P}$	[W]	The pumping power dissipation
Q_m	[m^3/s]	Volume flow rate of hot gas

ORIGINAL MODEL DISCRIPTION

A plate-fin air to air intercooler shown as below is taken as original research object, with the size of the core is 300mm*300mm*80mm. Both mediums are air: hot, turbo-charged air flow inside, normal air flow along the Z-minus direction as cold medium. The heat between cold medium and hot medium is transferred by partition boards and fins. The thickness of partition boards is 0.6mm and the fin thickness is 0.2mm. Considering the dirt and dust in normal air, the wavy-fin and plane-fin are adopted respectively in the hot- and cold-passes to avoid dust accumulation.

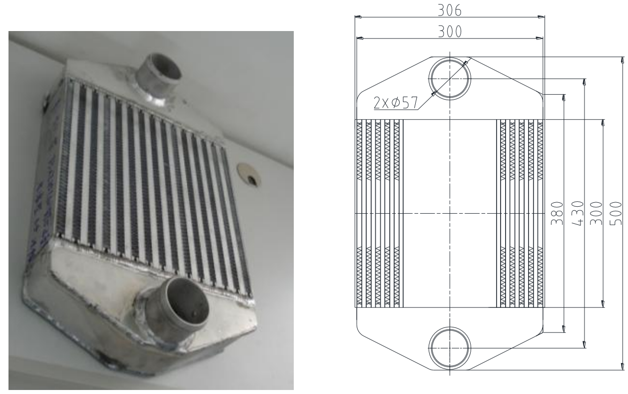
A 3-D model is established (Figure 2). The header tank, bottom tank and entire shape and size are as same as the actual intercooler, but leave out the fin structures and simplify it as porous medium. In order to deduce the porous parameters, empirical correlations given by regressing experimental data to a certain function are utilized in present study.

The calculation equation for the friction factor f of plane-fin is[6]:

$$f = 3.479\,\mathrm{Re}^{-0.389}\left(\frac{L_d}{D}\right)^{-0.396}\left(\frac{F_h}{D}\right)^{0.113}\left(\frac{\delta}{D}\right)^{0.21}$$

The calculation equation for the friction factor f of wavy-fin is[7]:

$$f = 1.16\,\mathrm{Re}^{-0.309}\left(\frac{F_p}{F_h}\right)^{0.3703}\left(\frac{F_p}{b}\right)^{-0.25}\left(\frac{L_d}{L}\right)^{-0.1152}$$



(a) (b)

Figure 1 Photo and model size of the intercooler

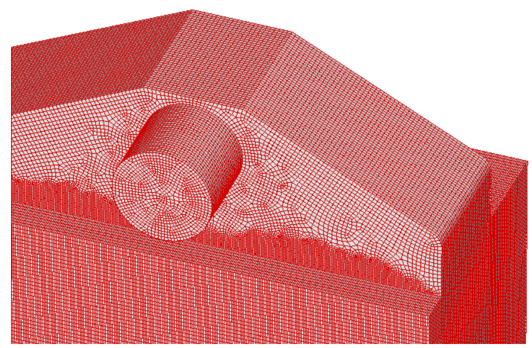


Figure 2 Computational grids of models

According to the definition of friction factor f,

$$f = \frac{\Delta P - \Delta P_i - \Delta P_e}{2\rho V_c^2 \dfrac{L_d}{D}}$$

This can be re-arranged to

$$\Delta P = 2\rho V_c^2 \frac{L_d}{D} f + \Delta P_i + \Delta P_e$$

There V_c is the average velocity of the flow field, D is hydrodynamic diameter of the fin and L_d is the entire flow length. ΔP_i and ΔP_e is the pressure differential produced by the circulation area abruptly narrow and widening. They are not separated from ΔP in the derivation of empirical correlations, so that they do not need to be particularly considered in this case.

Calculate the factor f based on the empirical correlations with different velocity, then, the pressure drop is derived. The porous parameters are estimated by the correlation between velocity and pressure differential accordingly. For the plane-fin, the porosity is about 0.89, the viscous resistance is about 619183α^{-1}/m^{-2} and the inertial resistance is about 2.7558 C_2/m^{-1}. For the wavy-fin, the porosity is about 0.931, the viscous resistance is about $164187\alpha^{-1}/m^{-2}$ and the inertial resistance is about $6.2677\ C_2/m^{-1}$.

The heat exchange power can be defined as:

$$\bar{q} = \frac{q_c + q_h}{2}$$

q_c and q_h are the heat absorbed by the cold air and supplied by the hot gas, can be written as[8]:

$$q_c = \dot{m}_c C_{p,c}(T_{c,out} - T_{c,in})$$

$$q_h = \dot{m}_h C_{p,h}(T_{h,out} - T_{h,in})$$

Where $\dot{m}_c$ and $\dot{m}_h$ are the mass flow rate of cold air and hot gas, $C_{p,c}$ and $C_{p,h}$ are the specific heat respectively.

The pumping power dissipation is written as:

$$q_{\Delta P} = \Delta P \times Q_m$$

The ratio of the heat exchange power ($\bar{q}$) and the pumping power dissipation ($q_{\Delta P}$) is used as an index to measure the working efficiency.

A CFD program Fluent 13 is introduced to study the internal flow and heat transfer. Many assumptions based on the calculation requirements and model characters are proposed: the flow is steady, hot and cold medium are incompressible, physical properties are not changed in simulation. Mesh the model with hexahedral and prism cells overall, and identify the cell size from 0.8mm to 2mm after a grid-independent test (Figure 2). Approximately 5.8 million mixture elements are generated for this model. It takes about 20h for calculation on a super-computer with 6 CPUs involved.

SIMULATION AND EXPERIMENTAL VALIDATION

The model is simulated under 9 cases on the basis of actual working conditions (listed in Table 1).

Table 1 Computational conditions

Case	Hot air flow Kg/hr	Hot air Temperature K	Cold air Velocity m/s	Cold air Temperature K
1	560	435	2	285
2	560	435	4.5	285
3	560	435	8	285
4	762	435	2	285
5	762	435	4.5	285
6	762	435	8	285
7	964	435	2	285
8	964	435	4.5	285
9	964	435	8	285

For knowledge of the inside flow and heat transfer status, a few typical cross-sections extracted from entire model are analyzed as below. Figure 3 displays the velocity distribution in 3 middle cross sections of the model under case1. The sketchy flow status is demonstrated in it: hot air flow in, impinge the tank wall with relatively high speed, then separate to 3 main streams, mostly turn into the core via middle and side passes. Hot air distributes to the hot-passes not only in different mass flow rate, but also in different flow directions. The non-uniformity could impact the heat transfer efficiency as we know, and the air impinging flow would increase the flow resistance. Thus the model should be modified for better performance.

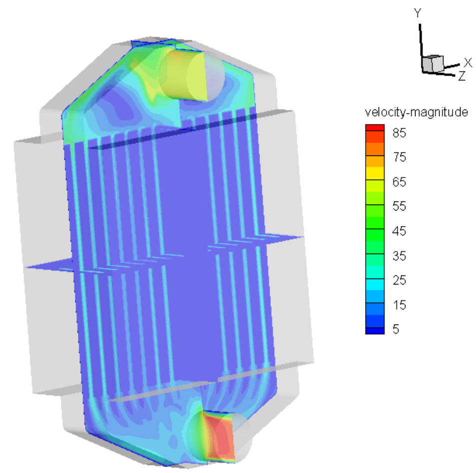
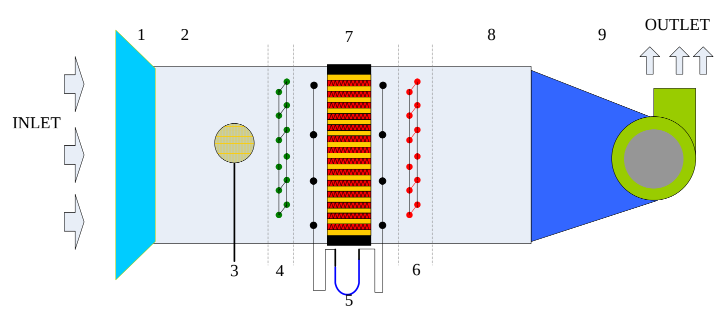


Figure 3 Velocity distributions of the cross sections (Case1)

In order to validate the simulating results, a number of wind tunnel experiments are carried out. The experimental setup includes the cold air system, hot air system and measure system as presented in Figure 4. The wind tunnel experimental system consists of air inlet section, former stable section, air flow rate test section, inlet air temperature test section, intercooler pressure test section, outlet air temperature test section, intercooler, back stable section, fan, outlet section, etc. The test section and stable sections are all made of blockboard to prevent heat loss. Temperature measuring nets are fitted in the upstream and downstream of the intercooler. Hot wire anemometer (Kanomax KA22) is adopted to test the air flow rate. The hot-side inlet and outlet temperature, pressure and flow rate are measured also. This equipments test the intercooler performance mainly by regulating the air flow to process experiments with different status combinations.



1.Air inlet section; 2. Former stable section; 3. Air flow rate test; 4. Inlet air temperature test; 5. Intercooler pressure test 6. Outlet air temperature test 7.Intercooler 8. Back stable section; 9. Fan and outlet

Figure 4 Experimental equipments

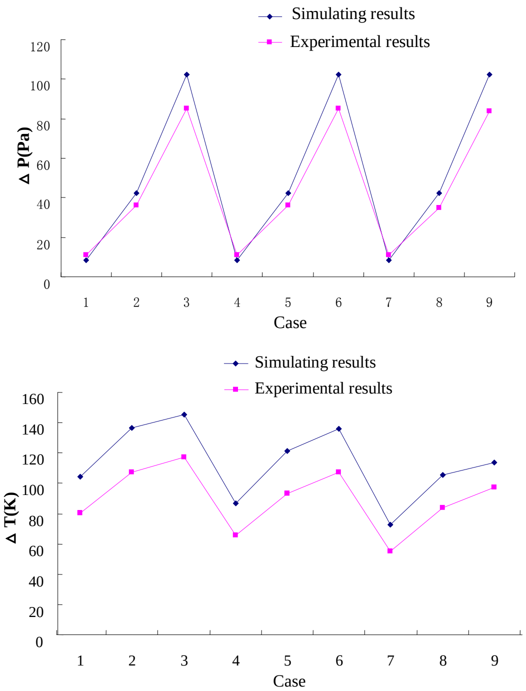


Figure 5 Contrast of cold side pressure drop and hot-side temperature difference between simulation and experiment

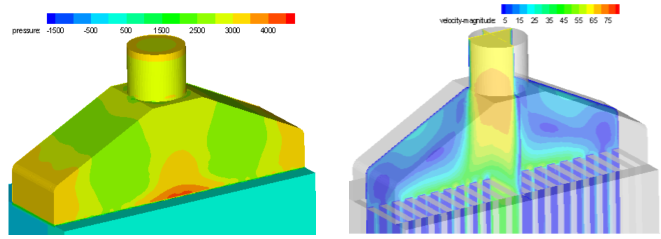
Fig.5 plots the pressure drop and temperature difference vs different cases. The calculated and tested results are compared also. As shown in the profiles, the trends of computational results are similar to the experiment although the data error is significant. That means the computational simulation can effectively study the variation rules of flow and heat transfer characters while boundary conditions changed.

The average relative error between the simulations and the experimental results is 9.9% in cold-side pressure drop, 13.2% in hot-side temperature difference. The probable causes of error are analyzed as below:

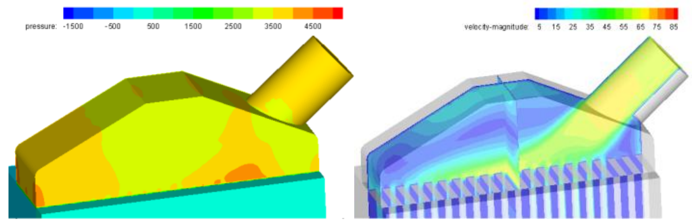
1. The porous parameters are converted by the empirical equations in former simulation. But empirical equations own some error, and the porous model may not be accurate on calculating energy equation, it cause bad results consequently.
2. Numerical uncertainties.
3. Experimental results have a certain error.

MODIFICATION AND COMPARATION

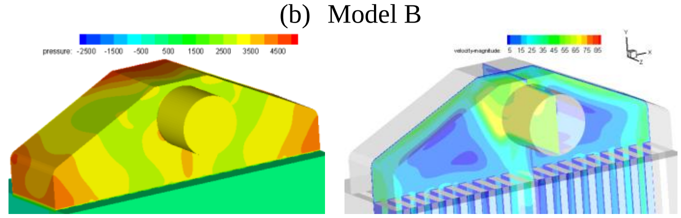
To deep explore the influence of inflow direction and location, 2 modified models are established, named as model A and model B. The inlet pipe is turned to erectly downward in model A and inclined in model B as shown below. Model O indicate the original model based on the real intercooler referred above. Only case1, case5 and case9 conditions are calculated with modified models to reduce the computational workload.



(a) Model A

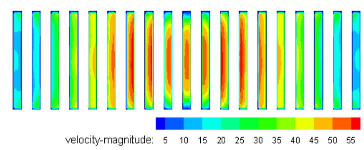


(b) Model B

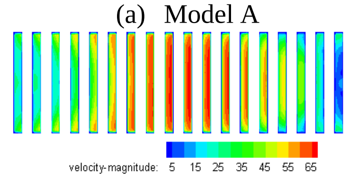


(c) Model O

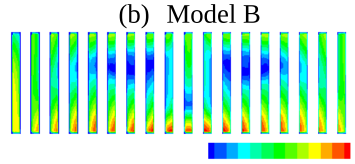
Figure 6 Pressure and velocity distribution of the header tanks (Case1)



(a) Model A



(b) Model B

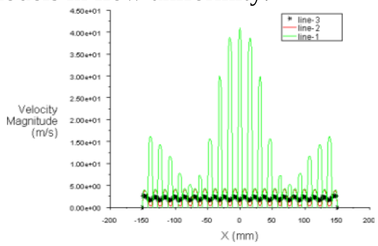


(c) Model O

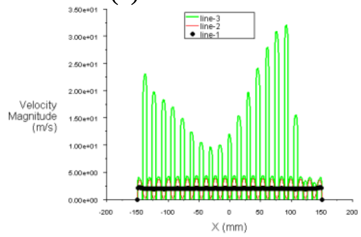
Figure 7 Mass flow distribution of the hot-passes (Case1)

Figure 6 displays the pressure and velocity distribution of the 3 header tanks under same cases. The vertical inlet pipe of model A avoids the direct air impingement to the tank wall, cuts down the wall pressure effectively. High pressure region surrounds in the middle hot-passes, causing a magnificent top speed reached about 40m/s. In model B, inclined inflow direction minimizes the velocity difference of most hot-passes at entrance's front. But the mass flow of two or three back passes is reduced to the lowest level. The inside flow circumstances are elaborated by Figure 7 either， allocations of the mass flow are all non-uniform in the 3 models.

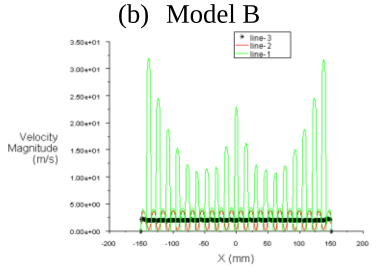
Comparatively speaking, the original model is slightly better than other 2 models in flow uniformity.



(a) Model A



(b) Model B



(c) Model O

Figure 8 Velocity curves along X (Case1)

The velocity curves along X in 3 models are plot in Figure8. Where line-1 is the straight line in the middle of the core, line-2 is the straight line on the downstream side of core and line-3 is the line on the upstream side of core. Three lines are paralleled in the horizontal level. The curves also indicate the mass flow distribution are more non-uniform in model A and model B. It could bring to even worse heat transfer capability of the 2 modified models. For the sake of contrast, simulating data under the same cases of 3 models are listed in Table 2.

Table 2 Hot-side temperature difference and pressure drop simulated by 3 models

Model	Case1		Case 5		Case 9	
	△T(K)	△P(Pa)	△T(K)	△P(Pa)	△T(K)	△P(Pa)
O	104.57	9368.12	121.23	36234.42	133.48	79865.84
A	96.3	7969.28	106.65	31131.32	114.14	68245.77
B	99.97	8124.54	108.53	31764.78	116.99	69421.45

The △T and △P indicate the temperature difference and pressure drop between hot-side inlet and outlet respectively. It is found that the △T and △P both decrease apparently in model A and model B. That means the pump energy-consumption aroused by flow resistance is lessened with the adjustment of inflow direction, but the power of heat exchange is cut down in the meantime. Compared the model B to model A, the △T improves up to 3%, yet the △P improves less than 2%. It suggests that the inclined inlet pipe might be more scientific than erected inlet pipe, the working performance of model B is better than the one of model A.

TANK MODEL OPTIMIZATION

The intercooler discussed in this paper is a typical cross-flow heat exchanger. Every channel can be thought as an individual cross-flow heat exchanger, composed of i*j units as shown in figure 9.

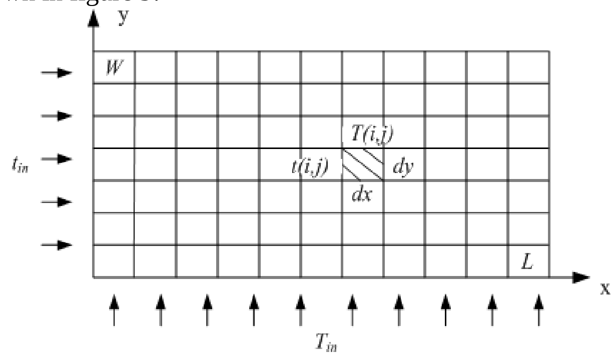


Figure 9 Sketch of the cross-flow heat exchanger

According to the field synergy principle[9], the more similar of the temperature difference between cold medium and hot medium in each unit, the more efficiently in overall heat exchange. In the test and simulations mentioned above, the temperature distribution of the coming air is uniformity in rough, thus the optimizing objective is to improve the temperature distribution of hot gas as much as possible. Control the temperature by regulating the flow pattern is an effective method.

Through the previous studying and analysis, it is clear that the air allocation in hot-passes has a great influence on the intercooler efficiency, and the structure of header tank and inlet pipe is the major factor affecting the flow distribution. Considering the practicability and feasibility, an improved model is fabricated as below: fit an erect inlet pipe, insert 6 flow deflectors in the header tank, other shape and size are remained the same with original model.

The deflectors have fairshaped design, with the space following certain rules. 3-D model (Figure 10) is constructed and the numerical simulations are studied to analyze the improved model.

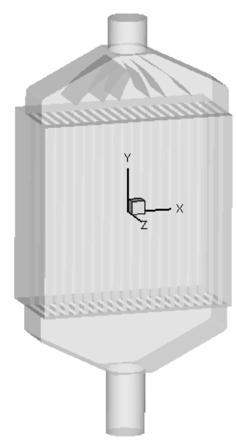


Figure 10 Body sketch

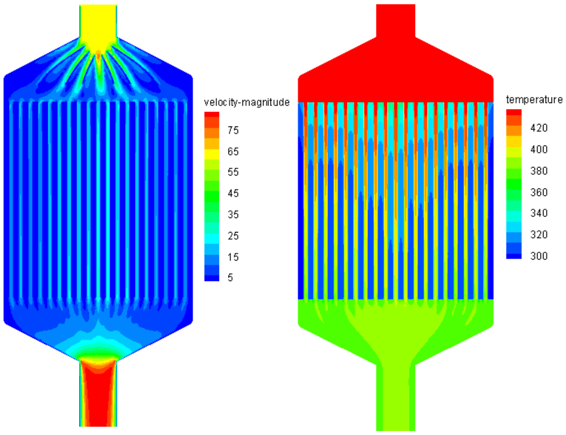


Figure 11 Velocity and temperature distribution(X-Y middle section, Case1)

Figure 11 illuminates the flow and thermal characters in the X-Y middle section of improved model under case1. The figure shows that the hot flow is exactly guided by using the deflectors, which not only avoids the strong impingement, but also minimizes the vortex and turbulence. The deflectors are designed with appropriate radian conforming to the fluid dynamics theory. Deflectors and core keep at a certain distance to make sure there is sufficient space for flow diffusion. The specific number and size could be adjusted according to the tank structure under different circumstances. To accomplish better performance, this design improves the uniformity of core temperature by optimizing the tank flow as same as the using of punched baffle presented in paper [7], but simpler and more convenient.

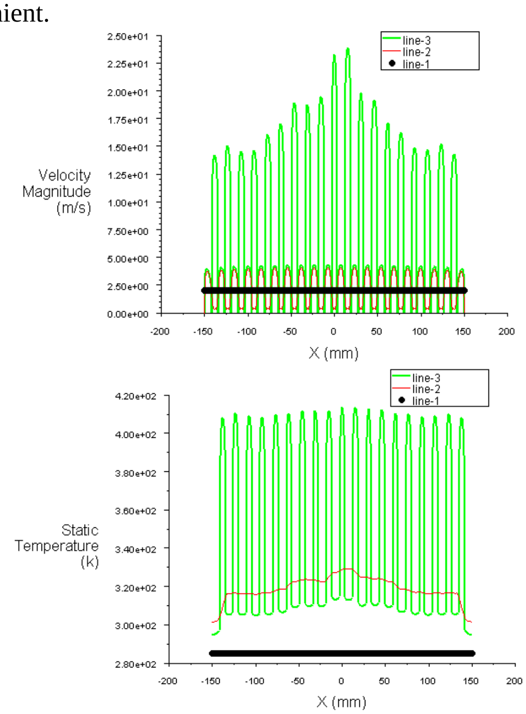


Figure 12 Velocity and temperature curve along X (Case1)

Figure 12 plots the velocity and temperature curves extracted from improved model result, along X, under case1. Where line-1 is the straight line in the middle of the core, line-2 is the straight line on the downstream side of core and line-3 is the line on the upstream side of core as before. It validates the great effect of deflectors also. Mass flow differences among the channels are apparently reduced. Temperature distributions are very similar in all the 19 channels.

Contrast the computational data between the improved model and original model. It is found that the improved model significantly reduces the internal resistance while also maintain the original heat exchange capability. Data from the point of view, the intensity of convection heat transfer increases slightly, but the resistance indicated by pressure drop is down more than 10%.

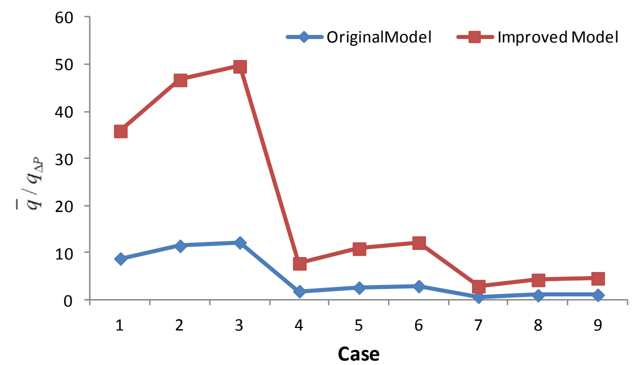


Figure 13 The working efficiency of both models

As displayed in Figure 13, the improved model is great efficient than original model, especially in less hot-flow conditions. This emphasizes again that uniform the temperature difference by adjusting flow pattern is a workable and effective method. The idea of optimization could be generalized to many other cross-flow heat exchangers.

CONCLUSION

The present study deals with a CFD analysis for the flow structures and temperature distributions of a real intercooler firstly. The numerical results are validated by wind tunnel experiment.

Based on the detail analysis of the internal and outside flow fields, it is noticed that the air allocation in hot-passes has a great influence on the intercooler efficiency, and the structure of header tank and inlet pipe is the major factor affecting the flow distribution. Two different tank models are fabricated and compared with the original one. Considering the practicability and feasibility, an improved model is designed to optimize the flow uniformity. It is found that the improved model significantly reduces the internal resistance while also maintain the proper heat exchange capability.

In conclusion, we suggest that the improved structures are more powerful than the traditional one. The method of uniform the temperature difference by adjusting flow pattern is workable and effective, and this optimization idea could be generalized to many other heat exchangers.

REFERENCES

[1] Lalot S., Florent P., Lang S.K., Berglles A.E.. Flow maldistribution in heat exchangers. *Applied Thermal Enginnering*, Vol. 19, 1999, pp. 847–863.

[2] Zhang Z., Li Y.Z., CFD simulation on inlet configuration of plate–fin heat exchanger. *Cryogenics*, Vol. 43, 2003, pp. 673–678.

[3] Zhang Z., Li Y. Z., Xu Q., Experimental research on the effects of distributor configuration on flow distribution in plate-fin heat exchangers. *Heat Transfer-Asian Research*, Vol. 33, 2004, pp. 402-410

[4] Wen J., Li Y. Z., Wang S.M., Zhou A. M., Experimental investigation of header configuration improvement in plate–fin heat exchanger. *Applied Thermal Engineering*, Vol. 27, 2007, pp. 1761–1770

[5] Guo Z. Y., Huang S. Y., Field synergy principle and new technique of heat transfer enhancement. Beijing: China Electric Power Press, 2004.

[6] Dong J. Q.. Reserch on Air Side Flow and Heat Trasfer Character of Vehiclecooling Systems. Shanghai Jiaotong University, 2007, PHD thesis

[7] Dong J. Q., Chen J. P., Chen Z. J. et al. Heat transfer and pressure drop correlations for the wavy fin and flat tube heat exchangers. *Applied Thermal Engineering*, Vol. 27, 2007, pp. 2066-2073

[8] Eiamsa-ard S., Promvonge P. Heat transfer characteristics in a tube fitted with helical screw-tape with/without core-rod inserts. *International Communications in Heat and Mass Transfer*, Vol. 34(2), 2007, pp.176-185.

[9] Guo Z. Y.，Zhou S. Q., Li Z. X. et al. Theoretical analysis and experimental confirmation of the uniformity principle of temperature difference field in heat exchanger. *International Journal of Heat and Mass Transfer*, Vol. 45, 2002, pp: 2119-2127