Simulation and CFD Analysis of heat pipe heat exchanger using Fluent to increase of the thermal efficiency

M. H. SABER, H. MAZAHER ASHTIANI
CFD department
Fanavar Petro Arya engineering Co.
Tehran, Saharak Gharb
IRAN
MSaber@PetroArya.com http://www.petroarya.com

Abstract: - In this paper, the heat pipe heat exchanger (HPHE) is considered and computational fluid dynamics (CFD) is used to analyze its evaporator's performance and based on it, will be try to increase the thermal efficiency and to optimize the distribution of fluid flow in this type of heat exchangers. The CFD principles which are made use in this article are effective and appropriate methods that using finite volumes to solve the processes that consist of transport phenomena. The necessary numerical computations are accomplished by Fluent (the CFD solver program) and the results are given in graphical representation. This analysis is done for an existing HPHE with the specified conditions of inlet flow to evaporator and finally the comparison and conclusion are presented.

Key-words: - heat pipe heat exchanger, HPHE, simulation, CFD, Fluent.

1 Introduction

Heat pipes are two-phase heat transfer devices with high effective thermal conductivity. Due to the high heat transport capacity, heat exchanger with heat pipes has become much smaller than traditional heat exchangers in handling high heat fluxes. With the working fluid in a heat pipe, heat can be absorbed on the evaporator region and transported to the condenser region where the vapour condenses releasing the heat to the cooling media [1].

A heat pipe is an evaporation-condensation device for transferring heat in which the latent heat of vaporization is exploited to transport heat over long distances with a corresponding small temperature difference. The heat transport is realized by means of evaporating a liquid in the heat inlet region (called the evaporator) and subsequently condensing the vapour in a heat rejection region (called the condenser). Closed circulation of the working fluid is maintained by capillary action and /or bulk forces. The heat pipe was originally invented by Gaugler of the General Motors Corporation in 1942, but it was not, however, until its independent invention by Grover [3, 4] in the early 1960s that the remarkable properties of the heat pipe became appreciated and serious development work took place. An advantage of a heat pipe over other conventional methods to transfer heat such an a finned heat sink, is that a heat pipe can have an extremely high thermal conductance in steady state operation. Hence, a heat pipe can transfer a high amount of heat over a relatively long length with a comparatively small temperature differential [1,2].

The increasing demand for energy efficiency in domestic appliances (such as a dishwasher, air conditioner, durable drier or fridge/freezer) and industrial systems and devices is the main drive for continuously introducing and/or improving heat recovery systems in these appliances, systems and devices. Heat transfer efficiency in such systems is the primary factor for efficient performance of the whole systems [5].

However, the design of the heat recovery systems with heat pipe units is the key to providing a heat exchanger system to work as efficient as expected. Without correct design of such systems, heat pipes are not able to transport enough heat and may function as an extremely poor thermal conductor in the systems [6].

Computational fluid dynamics is a powerful tool for fluid dynamics and thermal design in industrial applications, as well as in academic research activities. Based on the current capabilities of the main CFD packages suitable for industry (such as FLOTHERM, ICEPAK, FLUENT and CFX) and the nature of industrial applications, understanding the physics of the processes, introducing adequate simplifications and establishing an appropriate model are essential factors for obtaining reasonable results and correct thermal design [6].

Having any knowledge about flow type, temperature profile and turbulence zones in heat pipe heat exchanger are very useful aids for reliable, high efficiency and economical design or optimization. Although the most existent designs use experimental methods, but as a outcome of growth and development of numerical methods and the softwares which can solve the PDE, the tendency for analyses of fluid flow and heat transfer are appeared. So because of expense and time wasting of empirical assessments and hardness of turbulence conditions verification and details in heat exchangers, a reliable modeling is desired for HPHE designing. Computational fluid dynamics (CFD) is one of the robust methods for fluid flow simulation that analyzes the systems consists of momentum transfer, heat transfer and chemical reactions (mass transfer). In fact, CFD is more than a simulator because it reinforces designers having good sights about the operations and processes in the system. In this paper, goal is the analysis of fluid conditions on the tubes bundle in the HPHE using CFD packages and we'll make decision how flow enters to optimize distribution based on its results.

2 Structure and operation of heat Pipes

The main regions of the standard heat pipe are shown in Fig. 1. In the longitudinal direction (see Fig. 1a), the heat pipe is made up of an evaporator section and a condenser section. Should external geometrical requirements make this necessary, a further, adiabatic, section can be included to separate the evaporator and condenser.

The cross-section of the heat pipe, Fig. 1b, consists of the container wall, the wick structure and the vapour space.

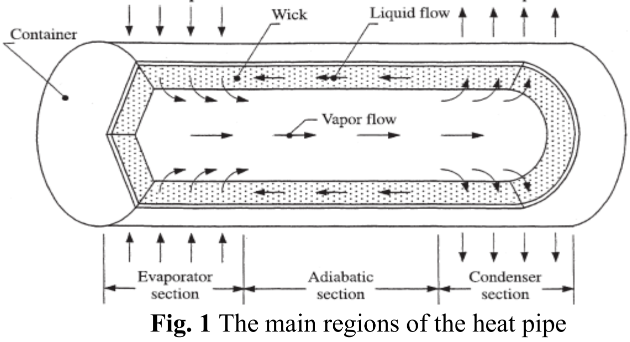

Fig. 1 The main regions of the heat pipe

The heat pipe is characterized by the following:

(i) Very high effective thermal conductance.

(ii) The ability to act as a thermal flux transformer.

(iii) An isothermal surface of low thermal impedance. The condenser surface of a heat pipe will tend to operate at uniform temperature. If a local heat load is applied, more vapour will condense at this point, tending to maintain the temperature at the original level.

The overall thermal resistance of a heat pipe, defined by equation (1), should be low, providing that it functions correctly.

The operating limits for a wicked heat pipe, are illustrated in Fig. 2.

$$R = \frac{T_{hot} - T_{cold}}{Q} \quad (1)$$

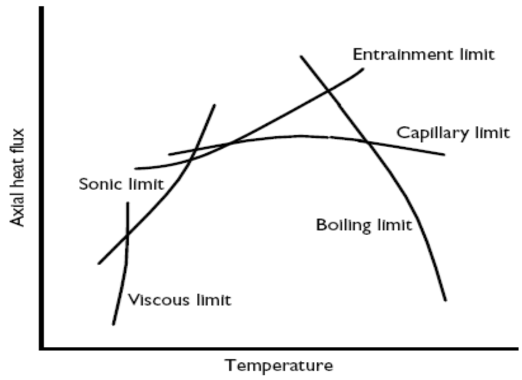

Fig. 2 Limitations to heat transport in a heat pipe.

Each of these limits may be considered in isolation. In order for the heat pipe to operate the maximum capillary pumping pressure, ΔPc, max must be greater than the total pressure drop in the pipe. This pressure drop is made up of three components.

(i) The pressure drop ΔPl required to return the liquid from the condenser to the evaporator.

(ii) The pressure drop ΔPv necessary to cause the vapour to flow from the evaporator to the condenser.

(iii) The pressure due to the gravitational head, ΔPg which may be zero, positive or negative, depending on the inclination of the heat pipe.

For correct operation; $P_{c,max} \geq \Delta P_l + \Delta P_v + \Delta P_g^l$. If this condition is not met, the wick will dry out in the evaporator region and the heat pipe will not operate [7].

3 CFD modeling

CFD methods consist of numerical solutions of mass, Momentum and energy conservation with other equations like species transport. Two main stages comprise the solution of CFD problems. First, whole of fluid field divides to small elements that their names are mesh, and then partial differential equations that explain transport phenomena in fluid flow are used for these elements. Consequently, many non-linear equations appear which have to solve simultaneously. The solution of these equations accomplish with numerical algorithm and methods.

Conservation equations for compressible flow are [8]:

Continuity equation:

$$\frac{\partial \rho}{\partial t} + div(\rho U) = 0 \qquad (2)$$

Reynolds equations:

$$\frac{\partial(\rho u)}{\partial t} + div(\rho u U) = -\frac{\partial P}{\partial x} + div(\mu . grad. u) + \left[-\frac{\partial(\rho \overline{u'^2})}{\partial x} - \frac{\partial(\rho \overline{u'v'})}{\partial y} - \frac{\partial(\rho \overline{u'w'})}{\partial z}\right] + S_{mx}$$

$$\frac{\partial(\rho v)}{\partial t} + div(\rho v U) = -\frac{\partial P}{\partial y} + div(\mu . grad. v) + \left[-\frac{\partial(\rho \overline{u'v'})}{\partial x} - \frac{\partial(\rho \overline{v'^2})}{\partial y} - \frac{\partial(\rho \overline{v'w'})}{\partial z}\right] + S_{my}$$

$$\frac{\partial(\rho u)}{\partial t} + div(\rho u U) = -\frac{\partial P}{\partial z} + div(\mu . grad. w) + \left[-\frac{\partial(\rho \overline{u'w'})}{\partial x} - \frac{\partial(\rho \overline{v'w'})}{\partial y} - \frac{\partial(\rho \overline{u'w'})}{\partial z}\right] + S_{mz} \qquad (3)$$

It is needed a model of the turbulence to solve above equations. There are many equations to model turbulence. k-ε model is one of the most reliable models that exist. This model applies blew turbulence parameters:

$$\mu_{eff} = \mu + \mu_T \, , \mu_T = \rho C_\mu \frac{k^2}{\epsilon}$$

$$\frac{\partial(\rho k)}{\partial t} + div(\rho k U) = div\left(\frac{\mu_{eff}}{\sigma_L} grad. k\right) + G - \rho\epsilon$$

$$\frac{\partial(\rho \epsilon)}{\partial t} + div(\rho \epsilon U) = div\left(\frac{\mu_{eff}}{\sigma_L} grad. \epsilon\right) + C_{1\varepsilon}\frac{\epsilon}{k} 2\mu_{eff}. E_{ij}. E_{ij} - C_{2\varepsilon}\rho\frac{\epsilon^2}{k} \qquad (4)$$

Where: C_μ , $C_{1\varepsilon}$, $C_{2\varepsilon}$, σ_L are coefficients of turbulent kinetic energy. μ , μ_t , μ_{eff} are laminar, turbulence and effective viscosity. E_{ij} is element deformation rate, ε is turbulent dissipation rate, and k is turbulent kinetic energy.

4 Heat pipe heat exchanger CFD modeling

It was made a specific geometry for the HPHE in dimensions of 150*180*150 (cm*cm*cm) that is based on the basic design and with attention to needs and process constraints in the operation unit of gas field of South Pars in IRAN. After making geometry of HPHE in Gambit (the CAD program), meshing of HPHE's evaporator drew a volume which has relatively 1500000 cells. Then, this model entered in Fluent (CFD solver program) to solve. In fluent user interface 3D double precision (3dd), segregated solution method and k-ε turbulence model are used to simulate this CFD problem.

The schematic of HPHE geometry for this case study is shown Fig. 2.

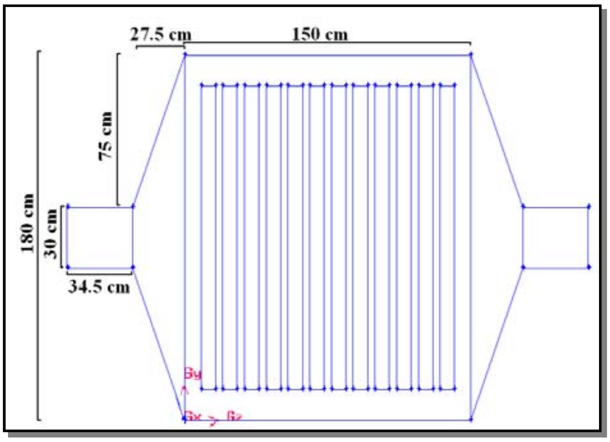

Fig. 2 case1: Model of HPHE (basic design)

There are 72 tubes in this heat exchanger that have been placed in 6 rows with 12 tubes in any row. Inlet flow conditions (physical specifications) which come from basic design condition are given in table 1 and they are same for all of cases.

Table 1 physical specification of combustion gases for CFD modeling

Physical parameters	Value
Temperature (K)	793
Mass rate (kg/sec)	3.75
Viscosity (kg/m.s)	$2*10^{-5}$
Density (kg/m^3)	0.88
Heat conduction (W/m.K)	95

So that is shown in Fig. 2, entry's dimensions of HPHE is 30*30 (cm*cm) and flow enters in center of tubes bundle. For simulation of evaporator where hot fluid (combustion gases) accompanies pressure drop, heat flux is supposed constant value. In Fig. 2 to 5 are shown geometry of four different cases that this paper purposes to compare with them. Case (1) is introduced as basic condition and its geometry specifications presented in Fig. 1. In the second case, dimensions of cross area of entry have been double size. In the case (3) with same dimensions to basic design, is used one horizontal plate after entry. Finally, in 4th case one imperfect cone is used in entry of HPHE to analyze the distributed flow.

In the assessment of this article, analysis of outlet temperature of combustion gases as well as flow distribution on the tubes bundle is presented.

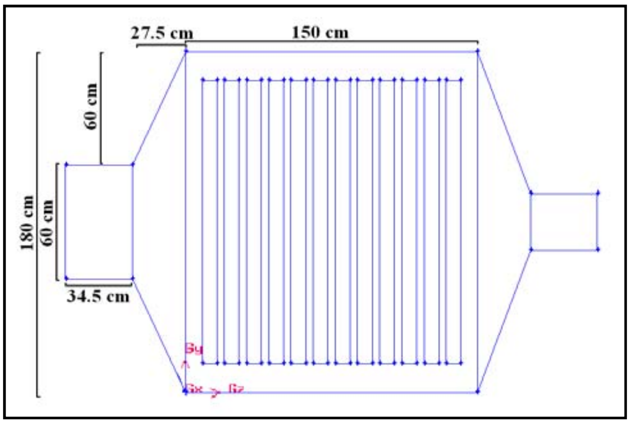

Fig. 3 Case 2: increase of entry cross area

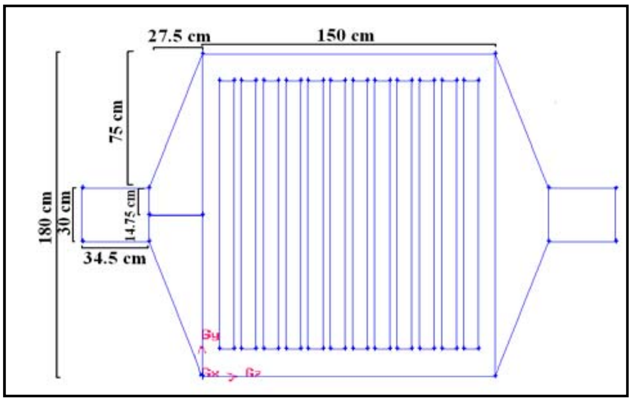

Fig. 4 Case 3: horizontal plate after entry

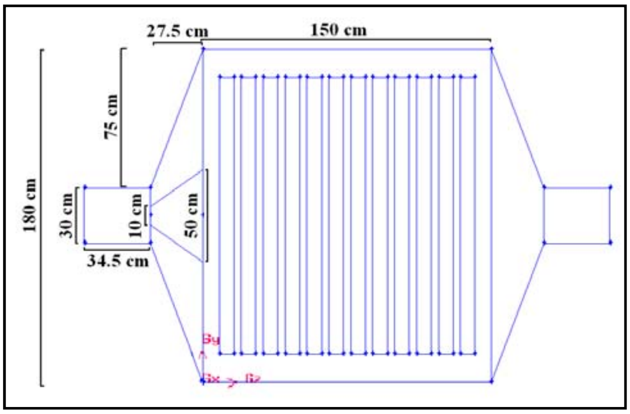

Fig. 5 Case 4: imperfect cone after entry

4.1 Distribution of flow

When simulated model solved in Fluent and the residual converged, the solutions are extracted and the results are illustrated by figures. Axial velocity distribution in central plate of HPHE for basic design is shown in Fig.6.

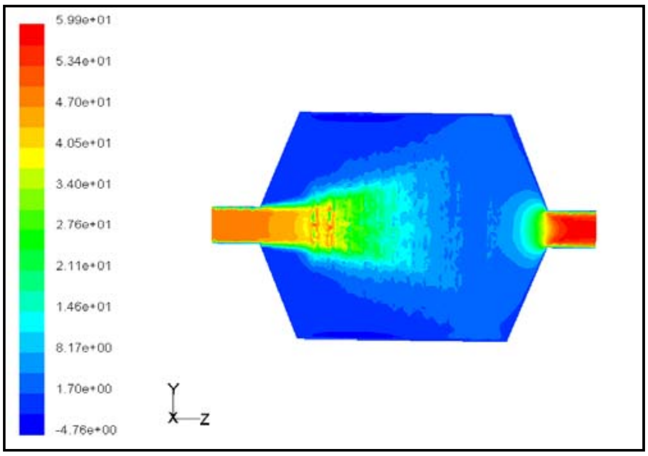

Fig. 6 Contours of axial velocity for case (1)

When heat flux is constant (-156568 W), temperature value in HPHE's outlet will be 750K. This specifies that combustion gases have temperature decrease about 43K. Here for presentation and comparison, geometries and results are given together and they are analyzed. In Fig. 6 to 9, axial velocity contours thorough of HPHE and in central plate are illustrated. In Fig. 6 so that is distinguished, from the begging to middle of HPHE, the flow hasn't been distributed over the whole tubes bundle completely. Therefore, in the first look this matter determines that it hasn't been used whole heat exchanger's volume effectively. In fact, some parts of tubes bundle's surfaces aren't exposed to hot gas flow, so the HPHE efficiency lowers. In Fig. 7 whose dimensions of entry cross area is double of basic design, in comparison with case (1), the flow profile approaches to fully distribution in shorter distance from entry. Set of this materials, it could be say that the entry which has larger cross area prepares better distribution on the tubes bundle. Of course in industrial units there are many constraints for cross area because growth of it increases pressure drop for gas flow. Using the distributers and baffles may leads to conduct the flow on the tubes bundle. This paper considers two cases that the baffles are used in those. One case uses a horizontal plate after entry of HPHE which can divide flow to two parts (case (3)), and in another case using one imperfect cone conducts the flow in 3 paths comprises up, down and middle of HPHE. Case (3) is better than case (1) approaches to fully distribution. This means that case (3) reaches to suitable distribution in shorter distance from the beginning of HPHE in comparison with case (1), but according to the graphical depiction, case (2) has still better distribution.

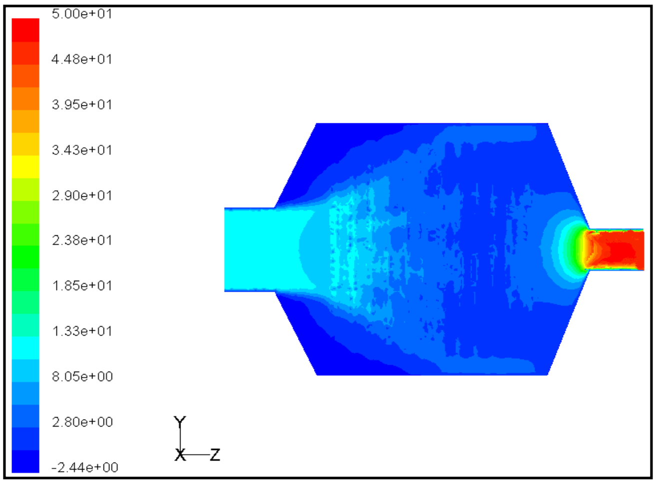

Fig. 7 Contours of axial velocity for case (2)

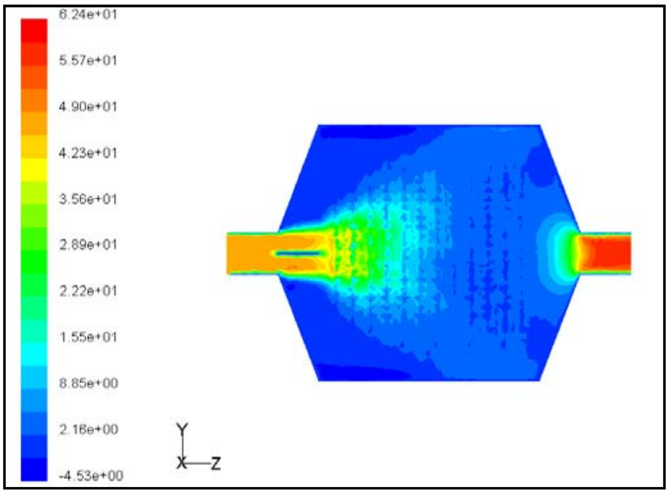

Fig. 8 Contours of axial velocity for case (3)

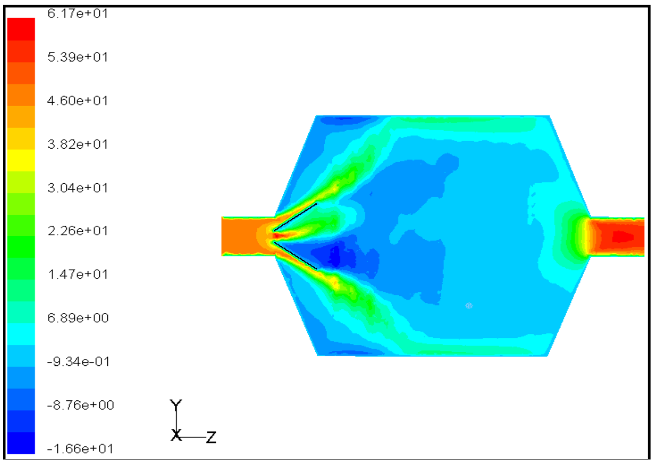

Fig. 9 Contours of axial velocity for case (4)

In the 4th case which has one imperfect cone baffle the flow is divided to three parts. One path of flow is conducted to the center of tubes bundle and other paths go to up and down sections of HPHE symmetrically.

4.2 Temperature's distribution

The contours of temperature for all of cases are shown in Fig. 10 to 13. Contours of temperature can explain performance of HPHE clearly. Certainly temperature's profiles are derived flow's profiles. Using contours of temperature that are exhibited graphically, the analysis of temperature differences in HPHE length is possible. In Fig. 10 to 13 that are illustrated temperature's profiles, the areas that aren't affected by heat of combustion gases are verified. According to Fig. 10 depend on case (1) it is considerable that the large tubes area doesn't expose combustion gases and so heat transfer is low and heat recovery is not appropriate. About case (2), so that the Fig. 11 shows, temperature's distribution is better and more developed so temperature's profile approximates plug flow. In Fig. 12 and 13 which are concern two cases that are used baffles, conditions of temperature's distribution are suitable and acceptable, specially case (4) which has a developed distribution and symmetrical profile of temperature.

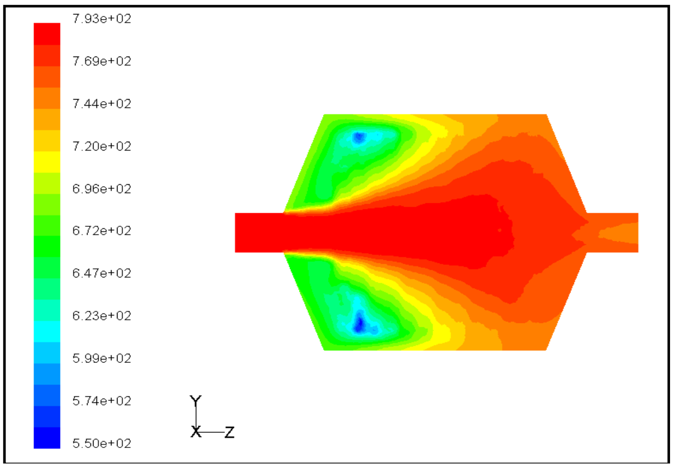

Fig. 10 Contours of temperature for case (1)

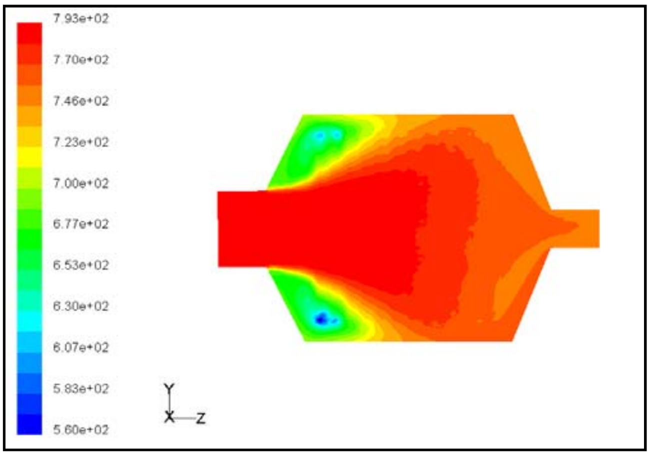

Fig. 11 Contours of temperature for case (2)

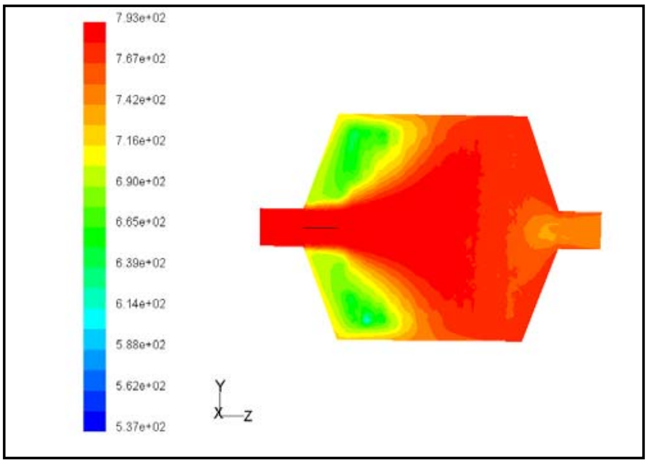

Fig. 12 Contours of temperature for case (3)

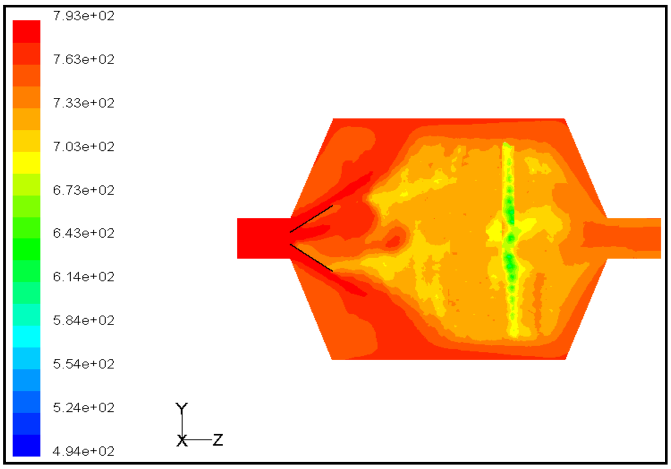

Fig. 13 Contours of temperature for case (4)

Finally, for a convenient comparison, table (2) is given that reports average of outlet flow temperature and temperature difference (ΔT). Inlet flow temperature is 793K for all of them.

Table 2 Outlet temperature and temperature difference for all of cases

	T_{out}	ΔT
Case 1	750	43
Case 2	747	46
Case 3	747.5	45.5
Case 4	738	55

So that is realizable from table (2), case (4) with imperfect cone baffle has highest rate of temperature change that causes the highest heat transfer and heat recovery between these cases. Case (2) with double size entry's cross area is after case (4) in heat recovery capability. Therefore, it could be state that type of the flow's distribution is effective parameter in HPHE's rate of heat transfer and consequently in heat recovery quantity of it.

5 Conclusions

Finally here some considerations for performance optimization in HPHE are presented:

1- Short baffles can be used in up and down sections of HPHE to avoid bypass flows.

2- Entry cross area affects distribution of flow and temperature. According to the results, although increase of entry cross area helps to make better distribution, but

ISSN: 1790-5095

ISBN: 978-960-474-158-8

large scale of it may increase pressure drop and operation costs.

3- Use of baffles is a very effective role for appropriate development of flow and temperature's profiles. The results show that using an imperfect cone with 1/5 diameter ratio, can optimizes performance of HPHE very well.

4- A combined method thorough of these methods can make highest efficiency. Use of the larger entry so process constraints allow, as well as one imperfect cone is a suitable way. By this method, the best distribution of flow and temperature with minimum pressure drop will be reached and so operation cost may decreases.

The above results and conclusions are usable in the design and optimization fields directly and cause increase of the thermal efficiency of heat pipe heat exchanger (HPHE) in both of the evaporator and condenser.

References:

[1] Firouzfar E., Attaran M., "A review of heat pipe heat exchanger activity in Asia", Proceedings of world academy of science, engineering and technology, volume 30 July 2008 ISSN 1307-6884.

[2] Gaugler, R.S. US Patent 2350348. Appl. 21 Dec, 1942. Published 6 June 1944.

[3] Grover, G.M. US Patent 3229759. Filed 1963.

[4] Grover, G.M. Cotter, T.P. and Erickson, G.F. "Structures of very high thermal conductance", J. App. Phys., Vol. 35, P. 1990, 1964.

[5] Sauciue I., Akbarzade A., Johnson P., Chracteristics of two-phase closed thermosuphons for medium temperature heat recovery applications, heat Recovery Systems and CHP, P. 631-640, 15(7) 1995.

[6] Lin S., Broadbent J., McGlen R., "Numerical study of heat pipe application in heat recovery sustems", Applied Thermal Engineering V. 25(2005) 127-133.

[7] Reay D.A., Kew P.A., "Heat pipes Theory, Design and Application", Fifth Edition, Elsevier 2006.

[8] Versteeg, H. K., Malalasekera, W.,"An introduction to computational fluid dynamics – The finite volume method", Longman Limited, England, 1995.