MATEC Web of Conferences **97**, 01077 (2017)
ETIC 2016
DOI: 10.1051/matecconf/20179701077

Numerical Simulation of Different Models of Heat Pipe Heat Exchanger Using AcuSolve

Nurul Amira Zainal [1,*], *Saiful Naim* Sulaiman[1], *Mohd Afdhal* Shamsudin[1], and *Iskandar* Waini[1]

[1] Faculty of Engineering Technology, Universiti Teknikal Malaysia Melaka, Hang Tuah Jaya, 76100 Durian Tunggal, Melaka, Malaysia

Abstract. In this paper, a numerical simulation of heat pipe heat exchanger (HPHE) is computed by using CFD solver program i.e. AcuSolve. Two idealized model of HPHE are created with different variant of entry's dimension set to be case 1 and case 2. The geometry of HPHE is designed in SolidWorks and imported to AcuSolve to simulate the fluid flow numerically. The design of HPHE is the key to provide a heat exchanger system to work proficient as expected. Finally, the result is used to optimize and improving heat recovery systems of the increasing demand for energy efficiency in industry.

1 Introduction

The heat exchanger is a device used to transfer heat between two or more fluid streams at different temperature. Heat exchangers find widespread application in space heating, refrigeration, air-conditioning, power stations, petroleum refinery, and automotive application. For efficiency purposes, heat exchangers are designed to maximize the surface area of the wall between the two fluids while minimizing resistance to fluid flow through the exchanger. There are several common heat exchanger designs. These include heat pipe heat exchanger (HPHE). The heat pipe heat exchanger is playing a considerable role in different fields in the industry. HPHE is a heat-transfer device that combines the principles of both thermal conductivity and phase transition with high effective thermal conductivity. Concisely, heat pipe is an evaporation-condensation device for transferring heat in which latent heat of vaporization is utilized to transfer heat over a long distance with a corresponding small temperature difference.

The aim of augmenting heat transfer is to accommodate high heat fluxes. An advantage of a heat pipe over other conventional methods to transfer heat is that HPHE can have an extremely high thermal conductance in steady state operation. Firouzfar [1] stated that heat pipe with liquid metal working fluids can have a thermal conductance of a thousand or even tens of thousands of times greater than the best solid metallic conductors, silver or copper. However, there are limitations on the ability of HPHE to transfer heat; commonly known as the sonic limit, the capillary limit, the viscous limit, the entrainment limit and the boiling

---

[*] Corresponding author: nurulamira@utem.edu.my

limit. Recently, HPHE found a wide variety of applications and research studies. Yau [2] reviewed on the application of horizontal HPHE air conditioning systems in tropics. It shows that on the basis of results obtained, the application of horizontal HPHE in terms of energy saving, dehumidification enhancement and condensate drainage is recommended for the tropics. Ma [3] studied heat pipe assisted heat exchanger used for industrial waste heat recovery experimentally. The results indicated that the exergy efficiency (also known as the second-law efficiency or rational efficiency) increased with the increment of waste water mass flow rate at constant fresh water mass flow rate, while the effectiveness decreased at the same operation condition. Meanwhile, investigation of HPHE integrated with a water spray foe the heat recovery from bile exhaust gas was studied by Yuan [4]. The study presents a thermodynamic analysis and a numerical simulation of a heat pipe heat exchanger which recovers both sensible and latent heat from the exhaust gases of a boiler.

Heat pipe technology has found increasing demands in enhancing the thermal performance of heat exchange for energy efficiency, especially in domestic appliances and industrial systems. Heat transfer efficiency in such systems is the primary factor for efficient performance of the whole systems [5]. Therefore, the aim for continuously introducing and improving heat recovery systems is highly needed. Based on the nature of industrial demands, understanding the physics of the processes, introducing adequate simplifications and establishing an appropriate design are essential factors for obtaining reasonable results and correct thermal design [6]. It is clearly that the key to provide a heat exchanger system to efficiently work as expected is the design of the heat recovery systems with heat pipe units. Heat pipe is unable to transport enough heat and may function poorly as thermal conductor in the systems without correct design. Lin [7] suggested that Computational Fluid Dynamics (CFD) modelling is able to predict thermal performance of the dehumidification solution with heat pipe heat exchangers. In the meantime, CFD modelling of flow and heat transfer in a thermosiphon was determined by Alizadehdakhel [8]. The studied concluded that CFD is a useful tool to model and explain the complex flow and heat transfer in a thermosyphon.

CFD is an effective and appropriate method that uses finite volumes to solve the processes that consist of transport phenomena. It is one of the robust methods for fluid flow simulation that analyses the systems consists of momentum transfer, heat transfer and mass transfer. In fact, CFD is actually not only for simulation work, but it is also a good guide for engineers, because it reinforces designers having good sights about the operations and processes in the system [9]. Moreover, CFD is a time-saving approach, cost effective and may reduce the overall needs for man-power compared to real experimental work. Referring to the advantages of the above mentioned, this paper aim to analyse fluid conditions on the tube bundle in the HPHE by using CFD programme packages i.e. AcuSolve. CFD modelling was carried out to show the ability of this approach as well as to investigate the flow pattern in HPHE.

2 Mathematical formulation

CFD uses numerical analyses and algorithms to solve and analyze problem that involves fluid flows. The fundamental basis of almost all CFD problems is the Navier-Stokes equations. Principally, the equations are a set of coupled differential equation and could be solved for a given flow problem mathematically. However, in practice these equations are too difficult to solve analytically. Thus CFD was employed in order to solve approximations to the equations using a variety of techniques including finite element method. The Navier-Stokes equation consists of conservation of mass (Eq. 1), conservation of momentum (Eq. 2) and conservation of energy (Eq. 3).

Conservation of Mass:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{V}) = 0 \tag{1}$$

Conservation of Momentum:

$$\rho \frac{D\vec{V}}{Dt} = -\nabla P + \rho \vec{g} + \mu \nabla^2 \vec{V} \tag{2}$$

Conservation of Energy:

$$\rho \left(\frac{\partial h}{\partial t} + \nabla \cdot (h\vec{V}) \right) = -\frac{Dp}{Dt} + \nabla \cdot (k \nabla T) + \phi \tag{3}$$

To solve the above equations, a turbulence model is needed. This study employed Spalart-Allmaras model. Spalart-Allmaras incorporates some of the recent advances in turbulence modelling. It regularly shows that it has equal or superior accuracy for nearly all classes of flows. In addition, the Spalart- Allmaras model is more computationally efficient compared to other model. Javaherchi [10] reviewed the Spalart- Allmaras model and its modification. The model is presented in Eq. (4).

Spalart- Allmaras model :

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \frac{\partial}{\partial x_i}(\rho \vec{v} u_i) = G_\upsilon + \frac{1}{\sigma_{\vec{\upsilon}}}\left[\frac{\partial}{\partial x_j}\left\{ \left(\mu + \rho \vec{\upsilon} \right) \frac{\partial \vec{\upsilon}}{\partial x_j} \right\} + C_{b2\rho} \left(\frac{\partial \vec{\upsilon}}{\partial x_j} \right)^2 \right] - Y_\upsilon + S_{\vec{\upsilon}} \tag{4}$$

3 Design of HPHE

Two idealized models of HPHE are designed in this study with different variants of entry's dimension as shown in Fig. 1 and Fig 2. The model dimensions were taken from previous studies which based on the basic design and with attention to needs and process constraints on the operation unit gas field [9]. The geometry of HPHE was designed in SolidWorks. SolidWorks is a CAD & CAE program and utilizes a parametric feature-based approach to create models and assemblies. The model is then imported to CFD Solver i.e AcuSolve to simulate the fluid flow numerically.

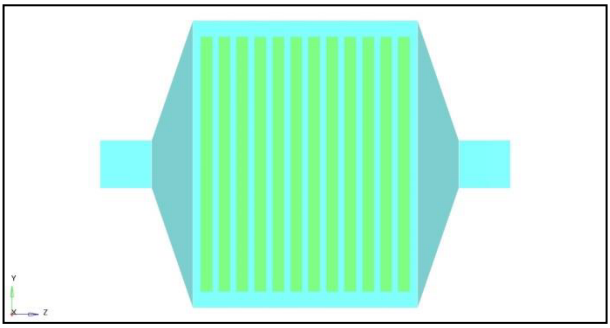

Fig. 1. Schematic model of HPHE Case 1

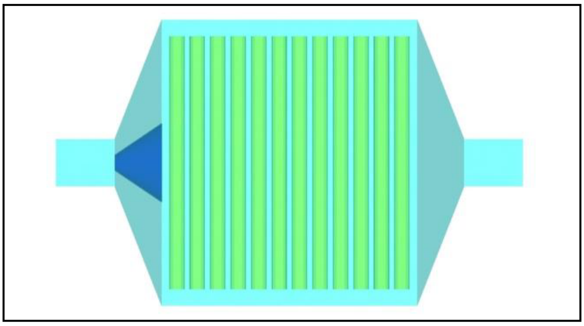

Fig. 2. Schematic model of HPHE Case 2

4 Simulation model of HPHE

There are several components that need to be established before the simulation process begins. The basic components include wall as the outermost component of the body, tube inside HPHE, a fluid that represents the volume in which the gas will flow, and inlet-outlet flow represents the area where the gas will flow in and out. Then, the model is meshed in Hypermesh with total number of elements is 1782371 relatively and exported to AcuSolve for further analysis hereafter. Fig 3 shows the complete meshed model; meanwhile Fig. 4(a) shows the side-view of HPHE. Meanwhile, the top-view of HPHE is shown in Fig 4(b).

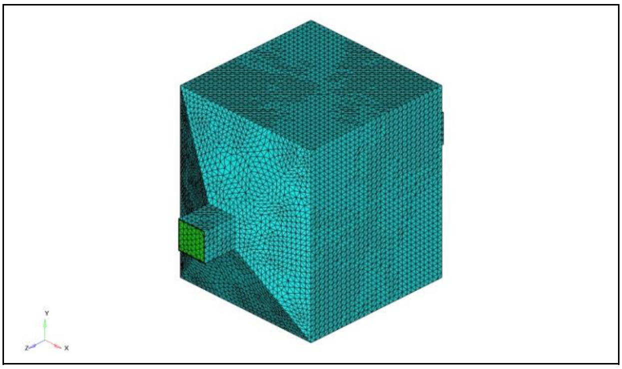

Fig. 3. Complete meshed model

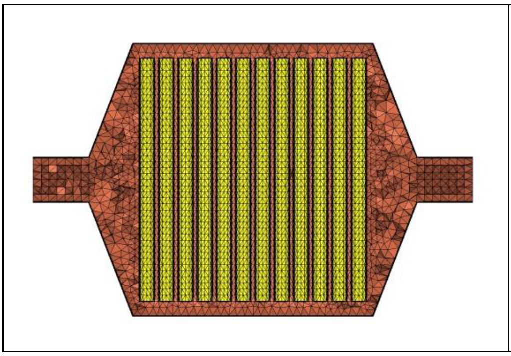

Fig. 4(a). Side view (cut) of HPHE after meshed.

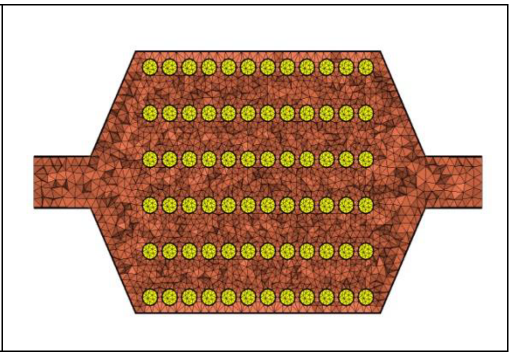

Fig. 4(b). Top view (cut) of HPHE after meshed.

The boundary condition specification for each component is given in Table 1 and the value implies to both cases.

Table 1. Boundary condition specification of HPHE

Components	Modelling details
Fluid	- Medium: Fluid - Material model: Air
Inflow	- BC type: Inflow - Inflow type: Mass flux - Mass flux: 3.75 kg/s - Temperature: 793 K
Outflow	- BC type: Outflow
Tube	- BC type: Wall - Temperature BC type: Flux - Heat flux : -2714.56 W/m2 (-156568 / 72 tubes)
Wall	- BC type: Wall

In case 2, the simulation model was derived from case 1 model, means that the elements are all the same. However a component is added to the model, which is a cone located right after the inlet. The property of the meshed cone is given below in Table 2.

Table 2. The property of the meshed cone in case 2

Components	Modelling details
Cone	- Modelled as a wall - Element size : 50mm - No of elements: 3616 - Element type: TRIA3 (tria elements with 3 nodes)

Henceforth, the total number of elements in case 2 is 1 868 865. On the other hand, the material modeling and CFD analysis for case 1 and case 2 was set as in Table 3 and Table 4 below.

Table 3. Material modeling of HPHE

Parameters	Value
Material Type	Air
Density	0.88 kg/m3
Viscosity	2 e-5 kg/ms
Conductivity	95 W/mK

Table 4. CFD analysis of HPHE

Parameters	Type
Analysis Type	Steady state
Temperature equation	Advective Diffusive
Turbulence model	Spalart Allmaras
Convergence tolerance	0.001
Temperature at initial condition	793 K

5 Results and analysis

As mentioned earlier, the main objective of the present study is to explore the ability of this CFD modelling approach i.e AcuSolve, as well as to investigate the flow pattern in HPHE. Fig.5 shows the velocity distribution of HPHE in case 1 meanwhile, Fig. 6 illustrates the velocity distribution of HPHE in case 2.

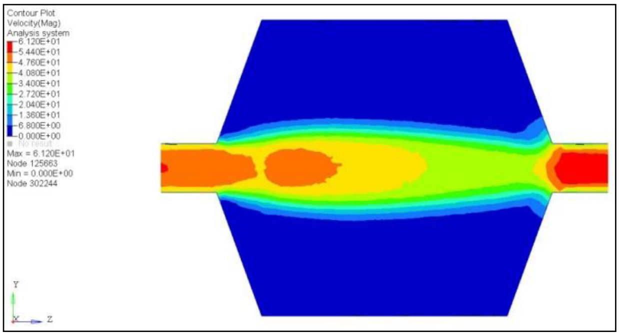

Fig. 5. Velocity distribution of HPHE in case 1

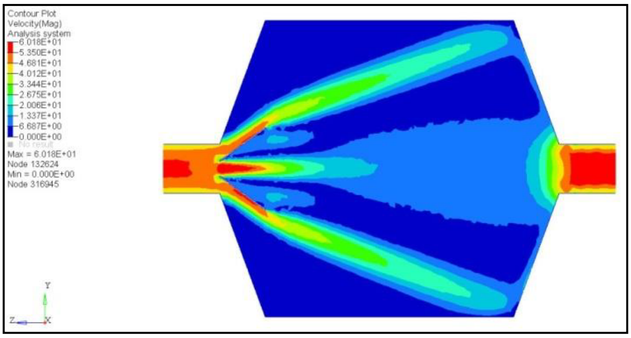

Fig. 6. Velocity distribution of HPHE in case 2

In case 1, the gas flow approaches the inlet at an average velocity of 54.15 m/s, whereas the gas flows out from the outlet with an average velocity of 61.05 m/s. In case 2, the gas enters the inlet and exits from the outlet at an average velocity of 54.28 m/s and 56.61 m/s respectively. From the results of the simulation, the distribution flow of velocity in case 2 shows a better dissemination compared to case 1. The flow is perfectly distributed along the center of the HPHE and also along the right and left side of the heat pipe. In contrast, the flow distribution of velocity in case 1 is only focused at the center of heat pipe which is not fully allocated over the whole tube bundle completely. Therefore, it can be concluded that in case 1, the basic design of HPHE was not fully utilized since some parts of the tube bundle did not expose to the gas flow thus the efficiency of HPHE to transfer the heat is low. Comparing this two model design, this study would recommend case 2 which has a better distribution flow that increases thermal efficiency.

As for temperature distribution, the result in both cases is shown in Fig. 7 and Fig. 8. In case 1, the average temperature at the inlet of HPHE is 793K, while the average temperature at the outlet is 779.6K. Fig. 7 illustrates the temperature distribution where the heat of the gas flow did not affect the areas of HPHE very much. This probably because of some parts of the tube bundle has not exposed really well to the heat. Thus, the heat transfer is low in case 1, which leads to inappropriate heat recovery. However, there are slightly significant differences shown in Fig. 8 for case 2. In case 2, the average temperature at the inlet is 793K and the average temperature at the outlet is 774.8K. The temperature distribution in case 2 is considerable compared to case 1. The development of dispersion in temperature distribution in case 2 shows the best distribution flow between both cases that had been studied earlier. Therefore, from this analysis it can be concluded that case 2 has the highest rate of temperature distribution that leads to higher heat transfer and boost the thermal efficiency in HPHE.

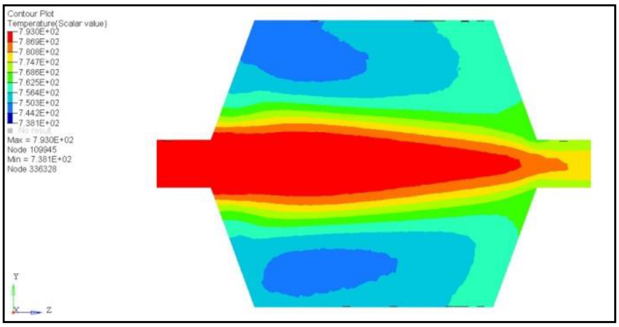

Fig. 7. Temperature distribution of HPHE in case 1

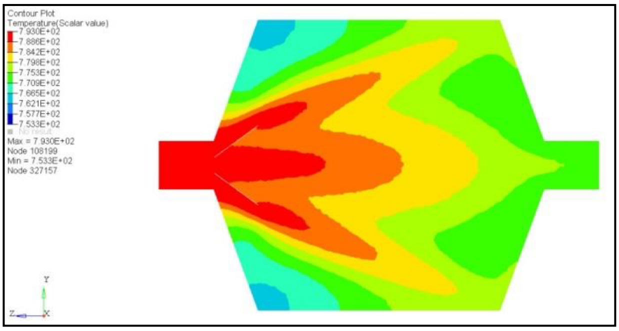

Fig. 8. Temperature distribution of HPHE in case 2

6 Results and analysis conclusion

The aim of this paper is to analyse fluid conditions on the tube bundle in the HPHE by using CFD packages i.e. AcuSolve. CFD modelling was carried out to show the ability of this approach as well as to investigate the flow pattern in HPHE. Two idealized models of HPHE were designed in this study with different variants of entry's dimension set to be case 1 and case 2. From the present investigation, the following conclusions can be drawn:

(1) Cone inserted in case 2 are very effective and can increase the performance of HPHE. An appropriate development of HPHE with differing size of inserting cone should be discussed in further study.

(2) CFD solver program that has been used in this study are acceptable and suitable to simulate and analyse the HPHE performance.

(3) The design of HPHE is the key to provide a heat exchange system to work proficiently as expected. The result is used to optimize and improving heat recovery systems of the increasing demand for energy efficiency in industry.

The authors would like to acknowledge Universiti Teknikal Malaysia Melaka (UTeM) for financial support under UTeM short term grant PJP/2015/FTK(1C)/S01394 and PJP/2015/FTK(8C)/S01411.

References

1. E. Firouzfar, M. Attaran, Int. Sch. Sci. Res. Innov., **2**, 266 (2008)
2. Y. H. Yau, M. Ahmadzadehtalatapeh, Appl. Therm. Eng., **30**, 77 (2010)
3. H. Ma, L. Yin, X. Shen, W. Lu, Y. Sun, Y. Zhang, N. Deng, Appl. Energy., **169**, 177 (2016)
4. Y. Yuan, Y. Lu, H. Bao, Y. Wang, W. Wang, A.P. Roskilly, Energy Procedia, **61**, 2141 (2014)
5. I. Sauciuc, A. Akbarzadeh, P. Johnson, Heat Recover. Syst. CHP., **15**, 631 (1995)
6. M. Cruchaga, D. Celentano, Int. J. Comut. Fluid Dyn., **16**, 247 (2002)
7. S. Lin, J. Broadbent, R. McGlen, Appl. Therm. Eng., **25**, 127 (2005)
8. A. Alizadehdakhel, M. Rahimi, A.A. Alsairafi, Int. Commun. Heat Mass Transf., **37**, 312 (2010)
9. M. Saber, H. Ashtiani, in Proceedings of 5th IASME/WSEAS International, (2010)
10. T. Javaherchi, *Wind Engineering*, (University of Washington, 2010)