Experimental and Numerical Analysis of Flow and Heat Transfer in a Gas-Liquid Thermosyphon Heat Exchanger in a Pilot Plant

Zare Aliabadi, Hassan*+; Atashy, Hossein
Department of Chemical Engineering, University of Sistan and Baluchestan, Zahedan, I.R. IRAN

Noie, Seyed Hossein
Department of Chemical Engineering, Ferdowsi University, Mashhad, I.R. IRAN

Khoshnoodi, Mohamad; Khorram, Mohammad
Department of Chemical Engineering, University of Sistan and Baluchestan, Zahedan, I.R. IRAN

Khoshvaght, Mohammad
Department of Chemical Engineering, Islamic Azad University, Shahrood Branch, Shahrood, I.R. IRAN

ABSTRACT: *A numerical and experimental investigation of flow and heat transfer in a gas- liquid Thermosyphon Heat Exchanger "THE" with built in heat pipes and aluminum plate fins for moderate Reynolds numbers has been carried out. It's module is composed of 6 rows and 15 columns copper pipes with aluminum plate fins with dimensions of 130cm height, 47cm width and 20cm depth. The tubes have been filled with water with filling ratios of 30%, 50% and 70%. The density and thickness of fins are 300 fin/m and 0.4mm, respectively. The configuration of tubes is in-line with 30mm pitch. This paper presents the distribution of temperature and thermal performance of "THE" by using CFD modeling. A good comparison of the present CFD modeling results for the modeling thermosyphon heat exchanger with experimental results of hydrodynamic and thermal behavior is achieved.*

KEY WORDS: *Thermosyphon heat exchanger "THE", CFD modeling, In-line configuration, Pressure drop, Thermal performance.*

INTRODUTCION

Heat recovery, one of the methods of energy conservation can be successfully implemented when the investment cost of additional equipment required is acceptably low. Thermosyphon based heat exchangers are very simple devices that can be used to transfer heat between two fluid phases. Features include no

* To whom correspondence should be addressed.
+ E-mail: h_a_zare@yahoo.com
1021-9986/10/2/139 9/$/2.90

cross-contamination between streams, no moving parts, compactness and no need for any external power supply [1]. In this section, a brief review of some of experimental and theoretical research has been presented. *Rosman et al.* [2] investigated numerically and experimentally one and two-row fin tube heat exchanger. They used isothermal fins for their calculations, the reason being the narrow space between the two adjacent fins. Two-dimensional simulations of flow past a tube bundle have been reported by *Launder & Masssely* [3], *Wung, & Chen* [4]. In recently published papers, it is observed that slender vortex generators can improve the heat transfer, keeping the pressure penalty at a modest level. *Biswas et al.* [5] and *Tsai et al.* [6] reported numerical investigations on related topics. In the above investigations, the enhancement of heat transfer from the fin surfaces was achieved by disrupting the growth of thermal boundary layer. *Song Lin et al.* [7] have presented a design method by using CFD simulation of the dehumidification process with heat pipe heat exchanger. Their studies illustrate that the CFD modeling is able to predict the thermal performance of the dehumidification solution with heat pipe heat exchanger (HPHE). *Tan & Liu* [8] have used the ε-NTU method to analyze an air-to-air heat pipe heat exchanger. They also have presented an equation to determine the optimum position separating a heat pipe into evaporator and condenser regions in a heat pipe heat exchanger by minimizing the total thermal resistance of the heat path. *Noie* [9] has carried out an experimental study of the performance of an air-to-air thermosyphon-based heat exchanger utilizing water as working fluid to investigate its behavior under different operating conditions. *Zare Aliabadi, H., et al.* [10] carried out an experimental research to investigate the hydrodynamic and thermal performance of a gas-liquid "THE". The effects of various parameters such as the heat capacity ratio of high- and low-temperature fluid streams "C_e/C_c", the inlet hot air temperature, and the mass flow rate or the inlet hot air velocity on thermal performance of a gas-liquid "THE" and the pressure drop across tube bundle of it have been investigated. In addition, a new correlation for "THE" with individual finned tubes and in-line geometry has been proposed for calculating pressure drop across tube bundle of a "THE".

In this research, we have investigated fluid dynamic and thermal performance of "THE" by using CFD modeling and experimental pilot plant results. Also we have illustrated several parameters such as mass flow inlet, maximum velocity of fluid in "THE" and inlet hot air temperature on the pressure drop across "THE" and the heat transfer of "THE". The effect of the hot air velocity at two constant values of 1 m/s and 1.5 m/s on the effectiveness and the rate of heat transfer are taken into account and are discussed.

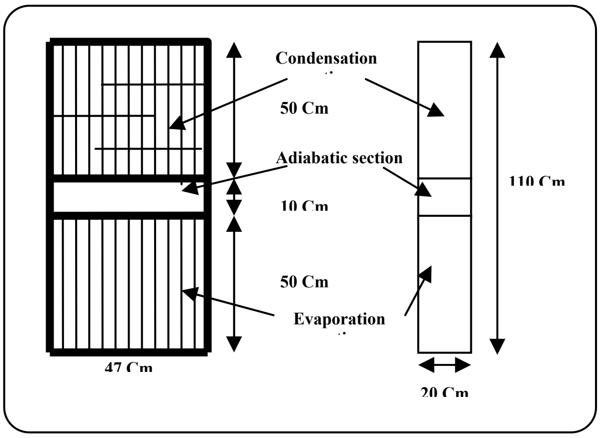

Fig. 1: The schematic diagram of "THE".

THEORETICAL SECTION

Statement of the Problem

Fig. 1 represents the schematic diagram of "THE". It has two sections, top and bottom. The top section is condensation part of the HPHE in which cold-water pumps into it with constant flow rate of 7L/min at about 17°C; the bottom section is evaporation part of the HPHE. The bottom duct is straight and forms a closed looped. Hot air is blown in the bottom duct across tube bundle within the range of 0.15-0.55kg/s.

The CFD modeling only illustrates in the bottom of "THE" (evaporation section). Air uses as the working fluid and the Prandtl number of the air assumes equal to 0.7.

The three- dimensional Navier-Stocks equations for laminar flow of an arbitrary spatial control volume V, bound by a closed surface S, can be express in the following general forms:

Continuity equation

$$\frac{\partial}{\partial t}\int_V \rho dv + \int (\rho\vec{u}).d\vec{s} = 0 \qquad (1)$$

Where, ρ and u represent the fluid density and the fluid velocity, respectively.

Momentum equation (y direction)

$$\frac{\partial}{\partial t}\int_{v}\rho u dv+\int_{s}(\rho u^{2}-\mu\nabla u).d\vec{s}=\int_{v}\frac{\partial p}{\partial y}dv \qquad (2)$$

Energy equation

$$\frac{\partial}{\partial t}\int_{v}\rho T dv+\int_{s}(\rho u T-\frac{k}{c_{p}}\nabla T).d\vec{s}=0 \qquad (3)$$

Boundary conditions

Top and bottom plates: the bottom plate is the mass flow inlet and velocity inlet.

$$(u=u_{\infty}, T=T_{\infty}, V=W=0 \text{ and } \frac{\partial p}{\partial x}=0)$$

The top plate is the out flow and the air goes out the thermosyphon heat exchanger from this one. The other plates, such as the surface of the circular tube and the internal surface of fin plate are represented as wall plates.

$$\left| \begin{array}{l} u=w=v=0 \\ T=T_{W} \end{array} \right. \qquad (4)$$

The sidewalls have symmetry conditions (the behind and the front of model).

$$\left| \frac{\partial u}{\partial y}=\frac{\partial w}{\partial y}=0 \right. \qquad (5)$$

$$v=0,\ \frac{\partial T}{\partial y}=0,\ \frac{\partial p}{\partial y}=0$$

It is assume, the wall temperature of the circular tube is the saturation temperature of water at the saturated pressure equal to 200 mmHg.

The material of plate fins is aluminum with 0.4mm thickness and the material of tubes is copper with 0.5mm thickness. The physical properties of aluminum and copper are constant. However, the physical properties of air vary as polynomial of temperature. Laminar regime and steady state conditions have been assumed for flow pattern across tube bundle, because the Reynolds number of the fluid flow is low.

Grid generation

Figs. 2 and 3 show a schematic representation of the three-dimensional grid and the flow direction used for the present computation. The grid generated and meshed by the Gambit Software.

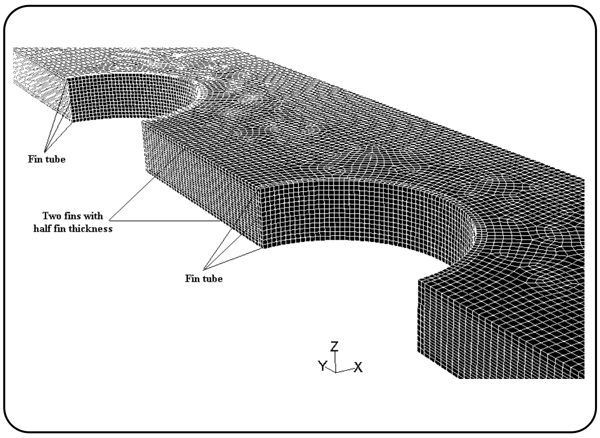

Fig. 2: Schematic of three-dimensional grid of "THE".

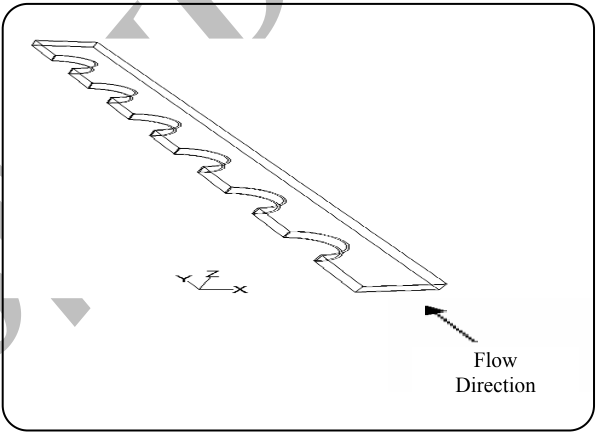

Fig. 3: Flow direction of fluid flow in defined model of "THE".

EXPERIMENTAL SECTION

In order to investigate the parameters affecting the hydrodynamics and thermal performance of Thermosyphon Heat Pipe Heat Exchanger (THPHE), a semi industrial pilot based on Fig. 4 was designed. This model consisted of the following parts: thermosyphon heat pipe heat exchanger, air channels, rotameter, a centrifugal blower, electrical heaters, thermometers, electrical board, fan speed regulator, orifice and manometer [10].

"THE" module is composed of 6 rows, 15 columns of copper pipes with aluminum plate fins with dimensions of 130cm(height), 47cm(width), and 20cm(depth) filled with water of filling ratio of 30%, 50% and 70% . The density and the thickness of fins are 300 fin/m and 0.4mm, respectively (Table 1). The configuration of tubes is in-line with 30mm pitch. The test rig has two sections, top and bottom. The top section is the condensation part of HPHE in which cold water pumps into it with constant

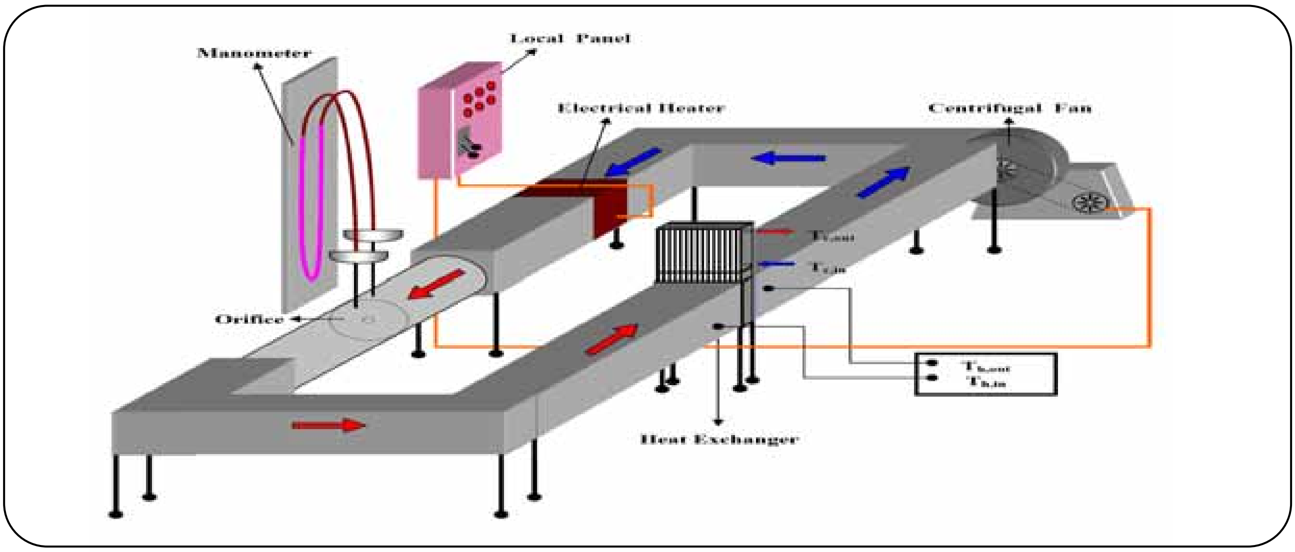

Fig. 4: Overall schematic of pilot plant.

Table 1: Specifications of the thermosyphon heat pipe heat exchanger.

Aluminum plate Thickness 0.4mm Density 300fin/m	Fin
In-line S_L=S_T=30mm	configuration
N_L=6 , N_T=156, N_T=15	Num. of heat pipe rows
N_{total}=90	Total num. of heat pipe
Copper-distillated water	Material & working fluid of HP

flow rate of 7 L/min at about 17°C; and then goes out from the top section after gaining the pipe's heat by passing above them. The flow of water is regulated by rotameter. The bottom section is evaporation part of HPHE. The bottom duct is straight and forms a closed looped. A centrifugal blower and 90 electrical heaters were installed inside the duct to circulate hot air through the evaporator section. In the bottom duct mass flow rate varied by changing the input electrical frequency of the blower in the range of 20-70H_Z; resulting in a variation of the mass flow rate within the range of 0.15-0.55kg/s. The pressure drop between inlet and outlet of "THE" was measured by inclined manometer. The inlet hot air temperature was controlled and kept constant at 100, 125, 150, 175, and 200°C while the inlet heat into the evaporator section was in the range of 6-42kW. The outlet hot air temperature was measured by digital thermometer. The experimental data was obtained by repeating the tests three times.

RESULTS AND DISCUSSION

Thermal analysis

The thermal performance of a gas to liquid "THE" was studied for the inlet hot air in the range of 125 to 225°C. Because of the similarity of the results, the result of inlet hot air at 125°C and 200°C are presented here. According to the experimental data and following correlations, the rate of heat transfer and thermal performance of "THE" was calculated by:

$$q = \left(\dot{m}_h c_{p_h}\right) \cdot \left(T_{h_i} - T_{h_o}\right) \tag{6}$$

To derive the thermal performance of "THE", it was necessary to calculate the maximum rate of heat transfer from the thermosyphon heat exchanger.

$$q_{max} = C_{min}\left(T_{h_i} - T_{c_i}\right) \tag{7}$$

Where, C_{min} is the minimum heat capacity fluid between hot and cold streams

$$C_{min} = \left(\dot{m}C_p\right)_{min} \tag{8}$$

$$\varepsilon = \frac{q_{act.}}{q_{max}} = \frac{C_e\left(T_{h_i} - T_{h_o}\right)}{C_{min}\left(T_{h_i} - T_{c_i}\right)} = \frac{C_c\left(T_{c_o} - T_{C_i}\right)}{C_{min}\left(T_{h_i} - T_{c_i}\right)} \tag{9}$$

In addition, C_e and C_c are the heat capacities of the fluid streams in evaporator and condenser sections of the heat pipe heat exchanger, respectively.

Table 2: Experimental results of thermal analysis on the thermosyphon heat exchanger (Inlet hot air at 200°C).

$\dot{m}_h$ (kg/s)	$\dot{m}_c$ (kg/s)	T_{ho}(°C)	T_{hm}(°C)	$\dot{m}_h C_{p,h}$	q_{exp} (kW)	ε	ΔP_{exp} (mmH_2O)
0.165	0.1	70	135	0.168	21.84	0.212	6
0.231	0.1	82	141	0.235	27.73	0.221	14
0.294	0.1	92	146	0.3	32.4	0.264	24
0.366	0.1	102	151	0.372	36.95	0.299	34
0.43	0.1	96	148	0.438	45.614	0.575	46
0.493	0.1	86	143	0.503	57.3	0.629	62

$$C_e = \left(\dot{m}c_p\right)_e \ , \ C_c = \left(\dot{m}c_p\right)_c \tag{10}$$

Now, Eq. (9) rearranges as equation:

$$\varepsilon = \frac{T_{c_o} - T_{c_i}}{T_{h_i} - T_{c_i}} \ , \text{ if } \ C_e < C_c \tag{11}$$

$$\varepsilon = \frac{T_{h_i} - T_{h_i}}{T_{h_i} - T_{c_i}} \ , \text{ if } \ C_e < C_c \tag{12}$$

The experimental results of the thermal performance for the inlet hot air at 200°C are illustrated in Table 2.

CFD modeling

This part of research dealt with investigating the use of our defined model that was meshed in the gambit software and modeled by fluent software 6.0. The effectiveness and the rate of heat transfer of "THE" are obtained according to hydrodynamic and thermal modeling results. Because of the similarity of results, the result of inlet hot air at 200°C is presented here (Fig. 5, Tables 3, 4).

Comparison between experimental results and CFD modeling

The comparison between experimental and CFD modeling results of pressure drop across "THE", the outlet temperature of hot air and the rate of heat transfer of a gas to liquid "THE" has been studied for inlet hot air in the range of 125-225°C. Because of the similarity of results, only the comparison of inlet hot air at 200°C is presented here (Figs. 6, 7, 8). It is observed that by increasing the hot air mass flow rake grater than 0.4 kg/s the CFD results varies from experimental data, because of the flow regime changed to turbulent. We have obtained the parameters of fluid flow in the bottom duct of the test rig as follows:

$$U_\infty = \frac{\dot{m}}{\rho_m A} \tag{13}$$

Where, A is the cross sectional area of the duct, here equal to 0.3 m^2 for our test rig of this research.

The maximum velocity of the inlet hot air through the thermosyphon heat exchanger was obtained as follows:

$$U_{max} = U_\infty \frac{S_L}{S_L - d} \tag{14}$$

The configuration of tubes is in-line with 30mm pitch and the outside diameter of tubes is 15mm.

Now, we can calculate Reynolds number of the airflow through the evaporation part of "THE" (Table 5).

$$Re_{max} = \frac{\rho_m u_{max} d}{\mu} \tag{15}$$

Where, ρ_m and μ represent density and viscosity of air at the average temperature between the inlet and the outlet temperature of air, respectively.

Effect of inlet hot air velocity

The effect of the hot air velocity at two constant values of 1 m/s and 1.5 m/s on the effectiveness and the rate of heat transfer are taken into account and are discussed. It is observed that due to decrement of the density of inlet hot air which results in less hot air mass flow rate, the effectiveness and the rate of heat transfer increased (Table 6 and Fig. 9).

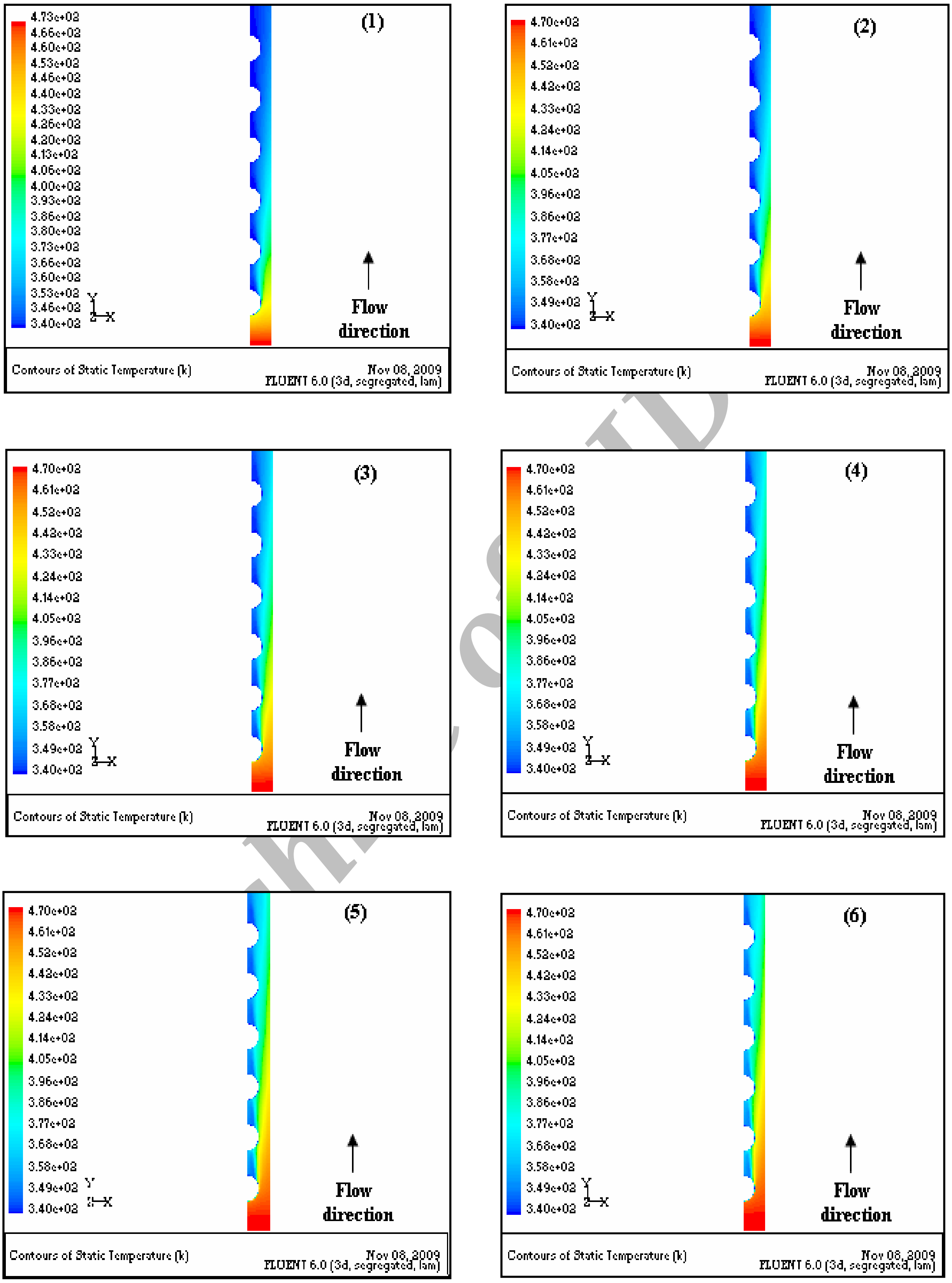

Fig. 5: Temprature distribution on thermosyphon heat exchanger obtained by CFD mod.

Table 3: CFD modeling results of outlet fluid temperature and pressure drop across "THE"(Inlet hot air at 200°C).

$\dot{m}_h$ (kg / s)	$\Delta P_{mod.}$ (mm H_2O)	$T_{h,\ out}$ (mod) (°C)
0.165	7.5	76.75
0.231	11.8	89.25
0.294	18.5	100.5
0.366	26.5	111
0.43	39	119.5
0.493	51	127

Table 4: CFD modeling results of thermal analysis on the thermosyphon heat exchanger (Inlet hot air at 200°C).

$\dot{m}_h$(kg/s)	$\dot{m}_c$(kg/s)	$T_{ho,\ mod}$ (°C)	T_{ci} (°C)	T_{co} (°C)	$q_{mod.}$ (kW)	ε
0.165	0.1	76.75	20	69.5	20.7	0.263
0.231	0.1	89.25	19.8	82.2	26.07	0.304
0.294	0.1	100.5	19.5	91.1	29.92	0.364
0.366	0.1	111	19.6	99.9	33.55	0.395
0.43	0.1	119.5	19.5	104.9	35.7	0.445
0.493	0.1	127	19.7	108.4	37.1	0.503

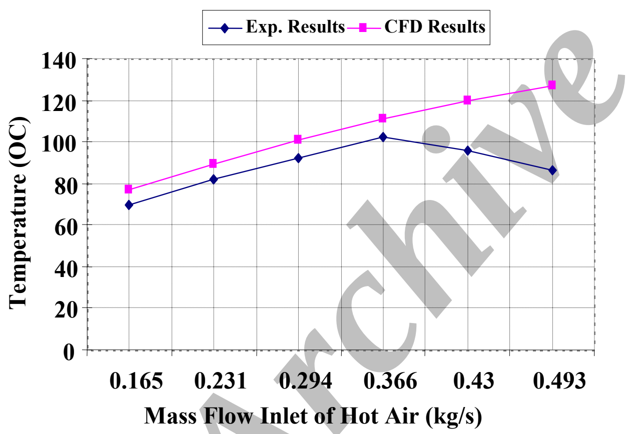

Fig. 6: The outlet air Temperature of "THE" vs. mass flow rate of inlet hot air.

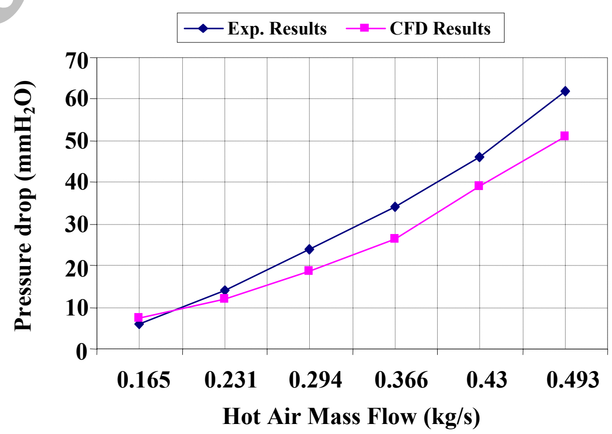

Fig. 7: Pressure drop vs. mass flow rate of inlet hot air to "THE".

CONCLUSIONS

A three-dimensional numerical study by fluent software and experimental study on the flow and transfer characteristics in a gas to liquid "THE" has been performed. The effects of various parameters such as the inlet hot air temperature, and the mass flow rate or the inlet hot air velocity on thermal performance of a gas-liquid "THE" and the pressure drop across tube bundle of it have been investigated. The following conclusions have obtained:

1) A good agreement between experimental results of the effectiveness and the rate of heat transfer with theoretical model has been achieved.

2) At constant inlet hot air temperature, by increasing the inlet hot air velocity, the effectiveness and the rate of heat transfer increased. Due to the heat transfer coefficient of high- temperature fluid stream increases.

Table 5: Characteristics of the fluid flow through the evaporation section of the thermosyphon heat exchanger.

$\dot{m}_h$ (kg / s)	U_∞ (m/s)	U_{max} (m/s)	Re_{max}
0.165	0.638	1.276	702
0.231	0.91	1.82	970
0.294	1.172	2.344	1251
0.366	1.472	2.944	1558
0.43	1.744	3.488	1950
0.493	2.01	4.02	2350

Table 6: The rate of transferred heat from "THE".

a - Inlet hot air velocity at constant value 1 m/s

T_{inlet} (°C)	q_{exp} (kW)	q_{mod} (kW)
125	13.5	13.159
150	16	17.284
175	18	19.709
200	21.5	24.034
225	24	27.658

b - Inlet hot air velocity at constant value 1.5 m/s

T_{inlet} (°C)	Q_{exp} (kW)	Q_{mod} (kW)
125	17	16.927
150	21	21.789
175	25	26.251
200	27.5	29.913
225	31	34.576

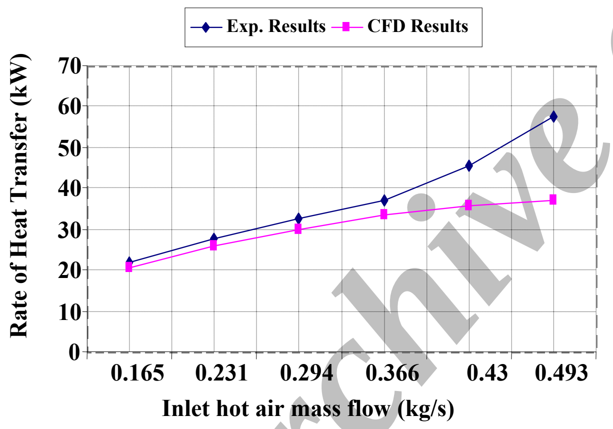

Fig. 8: The rate of heat transfer by a "THE" vs. the inlet mass flow rate.

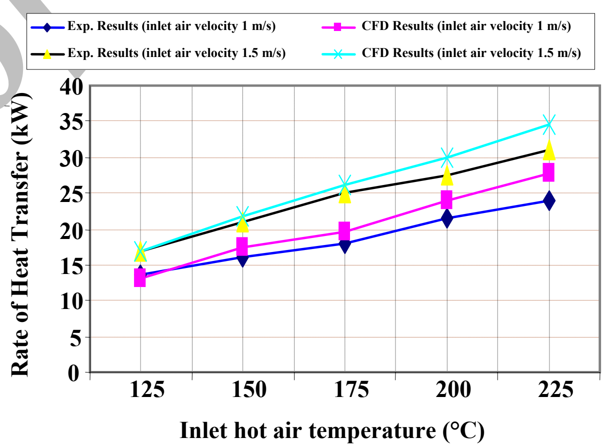

Fig. 9: The rate of heat transfer of by "THE" vs. the temperature of inlet hot air (At constant fluid velocity) (Comparison between experimental and theoretical results).

3) According to 40 experimental points for pressure drop across tube bundle of "THE", the variation of CFD results obtained for the pressure drop are in the range of 7 to 25 percent.

4) Also, the average variation between experimental and CFD modeling results obtained for the effectiveness and the rate of heat transfer are 22.5% and 10%, respectively.

5) Since we assumed a laminar regime for flow pattern of fluid through of "THE" in CFD modeling, it is observed that by increasing the hot air mass flow rate rather than 0.4 kg/s the CFD results deviated from experimental data, because of the flow regime become turbulent.

Nomenclature

C	Heat capacity of fluid, (W/°C)
C_e	Heat capacity of hot fluid, (W/°C)
C_c	Heat capacity of cold fluid, (W/°C)

$C_{p,h}$ and $Cp_{,c}$	Specific heat of capacity of hot and Cold fluid, (W/kg.°C)
C_{min}	The minimum Heat capacity between hot and cold fluids, (W/°C)
k	Thermal conductivity of fluid, (W/m°C)
$\dot{m}$	Mass flow rate of fluid in duct, (kg/s)
q	Heat transfer flux, (W/m^2)
$\Delta P_{exp.}$	Experimental pressure drop, (Pa, and mmH_2O)
$\Delta P_{mod.}$	Pressure drop obtained from the CFD modeling, (Pa, and mmH_2O)
S_L	Longitudinal tube pitch, (mm)
T_w	Wall temperature, (°C)
T_{hi}	Temperature of hot flow, inlet of THE, (°C)
T_{ho}	Temperature of hot flow, outlet of THE, (°C)
T_{ci}	Temperature of cold flow, inlet of THE, (°C)
T_{co}	Temperature of cold flow, outlet of THE, (°C)
u, v, w	Velocity, (m/s)
u_∞	Ambient velocity of fluid, (m/s)
u_{max}	maximum velocity of the inlet hot air Through the thermosyphon heat exchanger, (m/s)

Subscribes

c	Condenser, cold
h	Hot
e	Evaporator
i	Inside
o	Outside, Overall

Greek letters

ρ	Density of fluid, kg/m^3
ε	Effectiveness
μ	Dynamic viscosity of fluid,N.s/m^2

Received : Jan. 18, 2009 ; Accepted : Aug. 15, 2009

REFRENCES

[1] Vasiliev Leonard L., Heat Pipes in Modern Heat Exchangers, *Applied Thermal Engineering*, **25**, p.1 (2005).

[2] Rosman E.C., Carajilescov P., Performance of One and Two Row Tube and Plate Fin Heat Exchanger, *J. Heat Transfer*, **106**, p.627 (1984).

[3] Massey T.H., Launder B.E., The Numerical Prediction of Viscous Flow and Heat Transfer in Tube Banks, *J. Heat Transfer*, **100**, p.565, (1978).

[4] Wung T.S., Chen C.J., Finite Analytic Solution of Convective Heat Transfer for Tube Arrays in Cross Flow, Part 1 and Part 2, *J. Heat Transfer*, **111**, p.633, (1989).

[5] Biswass G., Mitra N.K., Heat Transfer Enhancement in Fin Tube Heat Exchangers by Winglet-Type Vortex Generators, *Int. J Heat Mass Transfer*, **35**, p.283, (1994).

[6] Tsai S.F., Sheu T.W., Heat Transfer in a Conjugate Heat Exchanger with a Wavy Fin Surface, *Int. J Heat Mass Transfer*, **42**, p.1735, (1999).

[7] Lin S., Broadbent,J., McGlan R., Numerical Study of Heat Pipe Application in Heat Recovery Systems, *Applied Thermal Engineering*, **25**, p.127 (2005).

[8] Tan J.O., Liu C.Y., Predicting the Performance of a Heat Pipe Heat Exchanger using the Effectiveness NTU Method, *Int. J. Heat Fluid Flow*, **11**(4), p.376 (1990).

[9] Noie S.H., Investigation of Thermal Performance of Air-to-Air Thermosyphon Heat Exchanger using ε-NTU Method, *Applied Thermal Engineering*, **26**, p.1073 (2005).

[10] Zare Aliabadi H., Atashi H., Noie S.H., Khoshnoodi M., Experimental Investigation on Hydrodynamic and Thermal Performance of a Gas to Liquid Thermosyphon Heat Exchanger in a Pilot Plant, *Iran. J. Chem. Chemi. Eng.*, **27(3)**, p.1073 (2008).