Academic Platform Journal of Engineering and Science

journal homepage: http://apjes.com/

Numerical Analysis of Heat Transfer of a Brazed Plate Heat Exchanger

[1]Gökhan Sevilgen, *[2]Halil Bayram
[1]Department of Automotive Engineering, Faculty of Engineering, Bursa Uludağ University, 16059, Bursa, Turkey,
gsevilgen@uludag.edu.tr,
[2]Department of Mechanical Engineering, Faculty of Technology, Amasya University, 05100, Amasya, Turkey
halil.bayram@amasya.edu.tr,

Research Paper | Arrival Date: 01.02.2020 | Accepted Date: 14.05.2020

Abstract

In this paper, the heat transfer characteristics of a brazed plate heat exchanger were investigated by using Computational Fluid Dynamics (CFD) method. Due to its compact structure, the plate type heat exchangers are used in many engineering applications such as manufacturing, automotive, etc. Heat exchangers are generally analyzed considering both rating and sizing parameters. We only examined the rating parameters such as flow direction, mass flow rate, and the inlet temperature value of hot water on the thermal performance of a brazed plate heat exchanger, numerically. We also conducted experiments under the same conditions for getting comparative results with the numerical simulations. Theoretical calculations were achieved by using the Logarithmic Mean Temperature Difference (LMTD) method for a single-phase flow and the thermal efficiency of the heat exchanger was evaluated for different conditions. The maximum total heat transfer rate was calculated about 3.1 kW for experimental study whereas this value was about 3 kW for numerical study under the counter flow conditions. Considering the increase in percentage for the total heat transfer rate, the mass flow rate had more effects than the other rating parameters such as flow direction and hot water inlet temperature values. Another important result was that the effectiveness values of the heat exchanger were calculated higher under the counter flow conditions. The numerical results were in good agreement with the experimental data used in the study.

Keywords: Brazed plate heat exchanger, counter-flow, LMTD method, thermal performance, CFD

1. INTRODUCTION

Heat exchangers are devices that perform heat transfer between two or more fluid zones and can be classified according to their different properties such as processes type, flow arrangements, and heat transfer mechanisms [1, 2]. On the other hand, mild steels and alloy steels, copper alloys and for special purposes ceramic and graphite were commonly used materials for the construction of heat exchangers. A great number of factors need to be considered such as durability under different environmental conditions, corrosion resistance, heat transfer coefficients, the efficiency of the heat exchanger, total pressure drop etc. during the manufacturing process. Moreover, the manufacturing methods include increasing the total surface area by using porous substrates or formation of adding surfaces to enhance the total heat transfer rate [3]. Direct Metal Laser Sintering (DMLS) method was suitable for producing complex geometries as an alternative to existing manufacturing methods. This method was more preferred when to produce a compact heat exchanger which had smaller volume and higher thermal performance [4]. For many industrial applications plate heat exchangers have shown advantages such as small size and weight, high thermal efficiency, less maintenance cost, higher total heat transfer coefficient, easily installation, compactness, etc. over the other types of heat exchangers [2, 5-9]. The plate heat exchanger consists of a series of adjacent corrugated or wavy thin metal plates with complex geometries. The heat transfer occurs through plates and brazed plate heat exchangers are more suitable for use with higher temperature and pressure values than the other type of plate heat exchangers. On the other hand, brazing the stainless-steel plates together eliminates the need for sealing gaskets and thick frame plates. Due to geometrical structures of plate surfaces, some mechanisms occur such as swirl and vortex flows, disruption and reattachment of boundary layers, small hydraulic diameter flow passages, and increased effective heat transfer area. And these mechanisms were the basic factors to get the higher heat transfer coefficient values compared to the tubular heat exchangers. The weight and the volume values of the plate heat exchanger for the same effective heat transfer area are approximately 30% and 20%, respectively, compared to the values of shell and tube heat exchangers. Moreover, there is no need for insulation because only the edges of the plates are in contact with the outdoor. And the heat transfer between the heat exchanger and outdoor can be acceptable as negligible [10]. A well designed and built computational fluid dynamics (CFD) numerical model can be a useful tool to obtain the flow and heat transfer

*Corresponding Author: Makine Mühendisliği Bölümü, Teknoloji Fakültesi, Amasya Üniversitesi, 05100, Amasya, Turkey, halil.bayram@amasya.edu.tr

Doi: 10.21541/apjes.683151

characteristics of a heat exchanger. The CFD method can be utilized to get the appropriate flow and heat transfer characteristics considering the desired heat transfer rate and pressure drop [11]. The CFD method can be used both in the rating and iteratively in the sizing of heat exchangers. In this numerical study, a three-dimensional CFD model of a brazed plate heat exchanger was developed, and the numerical results were also compared to the experimental data obtained from this study. We also employed theoretical calculations for the total heat transfer rate by using the LMTD method and evaluated thermal efficiency for different conditions. The novelty and the main contributions to the current literature of the present study may be stated as follow:

- To represent a numerical and experimental approach for the evaluation of a brazed plate heat exchanger considering rating parameters.
- The more realistic a three-dimensional CFD model includes both solid and fluid zones is performed to get the fluid flow and heat transfer analysis of a small type brazed plate heat exchanger.
- To present detailed calculations based on the LMTD method and effectiveness for getting comparative results in such numerical calculations.

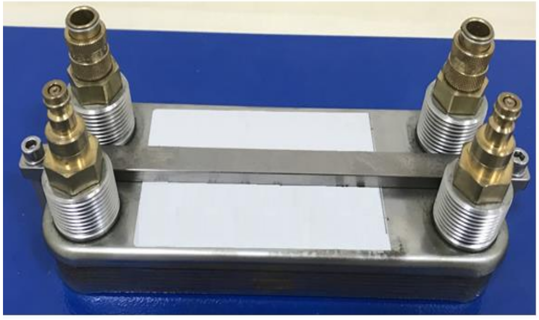
2. MATERIALS AND METHODS

2.1. Numerical Simulation

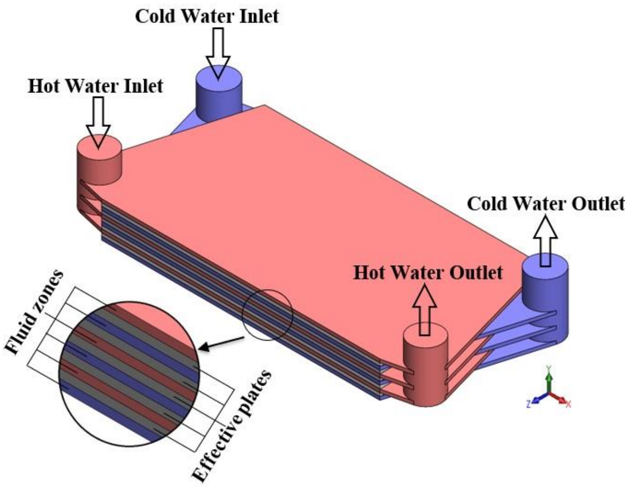
The Computer Aided Design (CAD) model of a brazed plate heat exchanger was developed by using the dimensions of the heat exchanger used in the experimental study and this heat exchanger is shown in Figure 1a. The plate heat exchanger had seven plates and the material of these plates was stainless steel in this study. The dimensions and the technical properties of this heat exchanger are listed in Table 1 and the CAD model is shown in Figure 1b. The computational domain consists of solid and fluid zones. In the solid zones, there were seven plates include cover and effective ones demonstrated in Figure 1b. However, the effective plates which shown in grey were only used in CFD calculations. In the fluid zones, the red and blue regions represent the hot and cold water, respectively.

Table 1. Technical details of the plate heat exchanger

Brazed Plate Heat Exchanger	
Plate type	Brazed
Total heat transfer area	0.048 m^2
Plate material	Stainless steel
Number of plates	7



(a)



(b)

Figure 1. (a) A brazed plate heat exchanger used in the experimental study (b) CAD model of a brazed plate heat exchanger used in the numerical study

The mesh structure of the numerical model is shown in Figure 2. Due to the importance of the stability and precision of the numerical results, we used the Cartesian mesh structure which consists of mostly hexahedral elements. The numbers of total elements were determined considering mesh independent results. Due to mesh structure significantly effects the numerical results and also computing time, mesh dependence test was performed. As a result, it was found that six million elements were quite enough to get the mesh-independent results and slightly high number of total elements had occurred because the plates are quite thin that to provide the heat conduction analysis.

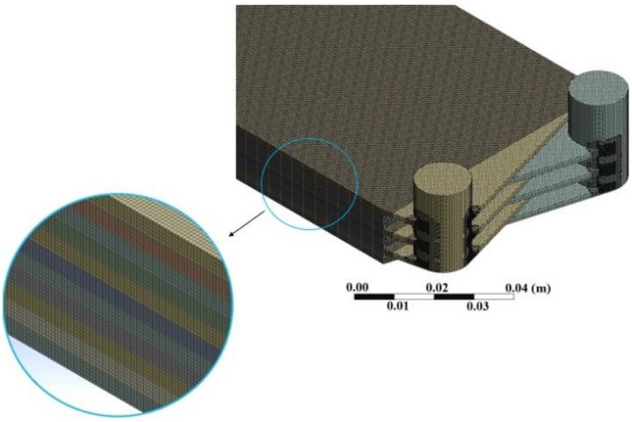


Figure 2. The mesh structure of the brazed plate heat exchanger used in the numerical calculations

In the numerical calculations, ANSYS-Fluent software package was used. In the numerical calculations, second order discretization method was used for convection terms and SIMPLE algorithm was chosen for pressure-velocity coupling. The numerical calculations were performed with steady-state conditions. The convergence is assumed when the normalized residuals of the flow equations are less than 10^{-3} and the energy equation is less than 10^{-6}. The equation for conservation of mass can be written in Equation 1. Fluent software package solves continuum, energy and transport equations numerically and the equation for conservation of mass can be written in Equation 1.

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \tag{1}$$

Conservation of momentum and energy equations are given in Equations 2-3.

$$\nabla.\left((\rho V)V\right)=-\nabla p+\nabla.(\bar{\bar{T}})+\rho g+F \tag{2}$$

$$\nabla.\left(V(\rho E+p)\right)=\nabla\left[k_{eff}\nabla T-\sum_{j}h_iJ_i+(\bar{\bar{T}}_{eff}.V)\right]+S_h \tag{3}$$

In these equations p is the static pressure, T is the stress tensor, g is the gravitational acceleration, ρ is the density and F can be given by the user of the term referring to other sources, E is unit of energy, $k_{eff}\nabla T$ is transmission, $\sum_j h_iJ_i$ is diffusion and $\bar{\bar{T}}_{eff}.V$ is viscous energy loss [4]. In the CFD model, three different computational domains named hot fluid zone, solid plates, and cold fluid zone were modeled that heat transfer occurred through these plates and the surfaces of each fluid zone and plates in which contact with the environment were assumed as adiabatic. In the numerical calculations, the mass flow rate ranged from 0.02 kg/s to 0.04 kg/s and hot water inlet temperature values were ranged from 45°C to 60°C, respectively. The boundary conditions used in the numerical study are shown in Table 2.

Table 2. The boundary conditions and solution settings used in the numerical calculations

Supply temperature of hot water	45 - 60 °C
Supply temperature of cold water	10 °C
Mass flow rate (both sides)	0.02 - 0.04 kg/s
Outer surfaces (fluid domains and plates)	Adiabatic conditions
Outlet nozzle (both sides)	Gauge pressure equals to 0 Pa
Simulation conditions	Steady state
Turbulence model	Standard k-ε turbulence model
Solver type	Pressure-based solver
Mesh structure and total elements	Cartesian mesh structure includes six million hexahedral
Pressure-velocity coupling algorithm	SIMPLE
Discretization method	Second order upwind scheme

The Reynolds (Re) number can be calculated by using Equations 4 and 5 to define the flow characteristics inside the heat exchanger [2]. In plate heat exchangers, due to the very small hydraulic diameters, turbulent conditions are observed at very low Re [12]. According to the calculated Re values for both mass flow rates, all the CFD analyses were performed for turbulent flow. The standard k-ε model was chosen for the turbulent modeling and this turbulence model is generally used for such calculations because of its stability and precision [3, 4, 7].

$$Re=\frac{G_{ch}D_e}{\mu} \tag{4}$$

$$G_{ch}=\frac{\dot{m}_{ch}}{N_{cp}bL_w} \tag{5}$$

2.2. Validation of the numerical results

The schematic view of the experimental set-up consists of two different fluid zones which provide hot and cold fluid recirculation is shown in Figure 3.

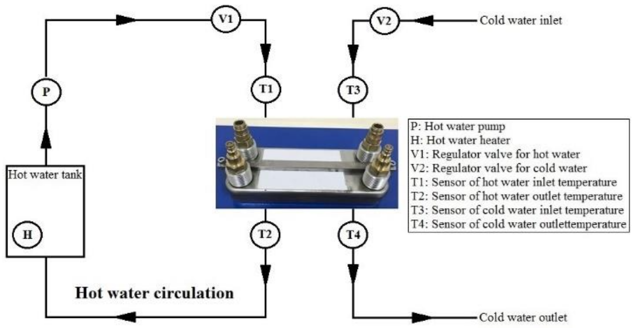


Figure 3. Schematic view of the experimental setup with brazed plate heat exchanger used in this study

In the experiments, we selected four different mass flow rates and inlet temperature values of the hot fluid side considering that the comparative results can be obtained when these type of heat exchangers works with low and high temperature sources. All experiments were performed until the steady state conditions were nearly achieved. Selected test conditions for all experiments were listed in Table 3 and the values of the mass flow rate of hot and cold water sides were set equal for each experiment and this value was ranged from 0.02 kg/s to 0.04 kg/s. The inlet temperature value of the cold water side was kept constant at 10°C during all experiments. On the other hand, the inlet temperature value of the hot water side increased gradually and changed between 45°C and 60°C considering low and high temperature sources.

Table 3. The performed experimental conditions in this study

Cases	Mass flow rate (both sides) (kg/s)	Inlet temp. values of hot water (°C)
Case 1	0.02	45
Case 2	0.04	45
Case 3	0.02	60
Case 4	0.04	60

2.3. Theoretical Calculations

In theoretical calculations, the total heat transfer rate and the total heat transfer coefficient can be calculated for brazed plate heat exchangers by using the LMTD method. In this method, the total heat transfer rate can be calculated by using Equation 6, where $\dot{m}_h$ (kg/s) is the mass flow rate, $c_{p,h}$ (kj/kgK) is the specific heat and the $T_{h,i}$ and $T_{h,o}$ are the inlet and outlet temperature values of the hot fluid, respectively. The total heat transfer rate can also be calculated from Equation 7 for the cold side, where $\dot{m}_c$ (kg/s) is the mass flow rate, $c_{p,c}$ (kj/kgK) is the specific heat and the $T_{c,i}$ and $T_{c,o}$ are the inlet and outlet temperature values of the cold fluid, respectively. The total heat transfer rate can be rewritten as stated in Equation 8, where U (W/m^{2}K) is the total heat transfer coefficient, A (m^2) is the total heat transfer area and the ΔT_{lm} is the logarithmic mean temperature difference which can be calculated from Equation 11, where ΔT_1 and ΔT_2 are the temperature differences described in Equations 9-10 considering flow direction. Thus, the total heat transfer coefficient can be determined from Equation 8 [2].

$$Q_h = (\dot{m}.c_p)_h.(T_{h,i} - T_{h,o}) \quad (6)$$

$$Q_c = (\dot{m}.c_p)_c.(T_{c,o} - T_{c,i}) \quad (7)$$

$$Q = U.A.\Delta T_{lm} \quad (8)$$

For parallel flow; $\Delta T_1 = T_{h,i} - T_{c,i}$ $\quad \Delta T_2 = T_{h,o} - T_{c,o}$ (9)

For counter flow; $\Delta T_1 = T_{h,i} - T_{c,o}$ $\quad \Delta T_2 = T_{h,o} - T_{c,i}$ (10)

$$\Delta T_{lm} = \frac{\Delta T_1 - \Delta T_2}{\ln(\Delta T_1/\Delta T_2)} \quad (11)$$

The heat exchanger effectiveness allows for getting comparative results. This can be calculated and written as follows in Equations 12 and 13 [2].

$$C_h = (\dot{m}c_p)_h \qquad C_c = (\dot{m}c_p)_c \quad (12)$$

$$\varepsilon = \frac{Q}{Q_{max}} = \frac{C_h(T_{h,i} - T_{h,o})}{C_{min}(T_{h,i} - T_{c,i})} = \frac{C_c(T_{c,o} - T_{c,i})}{C_{min}(T_{h,i} - T_{c,i})} \quad (13)$$

3. RESULTS AND DISCUSSIONS

The measured and calculated hot water side temperature data for the outlet section of the heat exchangers are shown in Table 4. The calculated temperature differences between inlet and outlet temperature values of hot side water are shown in Figure 4. According to these results, we can easily say that higher temperature values were obtained from parallel flow conditions compared to the counter ones for both experimental and numerical results. In general, higher temperature difference values were measured in cases that had low mass flow rates and counter flow for both experimental and numerical studies.

The maximum temperature difference was obtained for experimental study was for Case 3 for counter flow condition and its value was about 23°C in Case 3 for counter flow condition. Another important result is that the temperature values obtained from experimental and numerical studies were slightly different, but they had the same trends. In addition, for the all hot water outlet temperature values of both studies, the temperature values for parallel flow were calculated higher than the values of counter flow.

Table 4. Measured and calculated outlet temperature values of hot fluid sides for both parallel and counter flow (°C)

Cases	Experimental results		Numerical results	
	Parallel flow	Counter flow	Parallel flow	Counter flow
Case 1	32.2	29.6	31.86	30.21
Case 2	33.0	31.7	34.20	33.86
Case 3	40.9	36.9	40.17	37.92
Case 4	41.9	41.4	43.19	42.72

The calculated total heat transfer rate and the total heat transfer coefficient by using experimental data were shown in Table 5. From these results, when the inlet temperature value of the hot fluid side was 45°C, which was acceptable for low temperature source, the maximum calculated total heat transfer rate for counter flow conditions was about 1.9 kW for both experimental and numerical studies. When the inlet temperature value of hot fluid side was 60°C, which was accepted for high temperature source, the maximum calculated total heat transfer rate was nearly about 3.1 kW for experimental study, however, this value was about 2.9 kW for numerical study. The main reason for this difference is that the decrease in total effective heat transfer area due to simplifying the geometrical shape of plate which were selected as rectangular.

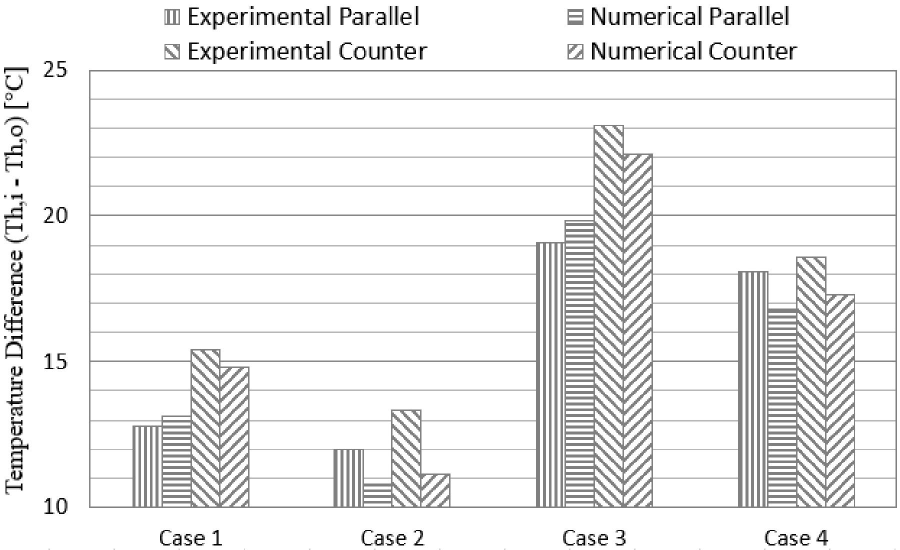


Figure 4. The calculated temperature difference values obtained from experimental and numerical study

Table 5. Calculated total heat transfer rate and total heat transfer coefficient values for both parallel and counter flow cases

Cases	Total heat transfer rate (W)					
	Exp. Results			Num. Results		
	Parallel flow	Counter flow	Diff. in Percent. (%)	Parallel flow	Counter flow	Diff. in Percent. (%)
Case 1	1069.66	1286.82	32.80	1098.28	1235.66	12.51
Case 2	2005.65	2222.86	10.83	1804.41	1862.09	3.20
Case 3	1597.31	1931.46	20.92	1657.96	1846.41	11.37
Case 4	3027.51	3111.07	2.76	2811.27	2891.01	2.84

Cases	Total heat transfer coefficient (W/m^{2}K)					
	Exp. Results			Num. Results		
	Parallel flow	Counter flow	Diff. in Percent. (%)	Parallel flow	Counter flow	Diff. in Percent. (%)
Case 1	1207.49	1474.50	22.11	1237.45	1301.11	5.14
Case 2	2095.01	2290.29	9.32	1681.45	1636.14	-2.69
Case 3	1349.01	1630.31	20.85	1323.65	1363.57	3.02
Case 4	2398.91	2296.72	-4.26	1846.14	1785.37	-3.29

From the comparison of the measured data for Case 2 and Case 4, when the hot water inlet temperature value was increased from 45°C to 60°C, the total heat transfer rate increased about 1.5 times in both parallel and counter flow conditions. The maximum enhancements in the total heat transfer rate were calculated by about 32.8% and 12.5% in the experimental and numerical study, respectively by changing the flow direction. According to the comparison of the total heat transfer coefficient values in Table 5, the maximum value was calculated about 2400 W/m^{2}K in the experimental study at high temperature heat source (60°C) with parallel flow conditions. The maximum total heat transfer coefficient value of the numerical study was approximately 1900 W/m^{2}K at high temperature heat source with parallel flow conditions, too. Calculated total heat transfer rates and the total heat transfer coefficients for different temperature values, mass flow rates and the flow directions for both experimental and numerical studies were shown in Figure 5 and Figure 6, respectively. In Figure 5, it can be easily seen that the total heat transfer rate of both studies increased with the increasingly hot water inlet temperature values at all mass flow rates.

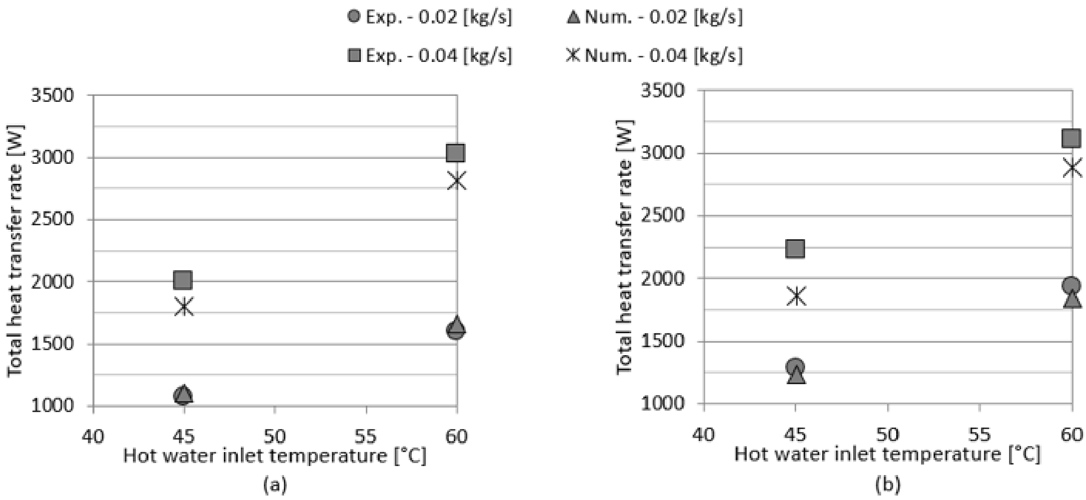


Figure 5. Calculated total heat transfer rate values for (a) parallel (b) counter flow conditions

According to results shown in Figure 6, the total heat transfer coefficients of both studies increased with increasing hot water inlet temperature values, too. However, these values were nearly the same in Case 4 under counter flow conditions.

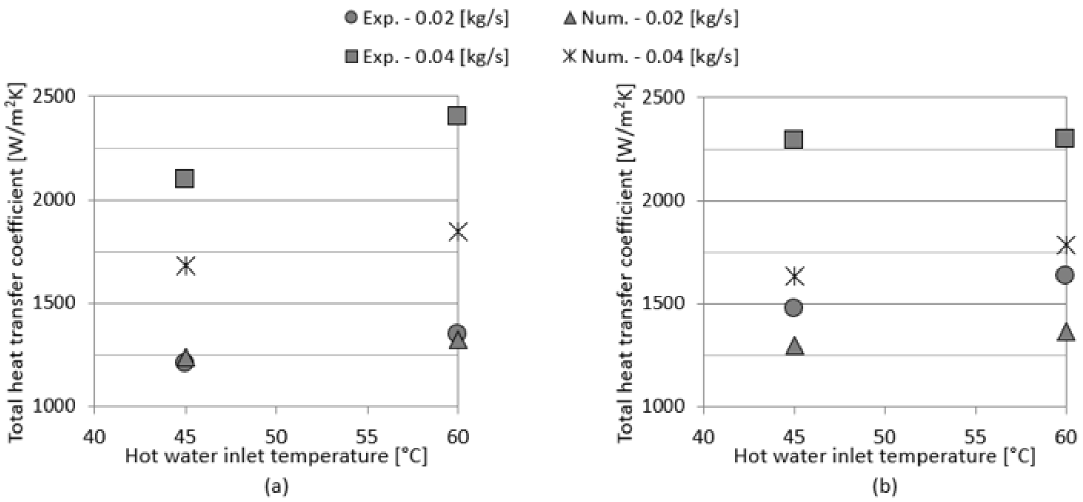


Figure 6. Calculated total heat transfer coefficient values for (a) parallel (b) counter flow conditions

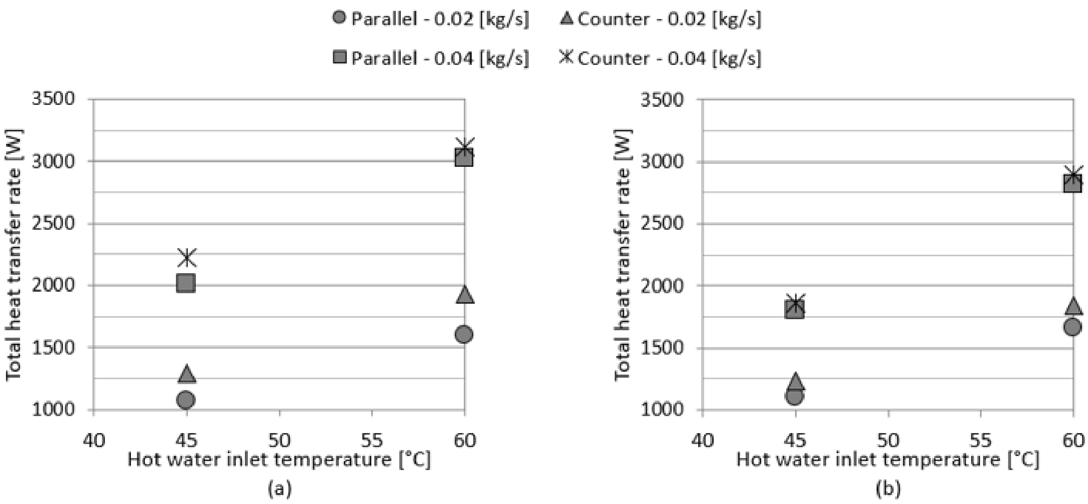


Figure 7. Comparison of calculated total heat transfer rate values of (a) experimental and (b) numerical studies under parallel and counter flow conditions

The effect of changing the flow direction on the total heat transfer rate of both experimental and numerical studies can be seen in Figure 7. According to the results, in the experimental study, the total heat transfer rate values of parallel and counter flow conditions at 60°C with a high mass flow rate (0.04 kg/s) were nearly equal. At all the hot water inlet temperatures, the calculated values of the total heat transfer rate for counter flow directions were higher than the calculated parallel ones under the same mass flow rates. The calculated total heat transfer rate and the total heat transfer coefficient were compared to the experimental data in Table 6. The total heat transfer rate and the total heat transfer coefficient values were calculated by using Equation 6 and these values were about 1085 W and 1225 W/m^{2}K, respectively, for the hot side and these calculated values were close to 1070 W and 1200 W/m^{2}K obtained from experimental results for Case 1 under parallel flow condition. The difference in the percentage of total heat transfer rate and total heat transfer coefficient obtained from numerical and experimental results was about 2.7% and 2.5%, respectively. From these results, we can easily say that the CFD tool can be useful for the prediction of the total heat transfer coefficient and the total heat transfer rate of the brazed plate of heat exchanger. The heat exchanger effectiveness values for all cases of the experimental study were calculated by using Equations 12 and 13 and the distribution values were shown in Figure 8. It can be easily seen that the counter flow conditions had better effectiveness values than parallel flow conditions for all cases. The predicted temperature data at the middle section plane of each fluid zone are shown in Figure 9 and 10. The predicted temperature values for the section planes of the cold side were ranged from 10°C to 27°C in general and these values were computed between 27°C and 45°C for the hot side.

Table 6. The calculated total heat transfer rate and total heat transfer coefficient for both sides and comparison to the experimental for Case 1

	Q (W)	U (W/m^2 K)
Experimental data	1069.66	1207.49
Numerical results	1098.28	1237.45

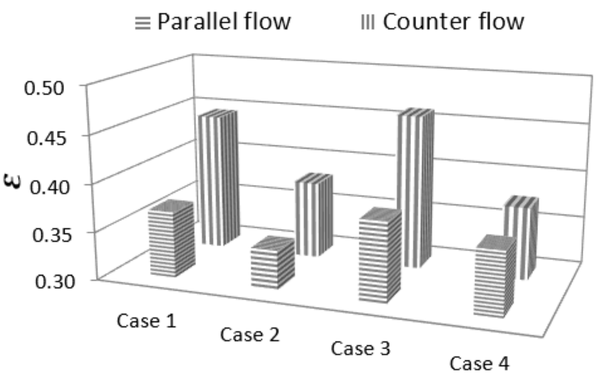


Figure 8. Effectiveness values of all cases calculated from experimental data

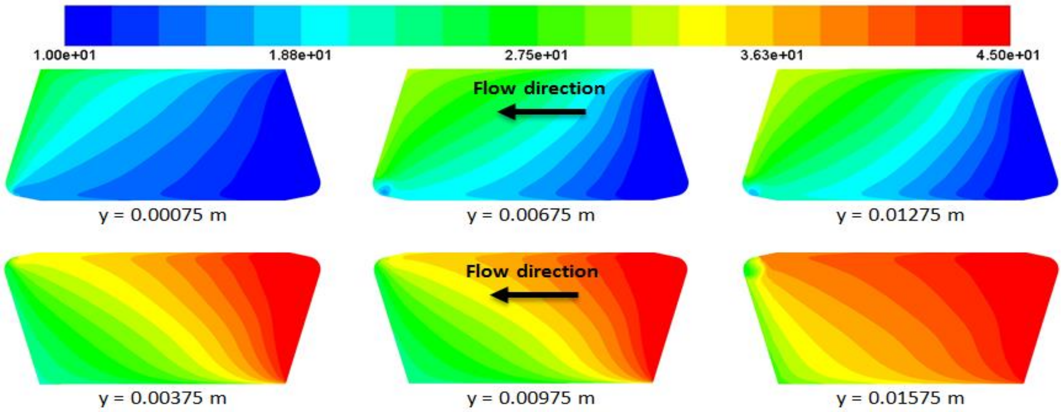


Figure 9. Predicted temperature distribution at the middle section plane of each channel for Case 1 and parallel flow conditions

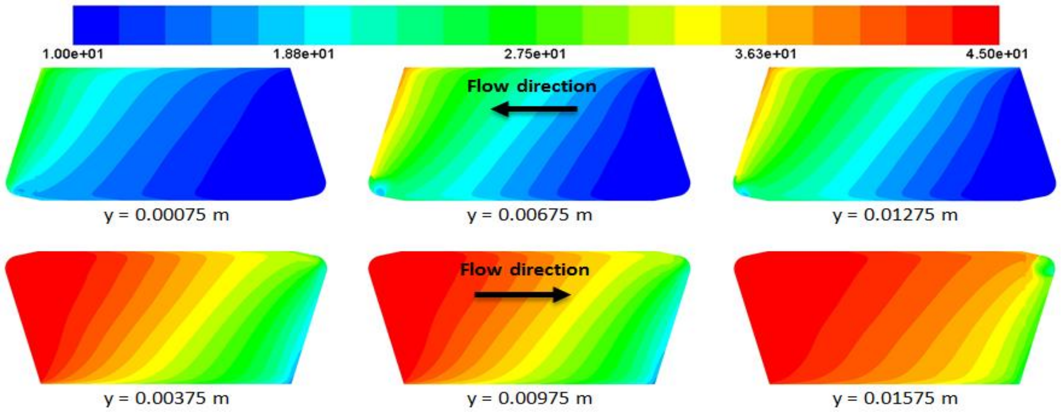


Figure 10. Predicted temperature distribution at the middle section plane of each channel for Case 1 and counter flow conditions

4. CONCLUSIONS

In this paper, the heat transfer characteristics of a brazed plate heat exchanger that used in many engineering applications due to its compact structure were performed numerically by using a three-dimensional CFD model. We also employed an experimental study for getting comparative results by changing the main parameters such as the mass flow rate, flow direction, and the inlet temperature value of hot water. These parameters were selected accordingly for getting the thermal performance of the heat exchanger by using lower and higher temperature sources. The theoretical calculations about the total heat transfer rate and the total heat transfer coefficient were achieved by using the LMTD method. Consequently, the main results of this study are listed below.

- The counter flow effects can easily be noticed in both experimental and numerical results. The enhancement in the total heat transfer rate was observed by changing the flow direction. The effectiveness results of counter flow conditions were considerably better than parallel ones for all cases.
- The maximum value of the total heat transfer coefficient was calculated about 2400 W/m^{2}K in the experimental study at a high temperature heat source (60°C) with parallel flow conditions. This value was obtained about 2300 W/m^{2}K at high temperature heat source with counter flow conditions. These values were close to each other and this indicates that source temperature had more effect on the total heat transfer coefficient than the flow direction especially at high temperature values.
- When the inlet temperature value of the hot fluid side was selected as a lower temperature source (45°C), the maximum calculated total heat transfer rate for counter flow conditions was obtained about 2 kW by using experimental and numerical data. When the inlet temperature value of the hot fluid side was selected as a higher temperature source (60°C), the maximum calculated total heat transfer rate was obtained about 3 kW by using experimental and numerical data for counter flow conditions. As a result, we can easily say that the increase in hot water inlet temperature values had a great effect on the heat transfer rate.
- Due to the rising mass flow rate from 0.02 kg/s to 0.04 kg/s, the total heat transfer rate was increased by 72% and 61% for low and high temperature sources used in the experimental study, respectively. Considering the increase in percentage for the total heat transfer rate, the mass flow rate had more effects than the other rating parameters such as flow direction and hot water inlet temperature values.
- The difference in percentage values of the total heat transfer rate and total heat transfer coefficient obtained from experimental and numerical data was about 2.7% and 2.5%, respectively. According to the numerical simulation results, the temperature difference calculated for the hot side was close to the experimental results. Thus, the numerical results were in good agreement with the experimental data used in this study.

In further studies, we will plan to investigate the effects of a wide range of mass flow rate on the heat transfer rate of a brazed plate heat exchanger with using dimensionless parameters through a detailed CFD model.

NOMENCLATURE

A	total heat transfer area	[m^2]
b	mean channel gap	[m]
cp	specific heat at constant pressure	[J/kg K]
C	flow stream heat capacity rate	[W/K]
De	equivalent channel diameter	[m]
ε	heat exchanger effectiveness	-
Gch	channel mass flow rate	[kg/m s]
Lw	plate width inside gasket	[m]
$\dot{m}$	mass flow rate	[kg/s]
Ncp	Number of channel per pass	-
Q	total heat transfer rate	[W]
Re	Reynolds number	-
T	temperature	[°C]
u, v, w	velocity components	[m/s]
U	total heat transfer coefficient	[W/m K]
x, y, z	position coordinates	-
μ	dynamic viscosity	[Pa s]

Subscripts

c	cold water
ch	channel
cp	channel per passes
c,i	cold water inlet
c,o	cold water outlet
h	hot water
h,i	hot water inlet
h,o	hot water outlet
lm	logarithmic mean
max	maximum
min	minimum
w	wall
Δ	delta operator

REFERENCES

[1].M. M. A. Bhutta, N. Hayat, M. H. Bashir, A. R. Khan, K. N. Ahmad and S. Khan, “CFD applications in various heat exchangers design: A review”, Applied Thermal Engineering, vol. 32, no 1, September 2012.
[2].S. Kakaç, H. Liu, A. Pramuanjaroenkij, Heat exchangers: selection, rating, and thermal design, CRC Press: Florida, 2012.
[3].O. Ipek, M. Kan and B. Gurel, “Examination of Different Heat Exchangers and the Thermal Activities of Different Designs”, Acta Physica Polonica A, vol. 132, no 3, pp. 580-584, 2017.
[4].M. Kan, O. Ipek and B. Gurel, “Plate heat exchangers as a compact design and optimization of different channel

angles", Acta Physica Polonica A, vol. 128, no 2-B, pp. B-49-B52, 2015.
[5].K. Thulukkanam, Heat Exchanger Design Handbook, CRC Press, 2013.
[6].O. F. Genceli, Isı Değiştiricileri, Birsen Yayınevi, 2017.
[7].A. K. Tiwari, P. Ghosh, J. Sarkar, H. Dahiya and J. Parekh, "Numerical investigation of heat transfer and fluid flow in plate heat exchanger using nanofluids", International Journal of Thermal Sciences, vol. 85, pp. 93-103, June 2014.
[8].V. S. Gullapalli, Ph.D. Thesis, Estimation of Thermal and Hydraulic Characteristics of Compact Brazed Plate Heat Exchangers, Lund University, 2013.
[9].M. Fernández-Torrijos, J. A. Almendros-Ibáñez, C. Sobrino and D. Santana, "ε–NTU relationships in parallel–series arrangements: Application to plate and tubular heat exchangers", Applied Thermal Engineering, vol. 99, pp. 1119-1132, February 2016.
[10]. L. Wang, B. Sundén, R. M. Manglik, Plate Heat Exchangers, WIT Press, 2007.
[11]. H. Bayram and G. Sevilgen, "Numerical Investigation of the Effect of Variable Baffle Spacing on the Thermal Performance of a Shell and Tube Heat Exchanger," Energies, vol. 10, no. 12, p. 1156, August 2017.
[12]. R. L. Pradhan, D. Ravikumar, D. L. Pradhan, Review of Nusselt Number Correlation for Single Phase Fluid Flow through a Plate Heat Exchanger to Develop C# Code Application Software, 2nd National Conference on "Recent Developments in Mechanical Engineering, Pune, India, 2013.