Penerbit Akademia Baru

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences

Journal homepage: www.akademiabaru.com/arfmts.html
ISSN: 2289-7879

Effect of Number of Baffles on Flow and Pressure Drop in a Shell Side of a Shell and Tube Heat Exchangers

Open Access

A. Sadikin[1,*], N. Y. Khian[1], Y. P. Hwey[1], H. Y. Al-Mahdi[1], I. Taib[1], A. N. Sadikin[2], S. Md Salleh[1], S. S. Ayop[3]

[1] Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia, 86400 Parit Raja, Batu Pahat, Johor, Malaysia
[2] Faculty of Chemical and Energy Engineering, Universiti Teknologi Malaysia, 81310 Johor Bahru, Malaysia
[3] Faculty of Civil and Environmental Engineering, Universiti Tun Hussein Onn Malaysia, 86400 Parit Raja, Batu Pahat, Johor, Malaysia

ARTICLE INFO

Article history:
Received 17 March 2018
Received in revised form 21 July 2018
Accepted 14 August 2018
Available online 16 August 2018

ABSTRACT

The effect of number of baffles on flow and pressure drop on a shell side of shell-and-tube heat exchangers was studied. In the present study, a set of CFD simulations using FLUENT version 17.0 from ANSYS were used to analyze the flow in the single shell and single tube pass heat exchangers consists of 20 mm diameter of tubes in staggered configuration with a variable number of baffles. The simulations were undertaken to inform on how the fluid flowed within the shell side of shell-and-tube heat exchangers. The results show that the variable number of baffles and baffle spacing in a heat exchanger strongly affect the flow pattern and pressure drop. This is consistent with other published data.

Keywords:
Heat exchangers, shell-and-tube, baffle spacing, CFD

1. Introduction

A heat exchanger is a piece of equipment built for efficient heat transfer from one medium to another. It is a device that allows heat energy in one process fluid to pass to another process fluid in controlled manner. The two fluids or gas may be separated by a solid wall, so that they never mix, but they can be in direct contact. Therefore, heat exchanger is one of the most important devices in industrial process. They are widely used in space heating, refrigeration, air conditioning, power plants, petroleum refineries and natural gas processing. There are baffles placed in the shell and tube heat exchanger to enhance heat transfer.

Among the heat exchangers, shell-and-tube heat exchanger is regularly and generally utilized because of flexibility, robustness, and unwavering quality as reported by Markowski *et al.*, [1]. The heat exchangers are broadly utilized as a part of numerous industry fields such as petroleum, chemistry engineering, energy, power, and so on, for its wide choice of material, straightforward structure, solid and sturdy, less cost, and helpful upkeep. Concentrate on thermodynamic attributes,

* *Corresponding author.*
E-mail address: azmah@uthm.edu.my (Azmahani Sadikin)

structure advancement, and utilization of heat transfer improvement methods in exchangers has constantly gotten considerations.

Heat transfer improvement in heat exchangers has been seriously researched to enhance the condition and setup of heat exchange framework by many researchers [2-4]. Ozden *et al*., [3] studied shell and tube heat exchangers with segmentally baffles and stated that baffles are placed in the shell to force the shell-side fluid to flow across the shell to enhance heat transfer. Other than that, the baffles are used to maintain uniform spacing between the tubes and for directing the flow inside the shell from the inlet and outlet while maintaining effective circulation of the shell side fluid hence providing effective use of the heat transfer area. In line with this thought, Yang *et al*., [4] and Pal *et al.*, [5] investigates shell and tube heat exchangers with segmental baffles by whether varying baffle spacing, baffle cut, type of baffle, modelling approaches, and turbulence models while Vukić *et al*., [6] reported the effect of segmental baffles on the shell-and-tube heat exchanger effectiveness and shows that the 22% of baffle cut segmental baffles give an impact to heat transfer rate. The exceptional consideration was given to the examination of the segmental baffles number impact of the shell-and-tube heat exchanger adequacy. Vukić *et al*., [6] also states that the higher the segmental baffles number, the higher influence on the shell-and-tube heat exchanger efficiency to compare with the increase in the heating fluid flow rate. The shell and tube heat exchanger heat exchange performance is strongly depending on the shell side geometry parameters such as baffles number, baffle size, distance between baffles, the first and the last baffle position to inlet and outlet shell side, size of the constructive clearances. Maakoul *et al*., [7] numerically studied the performances of a combined multiple shell-passes shell and tube heat exchanger with continuous helical baffles in outer shell pass which improve the heat transfer. Movassag *et al*., [8] experimentally studied the performance of tube bundle replacement for segmental and helical baffles. The results indicated that it not only improved heat transfer of the helix bundle over segmental bundle, but also increases up to two-three times of operational running times.

Sparrow *et al*., [9] provides useful information on the effect of interbaffle spacing on heat transfer and pressure drop in a shell-and-tube heat exchanger. They reported the effect of baffle cut to the heat transfer performance. The baffle cut is measured as a percent of the baffle diameter. For single number of baffles are placed along the shell in alternating orientations where cut facing up, cut facing down, cut facing up again in order to create flow paths across the tube bundle, thus forming cross flow windows. The results showed that the most common baffle cut value involved in designed a heat exchanger is 25% which refer to 25% of cut value is selected to place the cut just below or above the central row of tubes. The inter-baffle spacing effect the response of the heat and mass transfer and pressure drop on the shell side of a shell and tube heat exchanger where the pressure drop is strongly affected by the baffle spacing. Meanwhile, Kistler *et al*., [10] used analytical methods and their method under predict the pressure drop, if the window flow area is considerably less than the cross flow. Contrary to Taborek [11] and Mukherjee [12] suggested that the optimum baffle and diameter ratio should between 0.3 and 0.6.

The effects of the baffle spacing arrangement and baffles orientation on the Nusselt number are presented by Mellal *et al.,* [13]. To perform tests, six geometrical configurations are realized: three cases of baffle spacing (106.6, 80 and 64 mm) corresponding respectively to the number of baffles: 6, 8 and 10 and three cases of baffle orientation angles (45°, 90° and 180°). It is found that the variation in Nusselt number is very sensitive to the baffles spacing parameters. The comparison between the performance of these three angles (45°, 90° and 180°) shows that the highest Nusselt number is obtained with the angle 180°. However, it observed that further reduction in Nusselt number with the baffle orientation angle of 45°, this reduction due to the great space between the baffles (45°), where the fluid flows easily through the shell causing a decrease in the turbulence intensity and as a consequence leads to a decrease in the heat transfer.

Mahmoud *et al.*, [14] investigated the effect of friction characteristics of the heat exchanger model when using different number of swirl vanes at different locations along the pipe length to enhance the heat transfer rate using CFD (Computational Fluid Dynamics) simulation for shell and tube heat exchanger. They reported the multi inserted swirl vanes with higher diameter and blade angle is better in heat transfer enhancement, friction factor, and thermal enhancement factor than using a single inserted swirl vane with high swirl vane diameter and low blade angle. Batalha Leoni *et al.*, [15] also performed CFD simulations of a small shell and tube heat exchanger with single segmental baffles and without baffle clearances and discussed the effects of baffle clearances on pressure drop and overall temperature variation. Those are reduced when baffle clearances increase because the leakage streams act in the sense of smoothing the effects of stagnation zones. Recirculation zones and eddies affects the effectiveness of a heat exchanger in a heat transfer process. The baffle spacing or number of baffles in a heat exchanger effecting both pressure drop and heat transfer characteristics. Therefore, the objective of this paper is to investigate the flow phenomenon in a shell and tube heat exchanger with different number of baffles and spacing to improve the heat exchanger design. The velocity, pressure drop and temperature distribution were predicted using CFD simulation.

2. Computational Fluid Dynamics (CFD)

2.1 The Governing Equations

The governing equations of the flow are modified according to the conditions of the simulated case. Since the problem is assumed to be steady, time dependent parameters are dropped from the equations. Below shown the governing equation for heat transfer for a heat exchanger;

Conservation of mass;

$$\nabla.\left(\rho\bar{V}\right)=0 \tag{1}$$

x-momentum;

$$\nabla.\left(\rho u\bar{V}\right)=-\frac{\partial p}{\partial x}+\frac{\partial\tau_{xx}}{\partial x}+\frac{\partial\tau_{yx}}{\partial y}+\frac{\partial\tau_{zx}}{\partial z} \tag{2}$$

y-momentum;

$$\nabla.\left(\rho v\bar{V}\right)=-\frac{\partial p}{\partial y}+\frac{\partial\tau_{xy}}{\partial x}+\frac{\partial\tau_{yy}}{\partial y}+\frac{\partial\tau_{zy}}{\partial z} + \rho g \tag{3}$$

z–momentum;

$$\nabla.\left(\rho w\bar{V}\right)=-\frac{\partial p}{\partial z}+\frac{\partial\tau_{xz}}{\partial x}+\frac{\partial\tau_{yz}}{\partial y}+\frac{\partial\tau_{zz}}{\partial z} \tag{4}$$

Energy;

$$\nabla.\left(\rho e\overline{V}\right)=-p\nabla.\overline{V}+\nabla.\left(k\nabla T\right)+q+\phi \tag{5}$$

In equation (5), is the dissipation function that can be calculated from

$$\phi=\mu\begin{bmatrix} 2\left[\left(\frac{\partial u}{\partial x}\right)^2+\left(\frac{\partial v}{\partial y}\right)^2+\left(\frac{\partial w}{\partial z}\right)^2\right]+\left(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}\right)^2+ \\ \left(\frac{\partial u}{\partial z}+\frac{\partial w}{\partial x}\right)^2+\left(\frac{\partial v}{\partial z}+\frac{\partial w}{\partial y}\right)^2 \end{bmatrix} \\ +\lambda\left(\nabla.\overline{V}\right)^2 \tag{6}$$

2.2 The Modeling Details

A set of CFD simulations is performed for a single shell and single tube pass heat exchanger with a variable number of baffles, namely 6, 8 and 10. The tubes are in a triangular arrangement. The details of the geometry of the shell and tube heat exchanger are tabulated in Table 1.

Table 1
The model geometry

Shell size, D_s (mm)	90		
Heat exchanger length, *L* (mm)	600		
Tube diameter, d_0 (mm)	20		
Inlet and outlet diameter of shell, *d* (mm)	30		
Baffle cut, B_c	36%		
Number of tubes, N_t	7		
Central baffle spacing, *B* (mm)	6 baffles	8 baffles	10 baffles
	86	62	48

The model was created in Solidwork as shown in Figure 1. The cold fluid flows in the shell and it is being heated by a hot fluid flows through the tube. The working fluid is water. The meshing gave a total of 620,000, nodes and had 2,650,000 elements that consisted of prisms. The shell and tubes were set to solid surfaces with no slip. Zero gauge pressure is assigned to the outlet nozzle in order to obtain relative pressure drop between inlet and outlet. The inlet velocity profile is assumed to be uniform. The zero heat flux boundary condition is set to the shell outer wall, assuming the shell is perfectly insulated at the outside wall. The tubes were modeled as solid cylinders with the constant wall temperature of 450 K were set to the tube wall. The inlet shell boundary condition was set to a mass flow rate of 0.5 kg/s. The temperature in the shell was set to 300 K. The k-epsilon realizable model was chosen for the turbulence model and the numerical accuracy was set to first order. The SIMPLE is used for the pressure correction method. The simulation was run until the residual of the pressure and velocities were less than 0.00001.

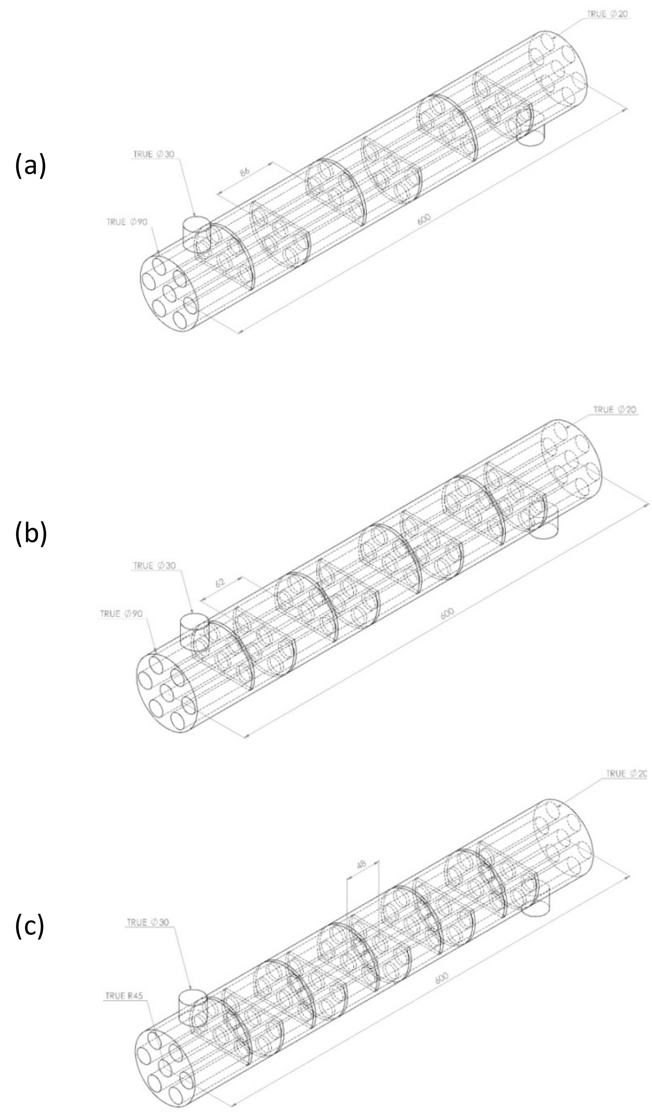

Fig. 1. The model geometry (a) 6 baffles (b) 8 baffles (c) 10 baffles

3. Results and Discussion
3.1 The Flow Pattern

The effect of baffle spacing on flow pattern investigated for three different numbers of baffles with 36% baffle cut. Figure 2 shows the velocity path line for 6, 8 and 10 baffles for the shell side mass flow rate of 5 kg/s. The fluid flow is higher and follows a more torturous path in 10 baffles configuration. As the fluid flows past the tubes, which was set to no slip at the wall, the fluid decelerates near the baffle surface and creates a thin layer, called the boundary layer, due to viscous effects. The flow is attached to the tube surface until the formation of a wake, evident to the rear of the baffle, where some of the fluid is flowing backward against the main flow. This is where the circulation happens. The flow re-attaches at the front of the baffles. It also shows that for the smaller number of baffles, the cross flow windows are not well utilized and some recirculation regions form behind the baffles. The recirculation zones provide low shear stress, creating suitable conditions for fouling growth on tubes external surfaces, which reducing heat transfer efficiency. Thus, increasing number of baffles has reduces the recirculation zones. The flow is also observed to be well developed and the cross flow paths are established throughout the shell volume. These observations are similar to the work of Ozden *et al.*, [3], Mellal *et al.*, [13] and Batalha Leoni *et al.*, [15].

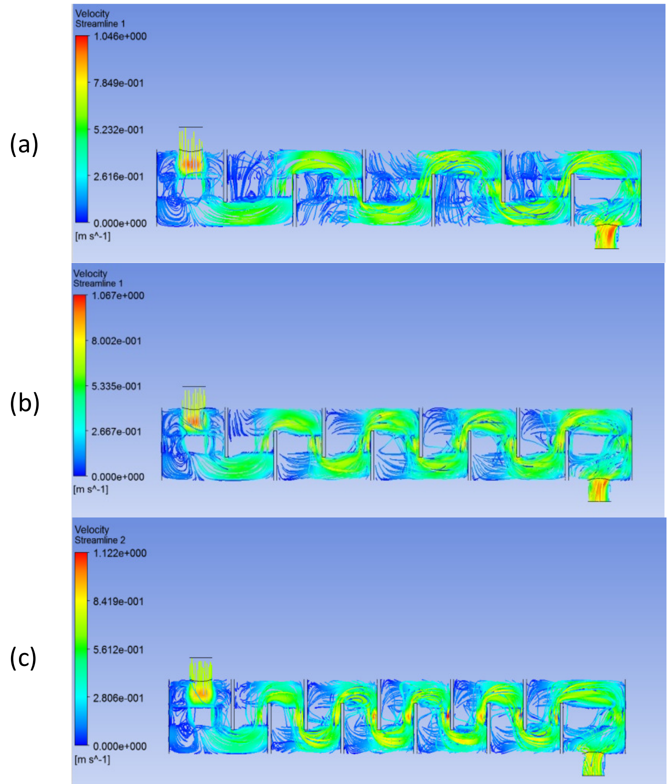

Fig. 2. The velocity path lines (a) 6 baffles (b) 8 baffles (c) 10 baffles

3.2 The Pressure Drop

The pressure drop is strongly affected by the baffle spacing. This finding is almost similar to those obtained in Ozden *et al.,* [3] and Batalha Leoni *et al.*, [15].The largest pressure occurred in 10 baffles configuration. The staggered alignment gives further reductions in pressure due acceleration and separation from the baffles. As a result, the friction pressure loss is higher in this arrangement. It's appears clearly that increase in number of baffle increases the generation of vortex because of the change of flow direction inside the tube heat exchangers. The decrease of the baffles spacing yields an increase in the friction factor. This is in agreement with the work of Mellal *et al.*, [13]. As expected, the pressure is shown to gradually decrease and increase as the flow moves around the baffles. The pressure drop in these tubes is caused by the reduction in flow area as the flow moves towards the baffles. Pressure recovery occurs as the flow separates from the tube just after the baffle and expands to reattach to the next baffle. There is a net pressure drop across each tube due to friction. The wake region at the rear of the baffle will cause a low pressure region due to turbulent dissipation. It also can be seen that the sudden pressure drop after each baffle in Figure 3 can be explained by the fact that when the fluid passes through a reduced area, such as the baffle window, in order to maintain its mass flow rate, its velocity increases, and consequently, the pressure drop. This is supported by the work of Batalha Leoni *et al.*, [15]. Overall, the loss of energy is in the direction of flow.

3.3 The Temperature Distribution

The 3D temperature profile along the heat exchanger is shown in Figure 4 CFD results show that for the shell side outlet temperature decrease by increasing the number of baffles. This is results of heat transfer is enhanced by vigorous mixing of the fluid due to turbulence induced by the baffles. This result agrees well by Ozden *et al*., [3]. It is also observed that the temperature in the shell side remains within the expected range, not exceeding significantly the average outlet temperature. It can also be seen that all profiles experience a rapid increase right after the baffles. It is possible to conclude that, as the number of baffles increase, overall heat transfer increases, implying in lower outlet temperatures for the cold fluid.

(a)

(b)

(c)

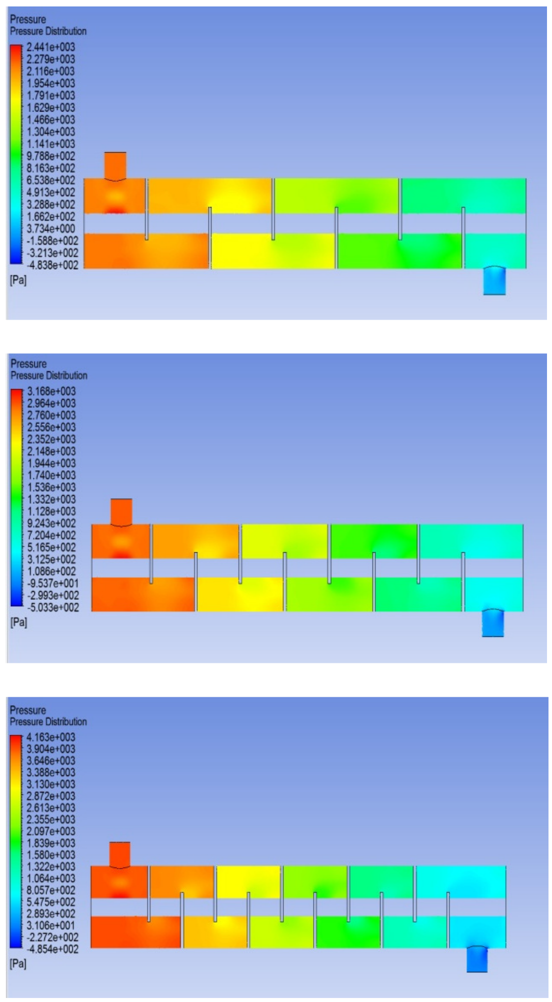

Fig. 3. The pressure contour (a) 6 baffles (b) 8 baffles (c) 10 baffles

(a)

(b)

(c)

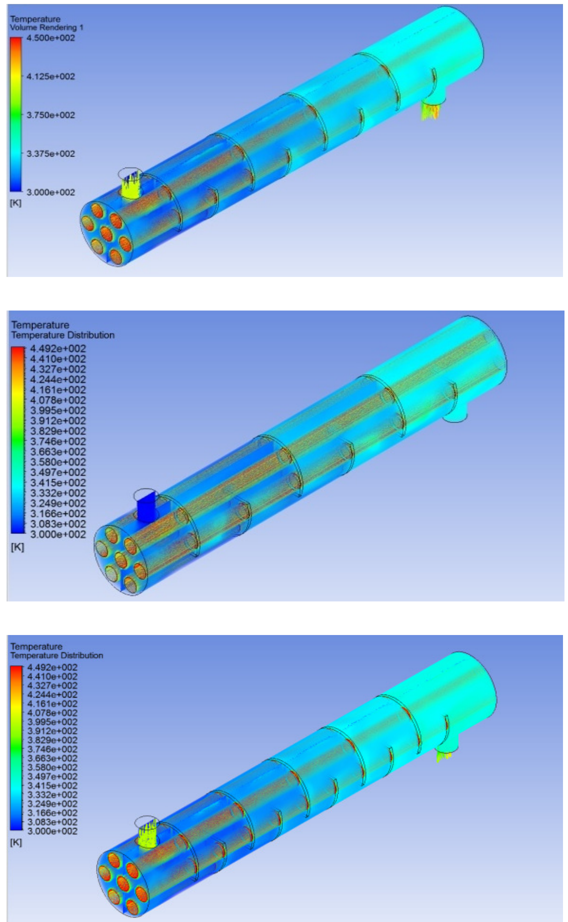

Fig. 4. The temperature distribution(a) 6 baffles (b) 8 baffles (c) 10 baffles

4. Conclusion

The shell side of a shell and tube heat exchanger is modeled with sufficient detail to resolve the flow and temperature fields. From the CFD simulation results, for fixed tube wall and shell inlet temperatures, the flow pattern, pressure drop and temperature outlet in the shell side of a shell-and-tube heat exchanger are obtained. As the number of baffle increases, the pressure drop and outlet temperature of the shell side increases. In addition, it can also be concluded that the number of baffles affect the space of recirculation zone in a shell side heat exchanger. As the number of baffles increases, the water also creates a significantly smaller re-circulation zone in the shell and the flow is well developed. Therefore, 10 numbers of baffles gives the best configuration in heat transfer properties. Hence, when come to simulation or design stages for a shell-and-tube heat exchanger, the number of baffles or baffles spacing are one of the consideration to obtain an effective heat exchanger. It can be concluded that the best configuration to transfer a large amount of heat is by increasing the number of baffles or baffle spacing. Therefore, the study of flow phenomenon and other mechanistic parameter for other working fluid and geometry is warranted.

Acknowledgement
The author gratefully acknowledges the financial support provided by Research and Innovation Fund of Research Management Center (RMC) Universiti Tun Hussein Onn Malaysia (UTHM).

References
[1] Markowski, Mariusz, Marian Trafczynski, and Krzysztof Urbaniec. "Identification of the influence of fouling on the heat recovery in a network of shell and tube heat exchangers." *Applied energy* 102 (2013): 755-764.
[2] Gu, Xin, Bing Liu, Yongqing Wang, and Ke Wang. "Heat transfer and flow resistance performance of shutter baffle heat exchanger with triangle tube layout in shell side." *Advances in Mechanical Engineering* 8, no. 3 (2016): 1687814016641015.
[3] Ozden, Ender, and Ilker Tari. "Shell side CFD analysis of a small shell-and-tube heat exchanger." *Energy Conversion and Management* 51, no. 5 (2010): 1004-1014.
[4] Yang, Jie, Lei Ma, Jessica Bock, Anthony M. Jacobi, and Wei Liu. "A comparison of four numerical modeling approaches for enhanced shell-and-tube heat exchangers with experimental validation." *Applied Thermal Engineering* 65, no. 1-2 (2014): 369-383.
[5] Pal, Eshita, Inder Kumar, Jyeshtharaj B. Joshi, and N. K. Maheshwari. "CFD simulations of shell-side flow in a shell-and-tube type heat exchanger with and without baffles." *Chemical Engineering Science* 143 (2016): 314-340.
[6] Vukić, Mića V., Mladen A. Tomić, Predrag M. Živković, and Gradimir S. Ilić. "Effect of segmental baffles on the shell-and-tube heat exchanger effectiveness." *Hemijska industrija* 68, no. 2 (2014): 171-177.
[7] El Maakoul, Anas, Azzedine Laknizi, Said Saadeddine, Mustapha El Metoui, Abdelkabir Zaite, Mohamed Meziane, and Abdelatif Ben Abdellah. "Numerical comparison of shell-side performance for shell and tube heat exchangers with trefoil-hole, helical and segmental baffles." *Applied Thermal Engineering* 109 (2016): 175-185.
[8] Movassag, Sirous Zeyninejad, Farhad Nemati Taher, Kazem Razmi, and Reza Tasouji Azar. "Tube bundle replacement for segmental and helical shell and tube heat exchangers: Performance comparison and fouling investigation on the shell side." *Applied Thermal Engineering* 51, no. 1-2 (2013): 1162-1169.
[9] Sparrow, E. M., and L. G. Reifschneider. "Effect of interbaffle spacing on heat transfer and pressure drop in a shell-and-tube heat exchanger." *International journal of heat and mass transfer* 29, no. 11 (1986): 1617-1628.
[10] Kistler, R. S., and J. M. Chenoweth. "Heat exchanger shellside pressure drop: comparison of predictions with experimental data." *Journal of heat transfer* 110, no. 1 (1988): 68-76.
[11] Taborek, J. *Heat exchangers design handbook.* (Vol. 3). New York: Begell House Inc. 2002.
[12] Mukherjee, Rajiv. "Effectively design shell-and-tube heat exchangers." *Chemical Engineering Progress* 94, no. 2 (1998): 21-37.
[13] Mellal, Mustapha, Redouane Benzeguir, Djamel Sahel, and Houari Ameur. "Hydro-thermal shell-side performance evaluation of a shell and tube heat exchanger under different baffle arrangement and orientation." *International Journal of Thermal Sciences* 121 (2017): 138-149.
[14] Yehia, Mahmoud Galal, Ahmed AA Attia, Osama Ezzat Abdelatif, and Essam E. Khalil. "Heat transfer and friction characteristics of shell and tube heat exchanger with multi inserted swirl vanes." *Applied Thermal Engineering* 102 (2016): 1481-1491.
[15] Leoni, Gabriel Batalha, Tânia Suaiden Klein, and Ricardo de Andrade Medronho. "Assessment with computational fluid dynamics of the effects of baffle clearances on the shell side flow in a shell and tube heat exchanger." *Applied Thermal Engineering* 112 (2017): 497-506.