# OPTIMIZATION AND CFD ANALYSIS OF A SHELL-AND-TUBE HEAT EXCHANGER WITH A MULTI SEGMENTAL BAFFLE

by

***Ahmet AYDIN*** [a], ***Halit YASAR*** [b], ***Tahsin ENGIN*** [b*], ***and Ekrem BUYUKKAYA*** [b]

[a] Metallurgical and Materials Engineering Department, Engineering Faculty, Sakarya University, Sakarya, Turkey
[b] Mechanical Engineering Department, Engineering Faculty, Sakarya University, Sakarya, Turkey

Original scientific paper
https://doi.org/10.2298/TSCI200111293A

*The shell-and-tube type heat exchangers have long been widely used in many fields of industry. These types of heat exchangers are generally easy to design, manufacturing, and maintenance, but require relatively large spaces to install. Therefore, the optimization of such heat exchangers from thermal and economical points of view is of particular interest. In this article, an optimization procedure based on the minimum total cost (initial investment plus operational costs) has been applied. Then the flow analysis of the optimized heat exchanger has been carried out to reveal possible flow field and temperature distribution inside the equipment using CFD. The experimental results were compared with CFD analyses results. It has been concluded that the baffles play an important role in the development of the shell side flow field. This prompted us to investigate new baffle geometries without compromising from the overall thermal performance. It has been found that the heat exchanger with the new baffle design gives rise to considerably lower pressure drops in the shell side, which in turn reducing operating cost. The new baffle design is particularly well suited for shell-and-tube heat exchangers, where a viscous fluid-flows through shell side with/out phase change.*

Key words: *shell-and-tube heat exchanger, optimization, multi segmental baffle, CFD analysis, optimal design*

## Introduction

Heat exchangers are devices used for transferring thermal energy between a solid object and a fluid, or between two or more fluids. The fluids may be separated by a solid wall to prevent mixing or they may be in direct contact. They are widely used in space heating, refrigeration, air conditioning, power stations, petrochemical, chemical and pharmaceutical industries, natural gas processing and wastewater treatment [1-4]. Among these, shell-and-tube heat exchangers are the most commonly used ones. In this system, heat transfer performance depends on many parameters such as placement of tubes, number of baffles, number of tubes and length. These heat exchangers have a lot of advantageous, such as having a high ratio of volume and heat transfer area, easier cleaning, manufacturing and repairing, and to be able to transfer high mass-flow rates. However, it is possible to improve the performance of a heat exchanger by

[*] Corresponding author, e-mail: engint@sakarya.edu.tr

changing baffle geometry. Since the flow direction may be guided with these component, the whole heat transfer area is involved in the heat transfer and the velocities and the turbulence may be higher due to the decreased flow section. Thus, this improved value may provide higher heat transfer coefficient and heat performance. However, one of the major constraints that stands in the way of optimizing its thermal design is the pressure drop.

Pressure drop is an important constraint in thermal design of shell-and-tube heat exchangers. Thermal design of a shell-and-tube heat exchanger is meaningful solely when it is optimum and the value of this is constrained by the pressure drop. As a result, optimization of thermal design requires maximization of overall heat transfer coefficient and effective mean temperature difference so as to minimize the heat transfer area subject to the constraints, pressure drop being the major one. The pressure drop should be managed in such a way that the calculated pressure drop is within and as close as possible to the allowable pressure drop. That is, when the pressure drop has a limiting effect during thermal design, the calculated pressure value should be reduced so that it does not exceed the permissible pressure drop. Moreover, drop in the determined pressure value should be as close as possible to the permissible pressure drop when the pressure drop during the thermal design is high [5].

Accurate determination of acceptable pressure drops in a heat exchanger design is possible by repeating several experiments many times. However, the fact that heat exchangers with a wide range of applications can be operated under the most economical conditions depends primarily on the fact that the pressure drops are objectively determined [2]. This pressure drop for both fluids sets the initial investment cost of the heat exchanger as well as the cost of energy and the initial investment cost of the pump or compressor required to heat the fluids. However, in many applications, the pressure drop values given for the heat exchanger design are usually not determined objectively.

Various studies have been carried out for the optimization of shell-and-tube type heat exchangers. Two different methods were used in these studies. One of them is Kern [6] and the other one is the Bell-Delaware method [7]. Kern method gives conservative results, suitable for the preliminary sizing. On the other hand, Bell-Delaware method is a detailed accurate in estimating heat transfer coefficient and the pressure drop on the shell side for common geometric arrangements. Bell-Delaware method indicates the existence of possible weaknesses in the shell side design, but does not point out where these weaknesses are.

Investigations were carried out taking into account the pressure drops in the heat exchanger. The first of these is McAdams [8]. This researcher derived two expressions that give optimum heat flux for the unit heat energy. In heat exchanger cost optimization, some of the researchers used Lagrange multipliers and geometric programming techniques. In order to apply these methods, algebraic expressions are needed which express the boundary functions and the objective functions correctly. Babu and Shaik [9] performed optimal design of shell-and-tube type heat exchangers using ten different strategies in differential evolution method. Leoni *et al.* [10] investigated the effect of the usage of different turbulence model while Ambekar *et al.* [11] studied four different segmental baffle types, such as single, double, triple and flower. Irshad *et al.* [12] done comparison for several shell-and-tube heat exchangers with segmental baffles. Their simulation studies shown how the temperature, pressure, velocity varies in shell due to different baffles orientation. Markosvska *et al.* [4] made the optimal design of trunk tube heat exchangers by providing simultaneous solutions of equations using a software package. Ravagani *et al.* [13] solved an optimization problem with a shell-and-tube heat exchanger design, the objective function cost being the least, by using the formulation and the particle swarm optimization method. Abd and Naji [14] examined the method of Kern to define the external heat transfer coefficient. Bhandurge *et al.* [15] done investigation along with CFD simulation on single pass, counter flow shell-and-tube heat exchanger at 0°, 15°, 30°, 45°, and orientation. They examined the heat transfer rate and pressure drop of shell side fluid with Bell-Delaware method. Edwards [16] evaluated the fundamental aspects of the thermal design of trunk tube heat exchangers. Ponce *et al.* [17] solved a compact formulation of the Bell-Delaware method proposed for optimal shell-and-tube heat exchanger design using genetic algorithm. Varga *et al.* [18] studied helical baffles for the more favorable flow regulation. Azad and Amidpour [19] used the new approach of structural theory to make the optimal design of shell-and-tube type heat exchangers economical. Shrikant *et al.* [20] replaced a segmental tube bundles by a bundle of tubes with helical baffles in a shell-and-tube heat exchanger to reduce pressure drop and fouling and hence reduce maintenance and operating cost in Tabriz Petroleum Company. Using the genetic algorithm, Sanaye and Hajabdollahi [21] solved objective function optimization using the genetic algorithm, with the shell-and-tubular heat exchangers being the most efficient and least expensive. Jegede and Polley [22] go for a very useful and simple method innovation for heat exchanger optimization. Engin and Gungor [23] have applied this method to different types of heat exchangers on the shell-and-tube type heat exchangers.

In this study, a new type of baffle called multi segmental baffle was proposed for use in shell-and-tube heat exchangers. Then, this heat exchanger was optimized using the method proposed by Jegede and Polley [22]. In the study, the results of CFD analysis of the heat exchanger with multi segmental baffle were compared to the heat exchanger with conventional baffles. The heat exchanger produced according to the optimization results was tested and the results were compared with the CFD analysis results for the same heat exchanger.

## Optimization methodology

In this study, the optimization method developed by Jegede and Polley [22] was adopted. The heat transfer rate of a heat exchanger, that is, the amount of heat transmitting from the hot fluid to the cold fluid is expressed:

$$\dot{Q}_\mathrm{h} = \left(\dot{m}C_p\right)_\mathrm{h} \left[\Delta T\right]_\mathrm{h}, \; \dot{Q}_\mathrm{c} = \left(\dot{m}C_p\right)_\mathrm{c} \left[\Delta T\right]_\mathrm{c}, \; \dot{Q}_\mathrm{h} = \dot{Q}_\mathrm{c} \tag{1}$$

where $\dot{m}$ is mass-flow rate, $C_p$ – the specific heat of the fluid, $\Delta T$ – the temperature difference of the fluid, and the subscripts c and h refer to cold and hot fluids, respectively. The following equation is used to express the heat transfer rate based on logarithmic temperature difference on the shell-and-tube sides:

$$\dot{Q} = KA\Delta T_m \tag{2}$$

where $\Delta T_m$ is the logarithmic temperature difference, fig. 1. By neglecting the wall thickness of tubes, as well as fouling effects, total heat transfer coefficient can be expressed:

$$K = \frac{1}{\dfrac{1}{h_\mathrm{s}} + \dfrac{1}{h_\mathrm{t}}} \tag{3}$$

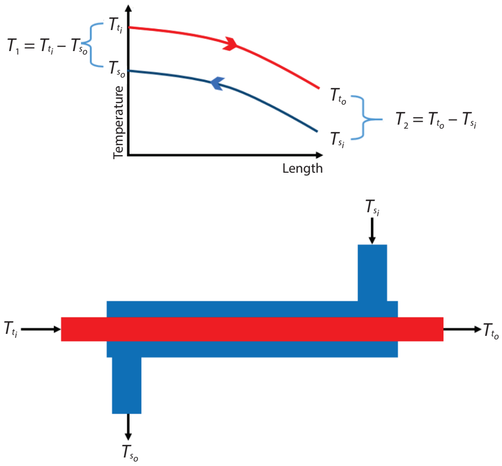


**Figure 1. Calculation of logaritmic temperature difference**

$$\Delta T_m = \frac{\Delta T_1 - \Delta T_2}{\log\left(\dfrac{\Delta T_1}{\Delta T_2}\right)} \tag{4}$$

Shell-and-tube side pressure drops were given by Jegede and Polley [22]:

$$\Delta P_\mathrm{t} = C_\mathrm{t} A h_\mathrm{t}^{3.5} \tag{5}$$

$$\Delta P_\mathrm{s} = C_\mathrm{s} A h_\mathrm{s}^{5.1} \tag{6}$$

where $C_\mathrm{t}$ and $C_\mathrm{s}$ are the constants depending on geometric properties of the heat exchanger as well as thermophysical properties of the fluids. The cost components of a heat exchanger system to be taken as basis for optimization are the initial cost and the operating cost:

$$C_\mathrm{t} = C_\mathrm{he} + C_\mathrm{op} \tag{7}$$

where $C_\mathrm{he}$ is the initial investment cost of the heat exchanger and $C_\mathrm{op}$ is the cost of energy consumption of the system. The initial investment cost of the heat exchanger is expressed by the following equation [23].

$$C_\mathrm{he} = C_1 + C_2\left(\frac{1}{h_\mathrm{s}} + \frac{1}{h_\mathrm{t}}\right) \tag{8}$$

The operation cost is the energy consumption cost required to overcome the pressure drop of the pump and was given by [23]:

$$C_\mathrm{op} = C_3\left(\frac{h_\mathrm{s}^{5.1}}{h_\mathrm{t}} + h_\mathrm{t}^{4.1} + C_4 h_\mathrm{s}^{5.1}\right) + C_5\left(\frac{h_\mathrm{t}^{3.5}}{h_\mathrm{s}} + h_\mathrm{t}^{2.5} + C_4 h_\mathrm{t}^{3.5}\right) \tag{9}$$

The total cost function is consequently given as a function of shell-and-tube sides convective heat transfer coefficients:

$$C_t = C_1 + C_2\left(\frac{1}{h_\mathrm{s}} + \frac{1}{h_\mathrm{t}}\right) + C_3\left(\frac{h_\mathrm{s}^{5.1}}{h_\mathrm{t}} + h_\mathrm{t}^{4.1} + C_4\, h_\mathrm{s}^{5.1}\right) + C_5\left(\frac{h_\mathrm{t}^{3.5}}{h_\mathrm{s}} + h_\mathrm{t}^{2.5} + C_4\, h_\mathrm{t}^{3.5}\right) \tag{10}$$

Minimizing the aforementioned equation will also optimize the cost. Thus, the following equations are obtained when the eq. (10) are derived according to $h_\mathrm{s}$ and $h_\mathrm{t}$ and equalized to zero:

$$f_1\left(h_\mathrm{s}, h_\mathrm{t}\right) = \frac{\partial C_\mathrm{t}}{\partial h_\mathrm{s}} = 0 \tag{11}$$

$$f_2\left(h_\mathrm{s}, h_\mathrm{t}\right) = \frac{\partial C_\mathrm{t}}{\partial h_\mathrm{t}} = 0 \tag{12}$$

The roots of the equation were calculated as $h_\mathrm{s}$ and $h_\mathrm{t}$. For the solution of two non-linear equations with two unknowns, the program created on MATLAB was used. After optimized $h_\mathrm{s}$ and $h_\mathrm{t}$ were evaluated. Geometric parameters were calculated as shown in tab. 1.

**Table 1. Parameters of the heat exchanger used in CFD analysis**

| Tube-side | | Shell-Side | |
|---|---|---|---|
| $h$ = 6817 W/m$^2$K | | $h$ = 3240 W/m$^2$K | |
| Velocity | 0.8 m/s | Velocity | 0.38 m/s |
| Tube number | 37 | Shell diameter | 0.161 m |
| Surface area | 2.8 m$^2$ | Distance between baffles | 0.193 m |
| Length | 1.4 m | Baffle number | 6 |
| Pressure drop | 736 Pa | Pressure drop | 736 Pa |

## Design parameters and CFD model

For optimization of the heat exchanger, the mass-flow rates of water through the inner tubes and the shell surface were 3.3 kg/s and 2.51 kg/s, respectively, which was given in tab. 2. For shell side, the inlet water temperature is 10 °C. The outlet water temperature is 30 °C. For tube-side, the inlet water temperature is 130 °C. The outlet water temperature is 115 °C In order to clearly see the characteristic differences, the analysis was carried out for the case where the temperature difference was highest for the shell side and the tube side. In the optimization studies, the economic life of the heat exchanger, the total working time, the pump efficiency, the total fouling resistance, the energy unit cost and the annual real interest rate were taken as 15 years, 8000 hours, 70%, 0.00036 kg/ms, 0.070 $/kWh, and 7%, respectively.

**Table 2. Fluid properties**

| Parameter | Water (Tube side) ($T_{mean}$ = 122.5 °C) | Water (Shell side) ($T_{mean}$ = 20 °C) |
|---|---|---|
| Mass-flow rate [kgs$^{-1}$] | 3.30 | 2.51 |
| Density [kgm$^{-3}$] | 941.25 | 998 |
| Specific heat [kJkg$^{-1}$K$^{-1}$] | 4.249 | 4.182 |
| Kinematic viscosity [kgm$^{-1}$s$^{-1}$] | 0.683 | 0.598 |
| Prandtl number | 1.3025 | 7.01 |

As a result of optimization of the heat exchanger, the input geometric parameters given in tab. 3 were obtained and these data were used in CFD analysis.

**Table 3. Geometric parameters of the heat exchanger used in CFD analysis**

| Tube-side | | Shell-Side | |
|---|---|---|---|
| Tube number | 37 | Shell diameter | 0.161 m |
| Length | 1.4 m | Baffle number | 6 |

Based on the optimization results obtained first in the CFD analysis developed with the ANSYS FLUENT program, the flow geometry is modeled with the separate design modeler for the conventional and multi segmental baffle shell-and-tube model. In these models, two separate control volumes are modeled to examine the shell side and tube side flows. For simplicity of solution, symmetry of the model showing symmetry feature was taken and the number of solution networks was reduced by half. The heat exchanger models with multi segmental and conventional baffles are shown in fig. 2.

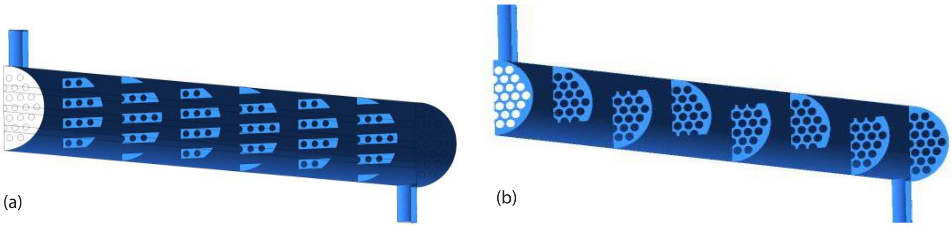


**Figure 2. Multi segmental and conventional baffle shell-and-tube heat exchanger models; (a) multi segmental baffle and (b) conventional baffle**

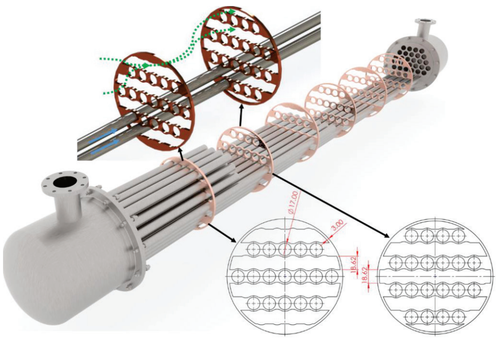

**Figure 3. Details of the multi segmental baffle [mm]**

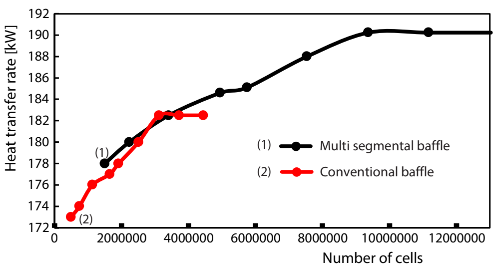


**Figure 4. Element number independency of the numerical solution**

The most important advantage of multi segmental baffle is creating local turbulence zone. Thus, the dead zones are eliminated for the shell side by using the multi segmental baffle. The details of multi segmental baffle are shown in fig. 3.

During analysis, 3530171 elements for conventional baffle shell-and-tube and 10096426 elements for multi segmental shell and baffle tube models were used in the solution network where tetrahedral elements were used. Figure 4 shows the element number independency of the numerical solution based on heat capacity. As can be seen from the figure, when the number of elements is increased over 10 million for multi segmental baffle and 3 million for conventional baffle there is almost no change in the heat capacity obtained from the analysis. The $k$-$\varepsilon$ turbulence model was used in the simulation studies. Mass-flow rate and pressure were defined as the inlet and outlet conditions, respectively. In order to model the tube surface fouling resistance, thermal conductivity was taken as 3.36 W/mK at the interface. The simulations were performed on a DELL T5600 Workstation (Intel® Xeon®, 3.30 GHz, 2 processors, 16 cores, 128 GB RAM). The solution time is observed to be approximately 2 hours for each a solution.

## Experimental set-up and procedure

The experimental studies were carried out on the heat exchanger which has the geometric dimensions obtained from the optimization studies. The tube side flow was supported by a frequency converter pump with a closed loop. The hot water tank was heated by electrical heaters in order to keep the temperature constant. On the other hand, the cold water was controlled with the frequency converter pump and the heated water was evacuated out in a tank. Flow and temperature control were done at the heat exchanger inlet and outlet points. The experimental set-up is shown in fig. 5. During the experiments, flow rate and temperature control were done with the control panel and necessary controls were provided.

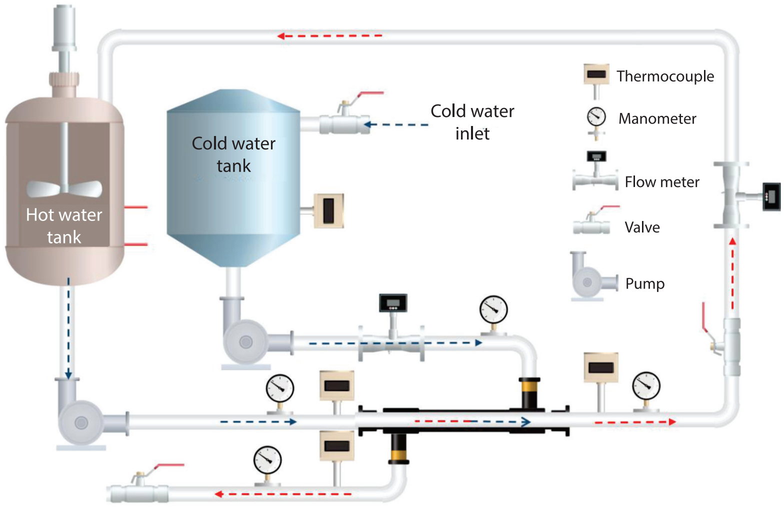


**Figure 5. Experimental set-up**

The basic elements used in the experimental set-up are hot and cold water tanks, shell--and-tube heat exchanger and control panel. The system also includes water valves, manometers to measure the pressure differences of the fluids entering and exiting the heat exchanger, and PT 100 thermocouples for measuring the temperature of the hot and cold fluids. Before getting the experimental data, the valve in the tube from which the water came from was opened and the system was expected to be filled completely. Then the cold water outlet valve was opened and the control panel provided hot and cold pumping at the desired flow rate. After a certain period of time, the system became stable and the necessary measurement results were taken.

Experiments were performed for four different input conditions. When the heater capacities were 15 kW, the analyses were performed for low flow rates to ensure the stability of the temperature, tab. 4. In this study, experimental uncertainties were calculated by Turchi *et al*. [24] method. Table 5 shows the measurement ranges, measurement accuracy of the measuring devices used in the experimental set-up, and uncertainty levels of the calculated parameters based on experimental data.

**Table 4. Heat exchanger input parameters**

| Tube-side | | Shell-side | |
|---|---|---|---|
| Mass-flow rate [kgs$^{-1}$] | Inlet temperature [K] | Mass-flow rate [kgs$^{-1}$] | Inlet temperature [K] |
| 0.3 | 323 | 0.4 | 295 |
| 0.7 | 323 | 0.7 | 295 |
| 1.1 | 323 | 1.0 | 295 |
| 1.5 | 323 | 1.4 | 295 |
| 2.1 | 323 | 1.9 | 295 |

## Results and discussion

A preliminary analysis was conducted for design parameters. In order to clearly see the characteristic differences, the analysis was carried out for the case where the temperature difference was highest for the shell side and the tube side. For the heat exchanger, the mass-

**Table 5. Measuring ranges and measurement accuracy of the devices used in the experiments and uncertainty levels of the calculated parameters**

| Equipment | Range | Accuracy | Uncertainty [%] |
|---|---|---|---|
| PT 100 | 0/100 °C | ±1 °C | |
| Manometer | 0/100 mbar<br>0/2.5 bar | 2 mbar<br>0.02 bar | |
| Flow meter | 0/50 lt/s | 0.01 lt/s | |
| Heat transfer rate | – | – | 1.5 |

flow rates of water through the inner tubes and the shell surface were 3.3 kg/s and 2.51 kg/s, respectively. For shell side, the inlet water temperature is 10 °C. The outlet water temperature is 30 °C. For tube-side, the inlet water temperature is 130 °C. The outlet water temperature is 115 °C.

Figure 6 shows streamlines in heat exchangers with conventional and multi segmental baffles. As can be seen from the figure, in the heat exchanger having conventional baffles, re-circulation zones are formed at the rear of the baffles. In the case of multi segmental baffle, these re-circulation zones are almost never formed. These re-circulation zones reduce the heat transfer from the hot fluid to the cold fluid on the one hand, while increasing the fouling resistance in these areas. Increased fouling resistance reduces the service life of the heat exchanger, increases the operating and maintenance costs of the heat exchanger. In addition, the heat transfer in these re-circulation zones decreases depending on time and the efficiency of the heat exchanger is reduced.

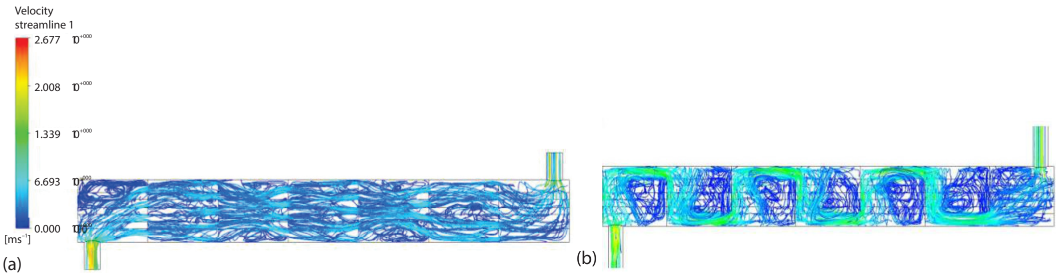


**Figure 6. Streamlines in the heat exchangers with conventional and multi segmental baffles; (a) multi segmental baffle and (b) conventional baffle**

Figure 7 shows the temperature distributions on tube surfaces in heat exchangers having conventional and multi segmental baffles. As can be seen from the figures, when the curtain multi segmental baffles are used, a much more uniform temperature distribution is obtained on the tube surfaces compared to the conventional situation. This indicates that the heat transfer efficiency of the conventional heat exchanger is lower than multi segmental baffle type heat exchanger.

Figure 8. shows velocity vectors in heat exchangers having conventional and multi segmental baffles. It can be clearly seen from figure that, with the use of the multi segmental baffle, the local turbulence regions occur in the heat exchanger and the velocity distribution is much more homogenous than the conventional heat exchanger. On the other hand, re-circulation zones are formed in the back of the baffles in the conventional heat exchanger. In certain local areas of these regions, the speed becomes zero, *i.e.*, the flow becomes stationary. This

reduces the heat transfer in these dead zones, resulting in reduced heat exchanger efficiency and increased fouling resistance.

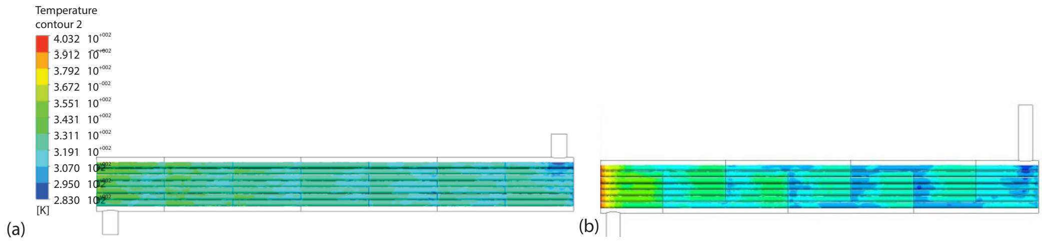


**Figure 7. Temperature distribution on the tube surface of the heat exchangers with conventional and multi segmental baffles; (a) conventional baffle and (b) multi segmental baffle**

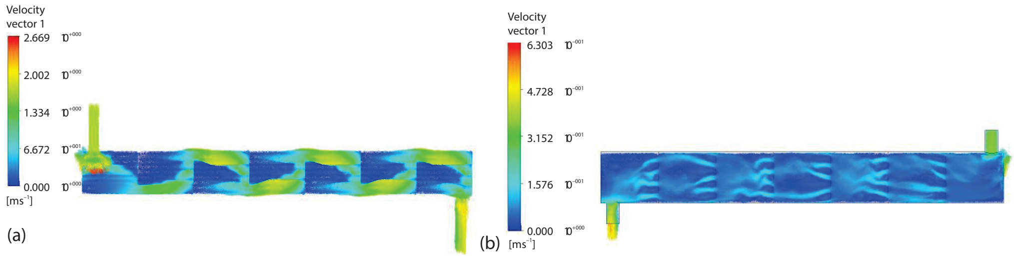


**Figure 8. Speed vectors formed in the heat exchangers with conventional and multi segmental baffles; (a) conventional baffle and (b) multi segmental baffle**

Figure 9. shows the pressure variation in the heat exchanger along the length of the shell. As can be seen from the figure, a uniform pressure distribution across the shell is seen in the heat exchanger having multi segmental baffles. However, in the heat exchanger with conventional baffles, sharp pressure drops occur due to increased pressure drop between the heat exchanger inlet and the outlet.

Figure 10. shows the temperature variations occurring between the inlet and outlet of the tube surfaces in the heat exchangers. Considering that the temperature change on the tube surface affects the thermal efficiency, the heat exchanger with multi segmental baffles is evident from the fact that there is a much more uniform heat transfer than the conventional heat exchanger.

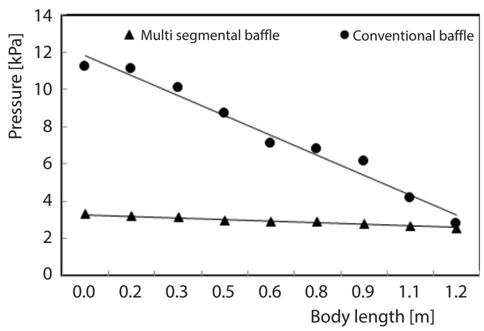


**Figure 9. Pressure variation across the heat exchanger body**

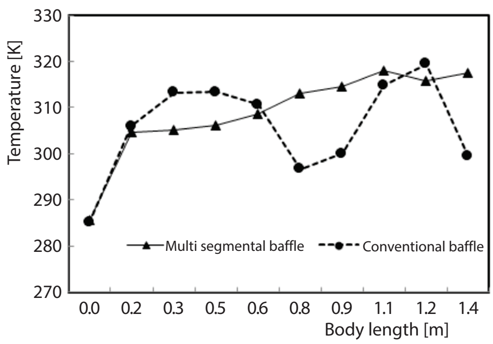


**Figure 10. Temperature variation on the tube surfaces across the inlet and outlet**

*Comparison of the experimental and simulation results*

The heat transfer rate calculated by experimental data and the heat transfer rate obtained by CFD analysis in the heat exchanger with multi segmental baffles are compared in fig. 11. As can be seen from the figure, in both cases the heat transfer rate increases as the mass-flow rate increases. A difference up to 9% was occurred between the experimental and the CFD results.

Figure 12 compares the experimental and CFD pressure drops occurring on the tube side of the heat exchanger with multi segmental baffles. As can be seen from the figure, the pressure drops in the tubes increase with the increase of the mass-flow rate as expected. The experimental pressure drops were determined between 0.1-2.0 kPa, and the pressure drop determined with CFD analysis were between 0.09-1.9 kPa. There was a difference up to 8% between the experiment and CFD analysis.

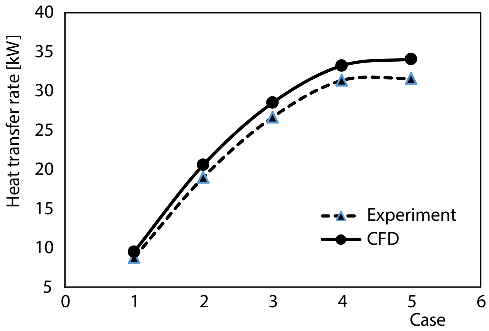


**Figure 11. Heat transfer rate of the heat exchanger for different working conditions**

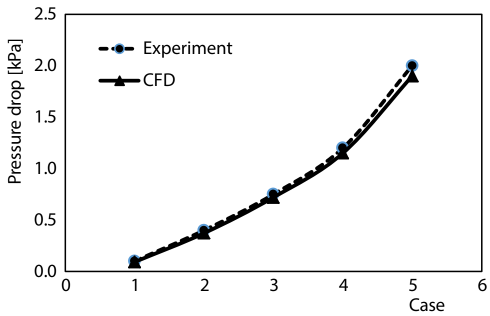


**Figure 12. Tube side pressure drop for different working conditions**

Table 6 compares CFD analysis results of the optimized multi segmental baffle heat exchanger with the conventional heat exchanger. The results indicate that the new design heat exchanger with multi segmental baffle gives much better results compared to the conventional heat exchanger in terms of both higher heat transfer rate and lower pressure drop. In the case of heat exchanger with multi segmental baffle, there was about 7% increase in heat transfer rate compared to the conventional heat exchanger. In the multi segmental baffle heat exchanger, a significant reduction in pressure drop was achieved compared to the conventional heat exchanger. The CFD analysis results show that the 12 kPa pressure drop in the conventional heat exchanger was reduced to 4.03 kPa in the multi segmental baffle heat exchanger. Thus, with the use of the multi segmental baffle, the operational cost was reduced 66.42% compared to the conventional baffle.

**Table 6. Performance values of the multi segmental and conventional baffle heat exchangers obtained with CFD analyses**

| | Multi segmental baffle | Conventional baffle |
|---|---|---|
| Heat transfer rate [kW] | 190 | 182 |
| Tube outlet temperature [K] | 389 | 390 |
| Shell side temperature [K] | 300 | 299 |
| Shell side pressure drop [kPa] | 4.03 | 12 |
| Tube side pressure drop [kPa] | 720 | 732 |
| Heat transfer rate/pressure drop (shell side) [$kWk^{-1}Pa^{-1}$] | 47 | 15 |

## Conclusions

In this study, a new heat exchanger with multi segmental baffles has been designed and optimized by using the method developed by Jegede and Polley. Then, the CFD analyzes of the new design and conventional heat exchangers were conducted and compared. In addition, the heat exchanger produced according to the parameters obtained as a result of the optimization was tested under certain operating conditions and the results were compared with the results obtained by CFD analysis. As a result of the study, the following conclusions were obtained.

- In the case of heat exchanger with multi segmental baffle, there was about 7% increase in heat transfer rate compared to the conventional heat exchanger for the same heat transfer surface area.
- In the multi segmental baffle heat exchanger, a significant reduction in pressure drop was achieved compared to the conventional heat exchanger. The CFD analysis results showed that the 12 kPa pressure drop in the conventional heat exchanger was reduced to 4.03 kPa in the multi segmental baffle heat exchanger.
- In the heat exchanger having conventional baffles, re-circulation zones are formed at the rear of the baffles. In the case of multi segmental baffle, these re-circulation zones are almost never formed. These re-circulation zones reduce the heat transfer from the hot fluid to the cold fluid on the one hand, while increasing the fouling resistance in these areas. Increased fouling resistance reduces the service life of the heat exchanger, increases the operating and maintenance costs of the heat exchanger. The operating costs of the multi segmental baffle heat exchanger were reduced by 197% compared to conventional heat exchanger.

## Nomenclature

$A$ – heat transfer area, [m$^2$]
$C_{\mathrm{he}}$ – cost of heat exchanger, [$ per year]
$C_{\mathrm{op}}$ – operational cost, [$ per year]
$C_p$ – specific heat of the fluid, [kJkg$^{-1}$K$^{-1}$]
$C_{\mathrm{t}}$ – total cost, [$ per year ]
$h$ – heat transfer coefficient, [Wm$^{-2}$K$^{-1}$]
$K$ – total heat transfer coefficient, [Wm$^{-2}$K$^{-1}$]
$\dot{m}$ – mass-flow rate, [kgs$^{-1}$]
$\dot{Q}$ – heat transfer rate, [kW]
$\Delta P$ – pressure drop, [kPa]
$\Delta T$ – temperature difference, [K]
$\Delta T_m$ – logarithmic temperature difference, [K]

*Subscript*

c – cold
h – hot
s – shell
t – tube
i – inlet
o – outlet

## References


[1] Sharma, S., Dewangan, R. K., A Review on Shell and Tube Heat Exchanger (Sthx) Using Various Orientation Angle of Baffle, *IJESRT International Journal of Engineering Sciences and Research Technology*, *6* (2017), 10, pp. 336-369
[2] Petrik, M., Szepesi, G. L., Shell Side CFD Analysis of a Model Shell-and-Tube Heat Exchanger, *Chemical Engineering Transactions*, *70* (2018), pp. 313-318
[3] Master, B. I., *et al*., Most Frequently Used Heat Exchangers from Pioneering Research to Worldwide Applications, *Heat Transfer Engineering*, *27* (2006), 6, pp. 4-11
[4] Markovska, L., *et al*., Optimum Design of Shell-and-Tube Heat Exchanger, *Bulletin of the Chemists and Technologits of Macedonia*, *15* (1996), Suppl. 1, pp. S39-S44
[5] Singh, S. K., Stephan, D., https://www.process-worldwide.com/, 2014
[6] Kern, D. Q., *Process Heat Transfer*, McGraw-Hill, New York, USA, 1950
[7] Bell, K. J., Delaware Method for Shell Side Design, in: *Heat Exchangers: Thermal-Hydraulic Fundamentals and Design*, (Eds. Kakac, S., Bergles, A. E., Mayinger, F.), Hemisphere, New York, USA, 1981, pp. 581-618
[8] McAdams, W. H., *Heat Transmission*, McGraw-Hill, New York, USA, 1954
[9] Babu, B. V., Shaik, M. A., Differential Evolution Strategies for Optimal Design of Shell-and-Tube Heat Exchangers, *Chemical Engineering Science*, *62* (2007), 14, pp. 3720-3739

[10] Leoni, G. B., *et al*., 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), Feb., pp. 497-506
[11] Ambekar, A. S., *et al*., The CFD Simulation Study of Shell and Tube Heat Exchangers with Different Baffle Segment Configurations, *Applied Thermal Engineering*, *108* (2016), Sept., pp. 999-1007
[12] Irshad, M., *et al*., Design and CFD Analysis of Shell and Tube Heat Exchanger, *International Journal of Engineering Science and Computing*, *7* (2017), 4, pp. 6453-6457
[13] Ravanagi, M. A. S. S., *et al*., Optimal Design of Shell-and-Tube Heat Exchangers Using Particle Swarm Optimization, *Industrial and Engineering Chemıstry Research*, *48* (2009), 6, pp. 2927-2935
[14] Abd, A. A., Naji, S. Z., Analysis Study of Shell and Tube Heat Exchanger for Clough Company with Reselect Different Parameters to Improve the Design, *Case Studies in Thermal Engineering*, *10* (2017), Sept., pp. 455-467
[15] Bhandurge, S. R., *et al*., Analysis and Experimentation of Shell and Tube Heat Exchanger with Different Orientation of Baffles, *International Journal for Research in Applied Science and Engineering Technology*, *4* (2016), 6, pp. 490-503
[16] Edwards, J. E., *Design and Rating Shell and Tube Heat Exchanger*, Teesside University, Teesside, UK, 2008
[17] Ponce, J. M., *et al*., *Optimal Design of Shell-and-Tube Heat Exchangers Using Genetic Algorithms*, 16$^{th}$ European Symposium on Computer Aided Process Engineering and 9$^{th}$ International Symposium on Process Systems Engineering, Elsevier, Amsterdam, The Netherlands, 2006, Vol. 21, pp. 985-990
[18] Varga, T., *et al*., Horizontal Scraped Surface Heat Exchanger – Experimental Measurements and Numerical Analysis, *Pollack Periodica*, *12* (2017), 1, pp. 107-122
[19] Azad, A. V., Amidpour, M., Economic Optimization of Shell and Tube Heat Exchanger Based on Constructal Theory, *Energy*, *36* (2011), 2, pp. 1087-1096
[20] Shrikant, A. A., *et al*., Comparison of Shell and Tube Heat Exchanger using Theoretical Methods, HTRI, ASPEN and SOLIDWORKS Simulation Softwares, *Int. Journal of Engineering Research and Application*, *6* (2016), Mar., pp. 99-107
[21] Sanaye, S., Hajabdollahi, H., Multi-Objective Optimization of Shell and Tube Heat Exchangers, *Applied Thermal Engineering*, *30* (2010), 14-15, pp. 1937-1945
[22] Jegede, F. O., Polley, G. T., Optimum Heat-Exchanger Design, *Chemical Engineerıng Research & Design*, *70* (1992), Nov., pp. 133-141
[23] Engin, T., Gungor, K. E., Design and Cost of Shell-Tube Heat Exchangers Optimization by Parameters (in Turkish), *TÜBİTAK-Türk Mühendislik ve Çevre Bilimleri Dergisi*, *20* (1996), pp. 313-322
[24] Turchi, A., *et al*., Thermochemical Ablation Modelling Forward Uncertainty Analysis – Part II: Application Plasma Wind-Tunnel Testing, *International Journal of Thermal Sciences*, *118* (2017), Aug., pp. 510-517

Paper submitted: February 22, 2020
Paper revised: September 7, 2020
Paper accepted: September 16, 2020