Overall Heat Transfer Analysis of Shell and Tube Heat Exchanger Using CFD

Abhay Raj, Ashish Nayyar, Arun Beniwal

Department of Mechanical Engineering, Swami Keshvanand Institute of Technology, Management & Gramothan, Jaipur-302017 (INDIA)

Email : **abhayraj43@gmail.com, ashish.nayyar@skit.ac.in, arun.beniwa@skit@ac.in**

Received 10.04.2023 **received in revised form** 09.09.2023, **accepted** 19.09.2023

DOI: 10.47904/IJSKIT.13.2.2023.54-59

***Abstract*-Analysis of overall Heat transfer in Shell and tube heat exchanger can be done by changing various structural parameters, such as baffle cut, space between baffles, shape of baffles or tube shape and size. In this report, all the analysis is done on the shell side. Analysis of shell side fluid temperature and heat transfer is done by using a commercial CFD package. Study is done on a single pass shell and tube heat exchanger with turbulent flow. Simulation data are obtained for shell and tube heat exchanger with helical tube and inclined baffles (25% and 36% cut) with inclination angles of 5º, 10º and 15º respectively. In this report, simulation is done using CFD (Ansys Fluent) for 5°, 10° and 15° inclination of baffle and it is observed that, for 10° baffle inclination, highest thermal performance is observed as compared to the 5°, and 15°.**

Keywords: **Heat exchanger, Shell and tube, Heat transfer coefficient, CFD**

1. INTRODUCTION

Heat exchanger is used to transfer enthalpy. The device needs no outside work for its operation. With its wide range of applications in the field of pasteurization, sterilization, fractionate distillation, etc., heat exchangers find their place in various domestic and industrial locations. However, selection of heat exchanger for a particular utilization requires proper calculation and investigation. They are widely used in automobiles, space air conditioning, process and chemical industries. Mechanical equipment like refrigerators and air conditioners also use heat exchangers to release latent heat of working fluid.

The application of heat exchangers can also be found in pharmaceutical industries, marine, food and beverage industries, polymer and power production industries.

1.1 Shell and tube heat exchanger

Among all the categories of heat exchangers, the shell and tube heat exchangers have gained a lot of attention from the industry due to its firm quality. In shell and tube heat exchangers, two liquid streams of different temperatures are made to flow in close contact without physical mixing. Heat from hot fluid tends to flow into the cold fluid due to temperature difference. The heat transfer occurs through the effective heat transfer area. In order to increase the effective heat transfer area, a significant number of tubes are provided inside a shell. The arrangement has proven to be an effective energy-saving tool.

Shell and tube heat exchanger is the most widely used heat exchanger, due to its simple construction, wide applicability, resistance to high pressure and temperature, easy to maintain and low cost. Shell and tube heat exchanger with segmental baffle is the most common one. The baffles keep the tube bundles intact and compels the fluid flowing through shell to experience a complex flow path. The size of the baffle can be varied as per requirement and its dimension is defined by the term “percent baffle cut”. A half-cut baffle means that the baffle size is half of the cross section of the shell. Another important aspect in heat exchanger design is baffle pitch. The distance between two successive baffles is known as baffle pitch. The working and the schematic diagram of shell and tube heat exchanger is shown in Figure 1.

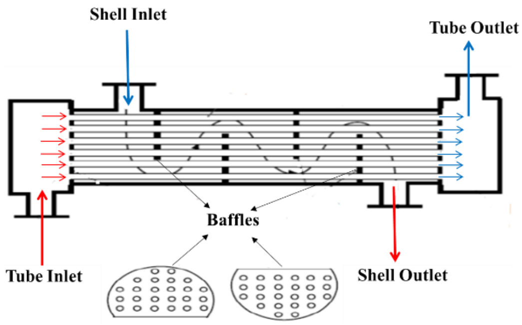

Figure 1: Schematic diagram of shell and tube heat exchanger

2. NUMERICAL METHODOLOGY

The performance of a heat exchanger is greatly influenced by the tube as well as the shell design. Heat transfer inside a straight tube can be enhanced by decreasing the tube diameter or increasing the fluid flow velocity. However, both the solutions have its limitations. Introducing flow turbulators or modifying the flow path can enhance the flow turbulence inside the tubes. Moreover, modification in flow path also enhances the effective heat transfer area. Similarly, heat transfer in the shell side can be enhanced by introducing baffles of optimized cut, pitch and angle. In the present investigation, effort is made to enhance the heat transfer rate on both sides simultaneously by introducing helical shaped tubes and baffles with three different inclination angles.

The overall investigation is carried out using computational fluid dynamics (CFD) tool. CFD is the science used to investigate the flow dynamics, heat and mass flow along with related processes by solving the governing equations using numerical processes. In the last decade, CFD utilization has shown an uprising trend as it allows investigation at minimal cost. CFD investigation is comparable to an ideal experimental study with a high level of accuracy. A wide variety of geometrical and operational parameters can be considered to achieve large number of performance parameters without the installation of measuring equipment. Many simulation tools have found their place in research and industrial organization. A CFD tool divides the overall area of study into many control volumes. For each control volume, the governing equations (partial differential equations) are converted into algebraic equations. The algebraic equations for each control volume are solved numerically to achieve all the properties of the fluid. CFD tools can yield an overall picture of flow over the computational domain. At present, the application of CFD tool can be seen in marine engineering, biomedical science, aircraft design, ship design, chemical engineering, building simulations, oceanography etc. One of the most reliable CFD tool is ANSYS FLUENT.

ANSYS FLUENT 18.0 is used for simulation work. Water is considered to be the working fluid both inside the tube and shell. Thermal and hydraulic behavior of the setup is investigated to study the effect of considered geometrical modifications.

2.1 Computational Domain

Many researchers have worked on the geometry of baffle, but a limited study on the modification in tube flow path could be found in the literature. It was also found to be true that changing the geometry of tubes produces lesser effect as compared to the geometry of baffle. In the present study, effort has been made to enhance the overall performance by providing modification in both the baffles as well as tubes. Two fixed baffle cut values of 25% and 36% are considered with an inclination angle. ANSYS modeler is used to design 3D model shown in Figure 2. The tubes are placed inside a shell of 100 mm diameter in a triangular tube bundle geometry with 30 mm center to center distance between the tubes. A total of seven tubes are modelled with constant diameter of 20 mm.

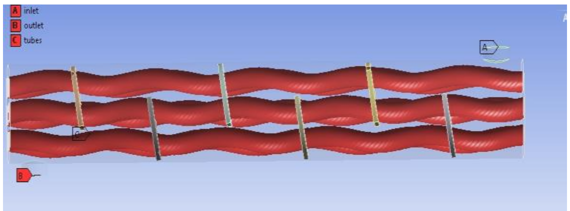

Figure 2: Heat exchanger with helical tubes and inclined baffles

The considered geometry is expected to provide more turbulence in the flow as compared to straight segmental baffle with straight tubes. Tubes are made helical with a helical pitch of 120 mm as shown in Figure 3. Baffles are placed with some inclination in order to reduce pressure drop. Different inclination angles are considered to investigate the effect of baffle inclination angle. The considered geometrical parameters in the investigation are summarized in Table 1.

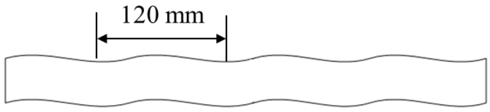

Figure 3: Schematic diagram of tube

Table1: Design and operational parameters

S. No.	Design parameters	Value
1	Shell diameter, D	100 mm
2	Tube diameter, d	20 mm
3	Tube bundle geometry and pitch	Triangular, 30 mm
4	Number of tubes, N_t	7
5	Heat exchanger length, L	600 mm
6	Shell side inlet temperature, T_{si}	300 K
7	Baffle cut, B_c	36%
8	Central baffle spacing	86 mm
9	Number of baffles	6
10	Helical pitch of tube	120 mm
11	Baffle inclination	5^o, 10^o, 15^o
12	Mass flow rate, $\dot{m}$	0.5, 1, 2 kg/s

2.2 Assumption*s and boundary conditions*

In order to simplify calculations, following assumptions are considered for every set of simulation:

1. No inlet disturbance
2. Steady state condition
3. Thermo-physical properties of shell water are considered to be constant and is calculated at inlet temperature of 300 K. The numerical value of the properties is summarized in Table 2.
4. The tube temperature is constant throughout the heat exchanger.
5. Fouling factors are neglected.

6. No leakage

Table 2: Thermo-physical properties of water

S. No	Property	Value
1	Density, ρ	997 kg/m^3
2	Specific heat, c_p	4183 J/kg K
3	Thermal conductivity, k_w	0.5948 W/m K
4	Dynamic viscosity, μ	0.0008905 Pa-s
5	Prandtl number, Pr	6.263

In order to solve the complex governing equations, following boundary conditions are defined:

1. No-slip condition at the stationary walls.
2. Assuming the shell is perfectly insulated from outside, zero heat flux in the outer shell wall, inlet and outlet nozzles is assigned.
3. 300 K is shell inlet condition. The inlet velocity profile is assumed to be uniform. Turbulence intensity and turbulence length scale are measured using Eq. 1 and 2 respectively.

$$I = 0.16Re^{-1/8} \tag{1}$$

$$l = 0.07D_h \tag{2}$$

Shell side flow is studied in this work.450 K temperature is assigned to tube wall.

3. CFD METHODOLOGY

ANSYS FLUENT is based on finite volume method and is generally used to analyze the flow and heat transfer characteristics within the system. The present study is carried out in 18.0 version of FLUENT, installed in an intel 3 core processor with 3.60 Hz clock speed. SIMPLE algorithm is employed to couple the pressure velocity terms whereas pressure-based solver is used to solve the equations. Convective terms are discretized using second order upwind scheme. Momentum and energy equations are discretized using a central difference scheme. Convergence criteria of 10^{-4} are assigned for k and ε terms whereas 10^{-5} and 10^{-8} are assigned for velocity and energy respectively.

3.1 Mesh selection and grid independence test

Grid independence test is done for the domain using proximity and curvature method with various mesh elements and with change in number of mesh elements, variation in results can be observed. Due to the complex geometry of tubes, proximity and curvature methods are used, resulting to higher number of mesh elements as compared to other methods. The simulated values of overall heat transfer coefficient on the tube wall for different number of mesh elements are obtained and summarized in Table 3. It was observed that when the number of elements is increased from 4789153 to 4911211, the variation in the result of heat transfer coefficient is found to be 0.07%. Hence, that number of elements is taken to be approximately 4800000 with maximum face size of 2 mm and growth rate of 1.150 for further investigation

Table 3: Grid independence test

S. No.	Number of elements	Heat transfer coefficient (W/m^2 K)	% change in heat transfer coefficient
1	3536527	2595	-
2	4452138	2618	0.8
3	4789153	2632	0.5
4	4911211	2634	0.07

4. RESULT AND DISCUSSION

Simulation data are obtained for shell and tube heat exchanger with helical tube and inclined baffles (25% and 36% cut) with inclination angles of 5º, 10º and 15º respectively. An increase in baffle angle is expected to decrease the number of streamlines of transversal component of velocity and increase in resultant cycloidal velocity, i.e., zig-zag pattern of flow is converted into cycloidal pattern. More heat transfer is expected to occur in cycloidal pattern, because of higher turbulence.

4.1 Thermal and hydraulic performance

Simulation is done using commercial package of CFD (fluent) and readings are obtained for heat exchanger with helical tube and baffle with inclination of 5º, 10º and 15º respectively having two different standards of baffle cut, i.e., 25% and 36%.

Table 4 and 5 show the results for 5º, 10º and 15º baffle inclination angle having baffle cuts of 25% and 36% respectively. Comparing the data shown in the table can lead us to the conclusion that the increase in baffle cut percentage results in a decrease in thermal performance. For a particular mass flow rate, heat transfer coefficient decreases with increase in baffle cut as the resistance to flow decreases. With increase in flow area, the fluid is allowed to flow without much resistance from the baffle resulting in decrease in flow turbulence. This leads to a decrease in heat transfer coefficient.

Moreover, increase in baffle inclination angle leads to decrease in number of streamlines of transversal component of velocity and increase in resultant cycloidal velocity. The zigzag pattern of flow is converted into cycloidal flow. More heat transfer is observed in cycloidal pattern as it covers more tube area. Moreover, turbulence is more for cycloidal pattern. The pressure drop is also observed to decrease due to inclination of baffle.

It is observed that heat transfer is more for 10-degree baffle inclination as compared to 5 and 15 degrees. It is due to the reason that, inclination of baffles converts the zigzag pattern into cycloidal pattern, but for 10º baffle inclination angle, a perfect s-shape flow is formed.

4.2 Validation

In the present study, validation of the numerical method is done with Ender *et al*. For the validation purpose, Flow inside the shell is observed by a small heat exchanger. A shell and tube heat exchanger are selected with seven number of tubes, six number of baffles and baffle cut of 36%. The considered geometrical and operational parameters considered for validation are shown in Table 6 and the computational domain is shown in Figure 4.

Table 4: Design and operational parameters considered for validation

S. No.	Design parameters	Value
1	Shell diameter, D	90 mm
2	Tube diameter, d	20 mm
3	Pitch and tube bundle geometry	30 mm triangular
4	Number of tubes, N_t	7
5	Length of Heat Exchanger, *L*	600 mm
6	Inlet temperature, Shell side T_{si}	300 K
7	Baffle cut, B_c	36%
8	Central baffle spacing	86 mm
9	Baffle's number	6
10	Mass flow rate, $\dot{m}$	0.5, 1, 2 kg/s

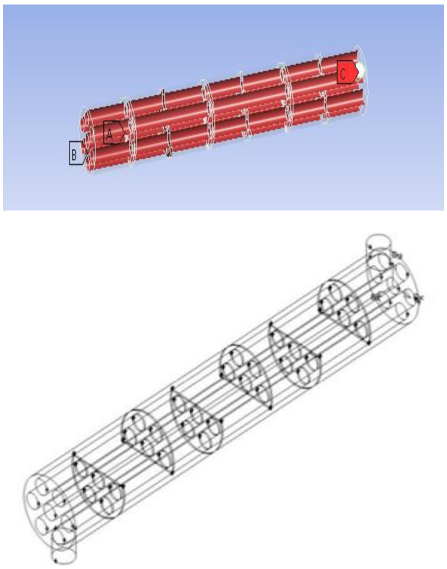

Figure 4: Computational domain of shell and tube heat exchanger used for validation

In the shell side, water flows in at 300 K and tube fluid is fixed at a constant temperature of 450 K. The tube fluid temperature is kept constant to ease the simulation as keeping the tube fluid properties constant makes it a single-phase flow problem instead of two-phase flow problem.

Following are the result of simulation obtained for different baffle angle and different baffle cut.

Table 5: Results for helical shape tube with 25% baffle cut and different baffle inclination angles

$\dot{m}$ (kg/s)	5^o inclination				10^o inclination				15^o inclination			
	T_{so} (K)	*h* (W/m² K)	*Q*(W)	Δ*P* (Pa)	T_{so} (K)	*h* (W/m² K)	*Q* (W)	Δ*P* (Pa)	T_{so} (K)	*h* (W/m² K)	*Q* (W)	Δ*P* (Pa)
0.5	342.8	2632	85763	979	342.9	2640	85769	857.6	342.6	2625	85750	840
1	333.5	3967	133560	5200	334.6	4353	135526	3407.3	332.3	3945	133452	3280
2	328.7	6944	242604	21463	329.3	7248	245793	13598.8	327.7	6920	240778	13281

Table 6: Results for helical shape tube with 36% baffle cut and different baffle inclination angles

$\dot{m}$ (kg/s)	5^o inclination				10^o inclination				15^o inclination			
	T_{so} (K)	*h* (W/m² K)	*Q* (W)	Δ*P* (Pa)	T_{so} (K)	*h* (W/m² K)	*Q* (W)	Δ*P* (Pa)	T_{so} (K)	*h* (W/m² K)	*Q* (W)	Δ*P* (Pa)
0.5	335.6	2581	74457	823	336	2592	75294	721.5	335.1	2577	73411	825
1	321.2	3911	88679	4923	322.1	4112	92444	3389.6	319.2	3895	80313	3160
2	314.8	6872	123816	20112	316.3	7123	136366	12224.7	314	6845	117124	12485

4.2 Graph and contour

Various graphs are obtained between presuure v/s baffle angle, temperature v/s baffle angle. Heat transfer rate v/s baffle angle, heat transfer coefficient v/s baffle angle

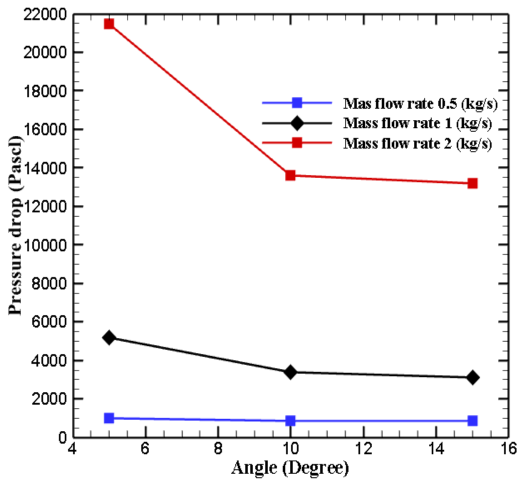

Figure 5: Pressure drop (Pascal) v/s baffle angle (Degree)

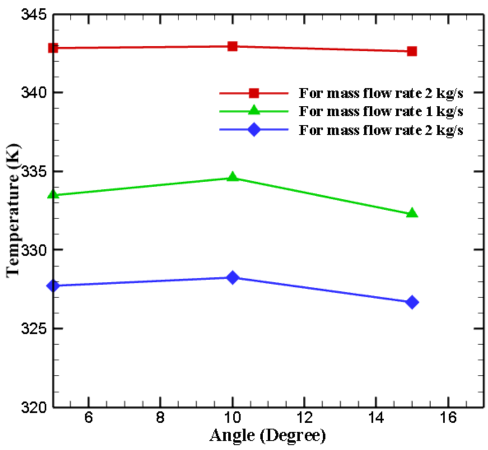

Figure 6: Temperature (K) variation v/s baffle angle (Degree)

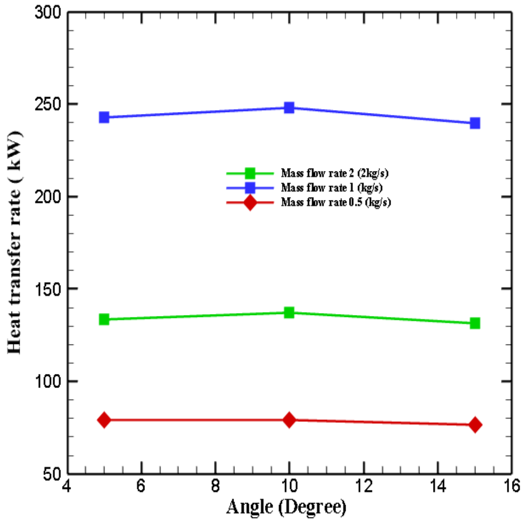

Figure 7: Heat transfer coefficient v/s baffle angle (degree)

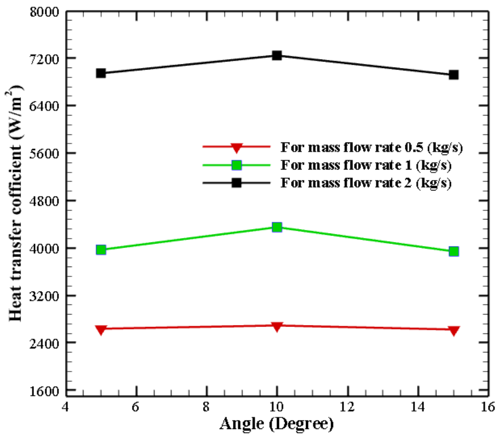

Figure 8: Heat transfer coefficient v/s baffle angle (Degree)

From the above graph it can be seen that thermal performances are higher by 10 degrees as compared to 5 and 15 degree and reasons behind this can be concluded from the contour of velocity path line which are shown below.

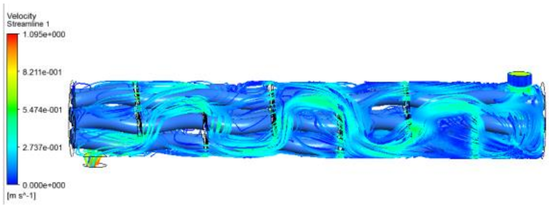

Figure 9: Contour of path line for 0.5 mass flow rate (kg/s) and 5-degree angle of inclination.

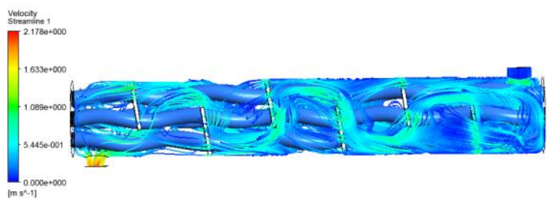

Figure 10: Contour of path line for 0.5 mass flow rate (kg/s) and 10-degree angle of inclination.

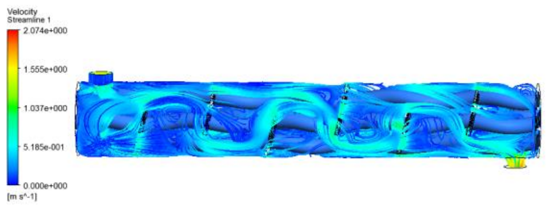

Figure 11: Contour of path line for 0.5 mass flow rate (kg/s) and 15-degree angle of inclination.

From the contour of pathline it can be seen that for 5 degree angle of inclination pathlines are following zigzag path and for 10 degree zigzag path is changed into cycloidal path and when the pathlines flows into cycloidal path contact surface area increases compared to zigzag path, but when angle is increased beyond 10 degree covering area of tubes by the streamlines is decreased and less area is covered by the pathlines. Hence, thermal performence is decreased and percentage decrease is more as compared to 5 degree.

5. CONCLUSION AND FUTURE SCOPE

Increase in mass flow rate causes decrease in outlet temperature. This is because a greater amount of water comes in contact of tubes.

Increasing the baffle angle will result in a decrease in pressure drop. This is because of the easy passage available for the fluid in the shell side. For 5-degree angle of inclination of baffle and mass flow rate of 0.5 kg/s, there is about 34% decrease in pressure drop as compared to the straight baffle.

A more heat transfer rate is observed at 10 degrees as compared to 5 and 15 degrees because, a perfect sinusoidal shape is observed. But in 5-degree, distortion of zigzag pattern is observed, and this distortion is converted at perfect sinusoidal shape at 10 degree. With further increase in baffle inclination, more distortion of sinusoidal shape is observed due to which covering area of tubes by streamline of fluid decreases and hence heat transfer rate decreases.

There is about 8% increase in the area of tube compared to the straight tube, which also results in higher thermal performance of heat exchanger.

As compared to straight tube, outlet temperature is higher for helical tubes (For shell side fluid). For every mass flow rate there is about 1 to 2 degrees increase in temperature.

5 .1 Future scope

In order to increase the intermixing between the fluid layers some holes can be done in the baffles. This might increase the turbulence in the flow and as a result thermal performance will increase.

REFERENCES

[1] R. Mukherjee, Use double-segmental baffles in the shell-and-tube heat exchangers, Chem. Eng. Prog. 88 (1992)

[2] M. Saffar-Avval, E. Damangir, A general correlation for determining optimum baffle spacing for all types of shell and tube exchangers, Int. J. Heat Mass.Transfer 38 (1995)

[3] H.D. Li, V. Kottke, Effect of baffle spacing on pressure drop and local heat transfer in shell-and-tube heat exchangers for staggered tube arrangement, Int. J. Heat Mass. Transfer 41 (1998)

[4] P. Sthlik, V.V. Wadekar, Different strategies to improve industrial heat exchange, Heat Transfer Eng. 23 (2002)

[5] K.J. Bell, Heat exchanger design for the process industries, ASME J. Heat Transfer 126 (2004)

[6] B.K. Soltan, M. Saffar-Avval, E. Damangir, Minimization of capital and operating costs of shell and tube condensers using optimum baffle spacing, Appl. Therm. Eng. 24 (2004)

[7] J. Lutcha, J. Nemcansky, Performance improvement of tubular heat exchangers by helical baffles, Trans. ChemE 68 (1990)

[8] P. Stehlik, J. Nemcansky, D. Kral, Comparison of correction factors for shelland- tube heat exchangers with segmental or helical baffles, Heat Transfer Eng. 15 (1994)

[9] D. Kral, P. Stelik, H.J. Van Der Ploeg, I.M. Bashir, Helical baffles in shell-andtube heat exchangers, part one: experimental verification, Heat Transfer Eng. 17 (1996)

[10] Zhang, Jian-Fei, Shao-Long Guo, Zhong-Zhen Li, Jin-Ping Wang, Ya-Ling He, and Wen-Quan Tao. "Experimental performance comparison of shell-and-tube oil coolers with overlapped helical baffles and segmental baffles." Applied thermal engineering 58, no. 1-2 (2013): 336-343.

[11] Q.W. Wang, G.N. Xie, B.T. Peng, L.Q. Luo, Experimental study and genetic algorithm-based correlation on shell-side heat transfer and flow performance of three different types of shell-and-tube heat exchangers, ASME J. Heat Transfer 129 (9) (2007) 1277-1285.

[12] B.T. Peng, Q.W. Wang, C. Zhang, G.N. Xie, L.Q. Luo, Q.Y. Chen, M. Zeng, An experimental study of shell-and-tube heat exchangers with continuous helical baffles, ASME J. Heat Transfer 129 (10) (2007) 1425-1431.

[13] Q.W. Wang, Q.Y. Chen, G.D. Chen, Numerical investigation on combined multiple shell-pass shell-and-tube heat exchangers with continuous helical baffles, Int. J. Heat Mass. Transfer 52 (2009) 1214-1222.

[14] Q.W. Wang, G.D. Chen, J. Xu, Y.P. Ji, Second-law thermodynamic comparison and maximal velocity ratio design of shell-and-tube heat exchangers with continuous helical baffles, ASME J. Heat Transfer 132 (10) (2010)

[15] Q.W. Wang, G.D. Chen, Q.Y. Chen, M. Zeng, Review of improvements on shell and- tube heat exchangers with helical baffles, Heat Transfer Eng. 31 (2010)

[16] Q.W. Wang, G.D. Chen, M. Zeng, Q.Y. Chen, B.T. Peng, D.J. Zhang, L.Q. Luo, Shell-side heat transfer enhancement for shell-and-tube heat exchangers by helical baffles, Chem. Eng. Trans. 21 (2010) 217-222.

[17] G.D. Chen, M. Zeng, Q.W. Wang, S.Z. Qi, Numerical studies on combined parallel multiple shell-pass shell-and-tube heat exchangers with continuous helical baffles, Chem. Eng. Trans. 21 (2010) 229-234.