JETIR.ORG ISSN: 2349-5162 | ESTD Year : 2014 | Monthly Issue
JOURNAL OF EMERGING TECHNOLOGIES AND INNOVATIVE RESEARCH (JETIR)
An International Scholarly Open Access, Peer-reviewed, Refereed Journal

Analysis of Shell-and-Tube Heat Exchanger Performance using Segmental Baffles through Computational Fluid Dynamics (CFD)

Yalla Vara Prasad[1], J D Venkatesh[2]
[1]MTech Scholar, Department of Mechanical Engineering, Aditya Engineering College (A), Surampalem, A.P, INDIA.
[2]Assistant Professor, Department of Mechanical Engineering, Aditya Engineering College (A), Surampalem, A.P, INDIA

ABSTRACT
A heat exchanger is a device that is used to transfer heat from one fluid to another or from a solid surface to a fluid, while keeping the fluids or solids separate. The choice of heat exchanger depends on the specific application requirements, such as the flow rate, temperature, and pressure of the fluids, as well as the available space, maintenance requirements, and cost considerations. A shell and tube heat exchanger is a type of heat exchanger that consists of a cylindrical shell containing a bundle of tubes. One fluid flows through the tubes, while the other fluid flows outside the tubes in the shell. Heat is transferred between the fluids through the tube walls. This project aims to achieve two main objectives, which are, designing and simulating a shell and tube heat exchanger with a segmental baffle, and investigating the impact of baffle spacing on both heat transfer and pressure drop.The reduction in spacing generates a smaller recirculation area and high turbulence, which leads to higher heat transfer rates. Additionally, an increase in the number of baffles causes the fluid to travel a greater distance within the shell, resulting in a larger effective heat transfer area and higher heat transfer rates.

Introduction
A heat exchanger is a device that is used to transfer heat from one fluid to another, or from a solid surface to a fluid, while keeping the fluids or solids separate. Heat exchangers are commonly used in a wide range of industries, such as power generation, chemical processing, HVAC (heating, ventilation, and air conditioning), and food and beverage production, among others. Heat exchangers are extensively used in various industries to transfer heat between two or more fluid streams. They are required in any process involving cooling, heating, condensation, boiling, or evaporation of fluids, where the fluids undergo a phase change or require heating or cooling before the process. The shell-and-tube heat exchanger is a widely used device for transferring heat from one fluid to another. It is an important component in many industrial processes, and its performance is critical to the efficiency of these processes. Segmental baffles are commonly used in shell-and-tube heat exchangers to enhance heat transfer and fluid mixing. The use of computational fluid dynamics (CFD) has become increasingly popular in the analysis and optimization of heat exchangers. The design and performance of the heat exchanger may be optimised by using CFD simulations to gain a thorough knowledge of the fluid flow and heat transfer properties inside. This literature review focuses on the analysis of shell-and-tube heat exchanger performance using segmental baffles through computational fluid dynamics. The review provides an overview of the existing literature related to the use of segmental baffles in shell-and-tube heat exchangers and the application of CFD in the analysis of their performance.

Problem definition and procedure
The study involves a model of a shell with a diameter (D) of 80 mm, a length of 400 mm, and circular inlet and outlet sections with a diameter (ds) of 20 mm. The shell contains 4 tubes with a square bundle configuration having a 16mm internal diameter (di) and 2mm thickness. There are 8 equally spaced segmental baffles in the shell. To investigate the effect of the number of baffles on heat transfer and pressure drop of the shell, three different models with different numbers of baffles are created while preserving the other parameters and boundary conditions. The flow situation is thought to be turbulent, and the fluid characteristics are supposed to be constant. The problem's In Table 1, boundary conditions are provided.

Table 1: Boundary conditions for the model

Fluid	InletVelocity	InletTemperature	Outlet
ShellSide	1.2m/s	363K	PressureOutlet
TubeSide	0.4m/s	300K	PressureOutlet

Governing Differential Equation

Continuity Equation:

$$\frac{\partial\rho}{\partial t}+\frac{\partial}{\partial x_i}(\rho u_i)=0$$

Momentum equation

$$\frac{\partial}{\partial t}(\rho u_i u_j)=\frac{\partial\rho}{\partial x_i}+\frac{\partial\tau_{ij}}{\partial x_i}$$

Where

$$\tau_{ij}=\mu\left(\frac{\partial u_i}{\partial x_i}+\frac{\partial u_j}{\partial x_j}\right)-\frac{2}{3}\frac{\mu\partial u_k}{\partial x_k}\partial_{ij}$$

Energy equation:

$$\frac{\partial}{\partial t}(\rho E)+\frac{\partial}{\partial x_i}(u_i(\rho E+\rho))=\frac{\partial}{\partial x_i}\left(k\frac{\partial T}{\partial x_i}\right)$$

E is the total energy, while k is the thermal conductivity.

Equation representing turbulent kinetic energy:

$$\frac{\partial\rho k}{\partial t}+\frac{\partial\rho u_j k}{\partial x_j}=\frac{\partial}{\partial x_j}\left(\mu\frac{\partial k}{\partial x_j}\right)-\frac{\partial}{\partial x_j}\left(\frac{\rho}{2}u_j u_i u_i+pu_j\right)-\rho u_i u_j\frac{\partial u_i}{\partial x_j}-\mu\frac{\partial u_i}{\partial x_k}\frac{\partial u_i}{\partial x_k}$$

Rate of dissipation equation is

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\frac{\partial(\rho u_j\varepsilon)}{\partial x_j}=C_{\varepsilon 1}P_k\frac{\varepsilon}{k}-\rho C_{\varepsilon 2}\frac{\varepsilon^2}{k}+\frac{\partial}{\partial x_j}\left(\frac{u_t}{\sigma_\varepsilon}\frac{\partial\varepsilon}{\partial x_j}\right)$$

The advent of powerful computers and advanced numerical methods for solving physical problems has revolutionized the study and practice of fluid flow and heat transfer. This approach, known as CFD (Computational Fluid Dynamics), has made it possible to analyse complex flow geometries with much greater ease than traditional methods used for solving complicated problems. CFD is essentially an interdisciplinary field that combines fluid dynamics and numerical analysis.

The geometries were created using ANSYS WORKBENCH 15.0 and there were three different geometries, each with varying number of baffles.The meshing process was carried out using ANSYS meshing tool and the resulting mesh for each geometry can be seen in figure 2.

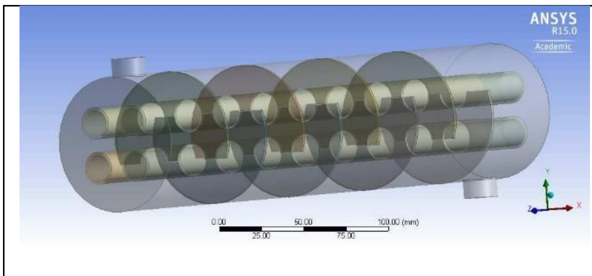

Figure 1: Shell and tube heat exchanger's overall shape with 8 baffles

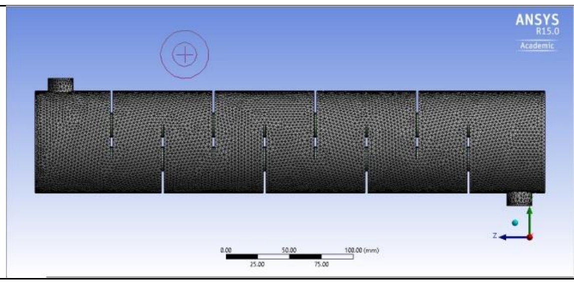

Figure 2: Overall meshing

Specification of flow condition

In conducting the analysis, the following presumptions were used:

(a) Since the flow is thought to be incompressible, the fluid's density doesn't change.

(b) No thought is given to any leakage from the area between the tube and baffle.

(c) Here is a heat transfer to the baffles not considered in the analysis.

(d) The working fluid is water, and its characteristics are considered to be constant.

(e) Analysis does not take into consideration the header impact.

To put it another way, the study was conducted using water as the working fluid with constant characteristics, assuming that the flow is incompressible, discarding any leaking through the space around the tube and baffle, ignoring heat transmission to the baffles, and removing the header effect.

Results and Discussion

Case 1

Figure 3's temperature contour shows how temperatures vary in various zones, indicating that the baffles' rear faces have lower temperatures than their front faces do due to the decreased flow in that area. Figure 4 also displays the pressure contour, which shows how pressure gradually decreases as fluid moves through the shell and pipe. As a bigger pressure drop requires more pumping power, this pressure drop is a crucial aspect in calculating the pumping requirements.

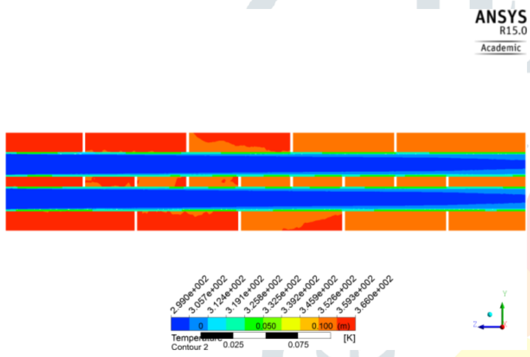

Figure 3: temperaturecontour

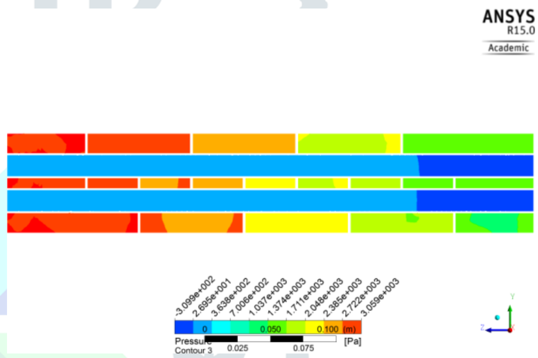

Figure 4: pressure contour

The part of the pipe immediately behind the baffles has the lowest temperature, as shown by Graph 5. The reason for this is heat transmission from the fluid to the surface of the pipe. The graph shows four minimum points because different baffles are in contact with the chosen pipe. Due to heat transmission, the temperature at the rear of each baffle is higher than the previous one. Because the temperature of the hot fluid is lower at the end of the shell, heat is transferred from the pipe to the cooler fluid, reducing the pipe surface temperature. Graph 6 shows how the axial variation of the surface the coefficient of heat transfer, which is dependent on fluid velocity and shape. The graph indicates fluctuations in the heat transfer coefficient due to high fluid turbulence. The back face of the baffle experiences reduced fluid flow and, consequently, lower heat transfer, resulting in a lower heat transfer coefficient compared to other regions. The heat transfer coefficient gradually increases after this region. Towards the end of the heat exchanger, where heat transfer is reduced, the heat transfer coefficient is also lower. On the inner surface of the pipe, Graph 7 indicates a change in surface Nusselt value along the axial direction. Given that the value of Nusselt is linearly proportional to the heat transfer coefficient, this curve is the same as the graph of the heat transfer coefficient vs length.

.

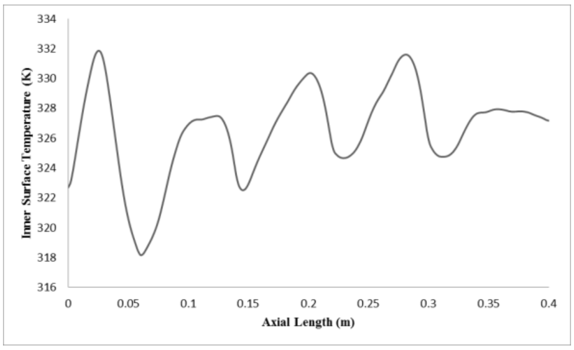

Figure 5: Temp. on the inside surface vs length

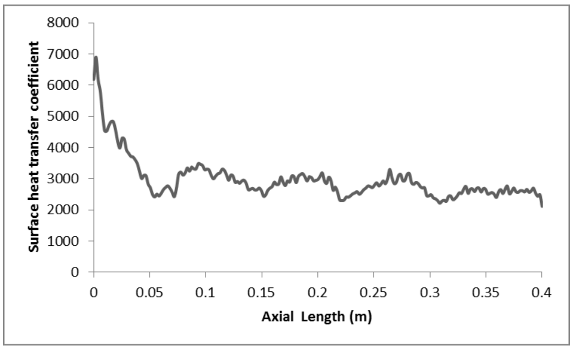

Figure 6: Coefficient of surface heat transfer vs length

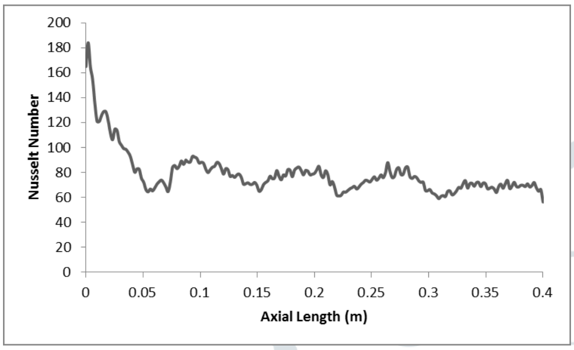

Figure 7:Nusseltnumbervslength

CASE 2

Figure 8's temperature contour reveals that the majority of heat transmission happens in the end part due to reduced contact between the hot fluid and tubes. As previously mentioned, the increased recirculation zone in this case results in reduced heat transfer compared to the previous two cases. Figure 9 shows that as the fluid advances, the pressure steadily decreases on both the shell and tube sides. However, compared to the previous two situations, this case's pressure decrease on the shell side is lower. Due to the lower pressure drop, less pumping force is needed to sustain the flow than in earlier cases.

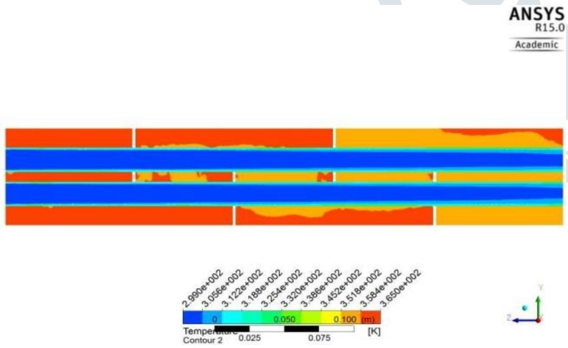

Figure 8: temperature contour

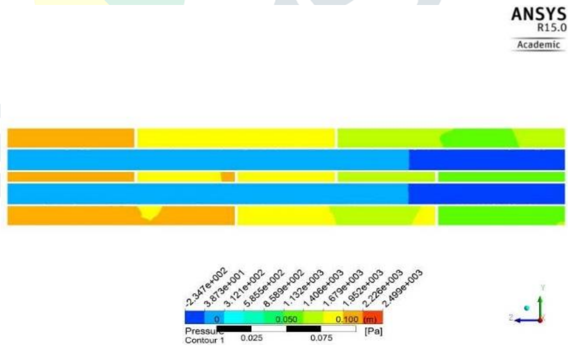

Figure 9: pressure contour

In graph 10, which depicts the temperature variation on the inner surface of the tube, it can be observed that the surface being monitored is connected to two baffles, resulting in the appearance of two back faces. The temperature of the back face region is lower than other portions, and the temperature at the back face of the second baffle is higher than that of the first baffle as a result of heat transfer. Graph 11 shows how changes in fluid velocity cause changes in the surface heat transfer coefficient with axial length. In the reticulating zone with reduced fluid flow, the heat transfer coefficient and the rate of heat transfer are lower. Since there are only two rear sides in this situation, two craters have formed. Given that the Nusselt number depends on the heat transfer coefficient, Graph 12 displays the Nusselt number's change with length along the same curve as the heat transfer coefficient, but with different values.

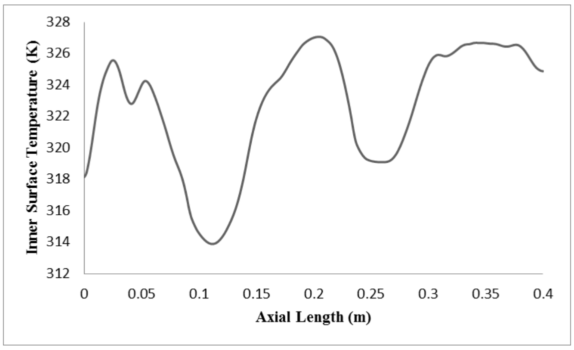

Figure 10: Inner surface temperature (K) vs axial length (m)

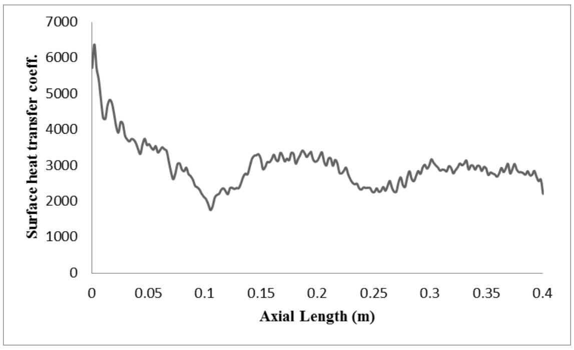

Figure 11: Heat transmission from surfaces vs axial length (m)

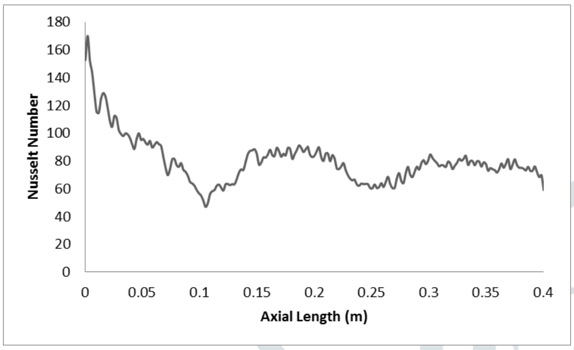

Figure 12: Nusselt number vs axial length (m)

The rate of heat transmission rate in different scenarios

According to Graph 13, heat transfer rises as the number of baffles increases. This is because the effective area of heat transmission increases when additional baffles are added. The heat transfer rates across various cases are mentioned in table number 2.

Table2: Heat transmission rate in different conditions

the quantity of baffles	rate of heat transfer (W)
4	8043.8305154292
6	9884.910562944
8	11053.37143

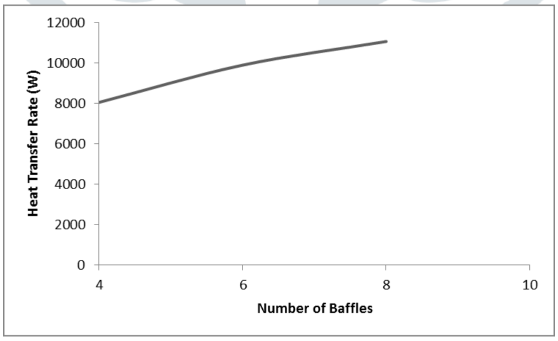

Figure13: Heat transfer rate vs number of baffles

Graph 14 displays the correlation between the number of baffles and pressure drop in the shell side. Increasing the quantity of baffles, pressure decreased in a shell's side also increases, which results in an increase in the required pumping power. The shell side pressure drop across various cases are mentioned in table number 3.

Table3: Different scenarios of shell side pressure drops

The amount of baffles	Decrease in shell side pressure (Pa)
4	2116.44502
6	2695.90933
8	3160.72314

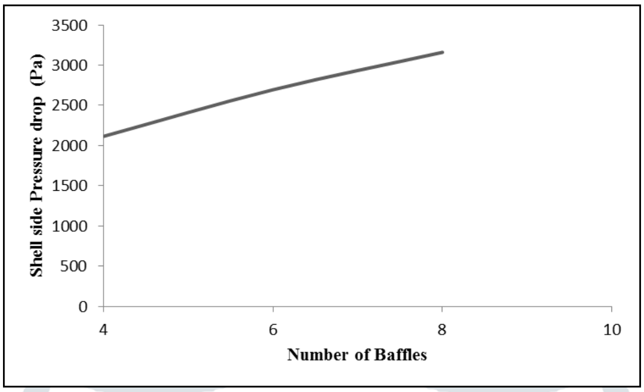

Figure14: Shell side pressure drop versus Baffle count

Conclusion

The impact of the number of baffles on heat transfer can be observed when the same shell length is used with an increasing number of baffles, which results in a decrease in the spacing between baffles. The reduction in spacing generates a smaller recirculation area and high turbulence, which leads to higher rate of heat transfer. Additionally, when the number of baffles rises, the fluid travels farther within the shell, resulting in a bigger effective heat transfer surface and faster heat transfer rates. The heat transmission rate with 8 baffles is 37.414% greater than with 4 baffles.

The number of baffles is rising the pressure decrease in the shell side also rises. This is due to the decreased space between baffles, resulting in a narrower path for fluid flow. As a result, the pressure drops, and the kinetic energy increases as the fluid passes through a narrower path. As the pressure decrease increases, so does the necessary pumping force to maintain the flow. It is important to consider pressure drop when designing a shell and tube heat exchanger. With 8 baffles compared to 4, there is a 49.34% larger pressure decrease on the shell side.

References

[1] Mohammed, H. A., et al. "Heat transfer and fluid flow characteristics in micro channels heat exchanger using nanofluids: a review." Renewable and Sustainable Energy Reviews 15.3 (2011): 1502-1512.

[2] Kakac, Sadik, Hongtan Liu, and AnchasaPramuanjaroenkij. Heat exchangers: selection, rating, and thermal design. CRC press, 2020.

[3] Shah, R. K., B. E. N. F. O. R. A. D. O. Thonon, and D. M. Benforado. "Opportunities for heat exchanger applications in environmental systems." Applied Thermal Engineering 20.7 (2000): 631-650.

[4] Carey, Van P. Liquid-vapor phase-change phenomena: an introduction to the thermophysics of vaporization and condensation processes in heat transfer equipment. CRC Press, 2018.

[5] Kakaç, Sadik, ed. Boilers, evaporators, and condensers. John Wiley & Sons, 1991.

[6] Warnakulasuriya, F. S. K., and W. M. Worek. "Heat transfer and pressure drop properties of high viscous solutions in plate heat exchangers." International Journal of Heat and Mass Transfer 51.1-2 (2008): 52-67.

[7] Selbaş, Resat, ÖnderKızılkan, and Marcus Reppich. "A new design approach for shell-and-tube heat exchangers using genetic algorithms from economic point of view." Chemical Engineering and Processing: Process Intensification 45.4 (2006): 268-275.

[8] T'Joen, Christophe, et al. "A review on polymer heat exchangers for HVAC&R applications." International journal of refrigeration 32.5 (2009): 763-779.

[9] Master, Bashir I., et al. "Most frequently used heat exchangers from pioneering research to worldwide applications." Heat Transfer Engineering 27.6 (2006): 4-11.

[10] Yang, Yahui, G. L. Morini, and Juergen J. Brandner. "Experimental analysis of the influence of wall axial conduction on gas-to-gas micro heat exchanger effectiveness." International Journal of Heat and Mass Transfer 69 (2014): 17-25.

[11] De, Dipankar, Tarun K. Pal, and SantanuBandyopadhyay. "Helical baffle design in shell and tube type

heat exchanger with CFD analysis." International Journal of Heat and Technology 35.2 (2017): 378-383.

[12] Yang, Sheng, Li Zhang, and Hong Xu. "Experimental study on convective heat transfer and flow resistance characteristics of water flow in twisted elliptical tubes." Applied Thermal Engineering 31.14-15 (2011): 2981-2991.

[13] Mohanty, Dillip Kumar. "Gravitational search algorithm for economic optimization design of a shell and tube heat exchanger." Applied Thermal Engineering 107 (2016): 184-193.

[14] You, Yonghua, et al. "Numerical modeling and experimental validation of heat transfer and flow resistance on the shell side of a shell-and-tube heat exchanger with flower baffles." International Journal of Heat and Mass Transfer 55.25-26 (2012): 7561-7569.

[15] Mukherjee, R. (1998). Effectively design shell-and-tube heat exchangers. Chemical Engineering Progress, 94(2), 21-37.

[16] Ohadi, M., Zhang, X., Keramati, H., Arie, M., Singer, F., Tiwari, R., &Shooshtari, A. (2018). Recent developments in high temperature heat exchangers: A review. Frontiers in Heat and Mass Transfer (FHMT), 11.

[17] Johnson, M., Vogel, J., Hempel, M., Hachmann, B., &Dengel, A. (2017). Design of high temperature thermal energy storage for high power levels. Sustainable Cities and Society, 35, 758-763.

[18] Teka, A. (2012). Thermal design of heat exchanger for a swimming pool.

[19] Fakheri, A. (2002, January). Log mean temperature correction factor: an alternative representation. In proceedings of the International Mechanical Engineering Congress and Exposition (pp. 17-22).

[20] Ozden, E., &Tari, I. (2010). Shell side CFD analysis of a small shell-and-tube heat exchanger. Energy Conversion and Management, 51(5), 1004-1014.

[21] G. V. S. Rao, C. J. Rao, and H. N. HariBabu, “Heat transfer analysis on shell and tube heat exchanger”, International Journal of Reasarch in Aeronautical and Mechanical Engineering, vol. 2, pp. 11–26, January 2014.

[22] Y. You, Y. Chen, M. Xie, X. Luo, L. Jiao, and S. Huang, “Numerical simulation and performance improvement for a small size shell-and-tube heat exchanger with trefoil- hole baffles”, Applied Thermal Engineering, vol. 89, pp. 220–228, 2015.

[23] G.-Y. Zhou, J. Xiao, L. Zhu, J. Wang, and S.-T. Tu, “A numerical study on the shell- side turbulent heat transfer enhancement of shell-and-tube heat exchanger with trefoil- hole baffles”, In The 7th International Conference on Applied Energy, Energy Proce- dia, pp. 3174–3179, 2015.

[24] J. Yang and W. Liu, “Numerical investigation on a novel shell-and-tube heat exchanger with plate baffles and experimental validation”, Energy Conversion and Management, vol. 101, pp. 689–696, 2015.

[25] J. Wen, H. Yang, S. Wang, Y. Xue, and X. Tong, “Experimental investigation on performance comparison for shell-and-tube heat exchangers with different baffles”, International Journal of Heat and Mass Transfer, vol. 84, pp. 990–997, 2015.

[26] E. Ozden and I. Tari, “Shell side CFD analysis of a small shell-and-tube heat ex- changer”, Energy Conversion and Management, vol. 51, pp. 1004–1014, 2010.

[27] A. D. Jadhav and T. A. Koli, “CFD Analysis of Shell and Tube Heat Exchanger to Study the Effect of Baffle Cut on the Pressure Drop”, International Journal of Re- search in Aeronautical and Mechanical Engineering, vol. 2, pp. 1–7, July 2014.

[28] C. N. Patil and N. S. Bhalkikar, “CFD Analysis of Shell and Tube Heat Exchanger to Study the Effect of Baffle Cut on the Pressure Drop and Heat Transfer Coefficient”, International Journal for Scientific Research and Development, vol. 2, pp. 649–654, 2014.

[29] S. P. Chit, N. A. San, and M. M. Soe, “Flow Analysis in Shell Side on the Effect of Baffle Spacing of Shell and Tube Heat Exchanger”, International Journal of Science, Technology and Society, vol. 3(5), pp. 254–259, 2015.

[30] B. Peng, Q. W. Wang, C. Zhang, G. N. Xie, L. Q. Luo, Q. Y. Chen, and M. Zeng, “An experimental study of Shell-and-Tube Heat Exchangers with continuous helical baffles”, Journal of Heat Transfer, vol. 129, pp. 1425–1431, 2007.

[31] E. S. Gaddis and V. Gnielinski, “Pressure drop on the shell side of shell-and-tube heat exchangers with segmental baffles”, Chemical Engineering and Processing, vol. 36, pp. 149–159, 1997.

[32] H. D. Li and V. Kottke, “Visualization and determination of local heat transfer coefficients in shell-and-tube heat exchangers for in-line tube arrangement by mass transfer measurements”, Heat and Mass Transfer, vol. 33, pp. 371–376, 1998.

[33] K. Kiran, A. M. M. Manoj, and C. J. Umesh, “Investigation of baffle spacing effect on shell side heat

transfer characteristics in shell and tube heat exchanger using computational fluid dynamics", Thermal Engineering, vol. 73, pp. 26022–26026, 2014.

[34] J. Prasanna, H. R. Purushothama, and D. K. Murugeshan, "A numerical analysis of hydrodynamic and heat transfer effects of shell and tube heat exchanger for different baffle space and cut", Mechanica Confab, vol. 2, June-July 2013.

[35] ANSYS 15.0 User Manual.