AUPET Press
Technical Journal of Daukeyev University
Volume 1, Issue 1, 2021, pp. 32-38

Experimental Study of a Concentric Tube Heat Exchanger with Helical Baffle Using CFD

Jithendra Sai Raja Chada, Akhil Yuvaraj Manda, Venkat Sandeep Gadi, Dharmalingam R.
Department of Mechanical Engineering, Pragati Engineering College, Surampalem, Kakinada 533001, India
Corresponding author: Chada Jithendra Sai Raja; csai0799@gmail.com
Received 13 March 2021; Received in revised form 18 March 2021; Accepted 25 March 2021; Published 14 April 2021
DOI: https://doi.org/10.52542/tjdu.1.1.32-38

Abstract. Heat exchangers are the most common equipment used to transfer heat from high-temperature fluid to low-temperature fluid without direct contact. The present study considers the analytical approach on a concentric tube heat exchanger with the helical baffle. The objective of the study is to reduce the size with effect to increase the effectiveness of the heat exchanger. A heat exchanger with 100 mm external diameter and 560 mm length contains a helical baffle with 20 degrees inclination. The designed heat exchanger is analysed by varying the mass flow rate of hot water from 0.25 Kg/s to 2 Kg/s at an interval of 0.25 kg/s at three different temperatures i.e. 363.16 K, 368.16 K, 373.16 K. A nanofluid is applied to cool the hot water without any loss. The mass flow rate of cold fluid is 2 Kg/s at 30 degrees Celsius. The results have displayed that the heat exchanger exhibited appreciable effectiveness at a flow rate of 0.25 Kg/s for hot water at 373.16 K temperature. There by suggesting it as the optimum model of the heat exchanger.

Keywords: Heat Exchanger, Baffle, Helical Structure, Concentric Tubes, Effectiveness.

1. Introduction

Heat exchangers are used in every processing and manufacturing industry globally. A heat exchanger consists of two fluids of different mass flow rates at different temperatures. The most commonly used heat exchanger is shell and tube, heat exchanger. In heat exchangers, the fluids are separated by a heat swapping surface. The area of this heat swapping surface is the key factor that decides the effectiveness of the heat exchanger. The fluid which is relatively at a higher temperature is the hot fluid, and the fluid which is relatively at a lower temperature is the cold fluid. The arrangement is made in such a way that one fluid flows through the tubes and another fluid flows over the tubes in the shell. The use of nanofluids increases the chance of heat transfer. But the use of nanofluids in heat exchangers increase the capital cost. So, the nanofluids are applied at some special case where the capital cost is not much important than performance. In this paper, we discussed a possible way to reduce to enhance the performance of the heat exchanger. The method opted for the enhancement of the heat exchanger is the reduction of the size of the heat exchanger. The considered model is evaluated at different parameters accordingly.

2. Literature Review

Karthik S., et al., worked on a numerical and analytical approach in terms of the design of a concentric tube heat exchanger. Sensible heat transfer is considered for operating in a chemical plant. ε-NTU and LMTD parameters are employed in terms of the design. The results depicted that both the numerical and analytical analysis generated the same results; making to consider that either of the two methodologies can be used [1]. Surender Kumar worked on the experimental analysis of a shell and tube type heat exchanger. The design was conceived according to Kern's method to speculate as per the norms of TEMA (Tubular Exchanger Manufacturer's Association). The concept model was designed using CAD software and the thermal analysis is tested using the ABAQUS 6.13. The generated results depicted the effectiveness values for the flow heat exchanger [2]. Bahuruteen H., et al., worked on the analysis in terms of the performance of a single shell four-tube heat exchanger. The heat exchanger was opted to be a simplified model of contour flow shell and tube heat exchanger. The heat exchanger was designed using CREO and the steady-state thermal analysis was simulated using ANSYS accordingly by applying several thermal loads on different faces and edges [3]. Vindhya V P D., et al., worked on the extensive effects of the severe loading conditions on the performance of a shell and tube type heat exchanger using Kern's method. The heat exchanger was analysed in terms of the steady-state thermal analytics on ANSYS 14.0. The practical model was further fabricated and tested under various parameters of insulations such as aluminium foil, cotton, wool, tape, etc.; at various ambient temperatures to see the effect. The generated results are counted on, to evaluate the performance of the heat exchanger [4]. Kevin Shah., et al., worked on the improvement of the efficiency of a shell tube heat exchanger. The tube diameter is varied and concerted into nine models based on the Taguchi method. The concept models are evaluated using the CFX analysis in ANSYS 14.5. The results are derived in terms of the best dimensions of the heat exchanger accordingly [5]. Raju V., et al., explained how to concept and design an oil cooler, especially for shell and tube heat exchangers. The general considerations are also mentioned in the work supportively. The final results are compared to the base design concepted in terms of the thermal and pressure drop calculations using the empirical formula [6]. Stephenraj V., et al., worked on the design parameters to obtain an optimum baffle design in terms of maximum heat transfer rate and also a suitable fluid for the working of the heat exchanger. The optimization is done concerning transfer efficiency and travel tube design and analysed it using CFD. The results are found out to be best with 35 degree baffle arrangement in terms of the heat transfer angle [7-9]. Amrutha Vijay D., et al., worked on the temperature distribution and stress value of a heat exchanger. The project dealt with the design and analysis of a heat exchanger using proE, Hypermesh, and ANSYS software [10]. Amarnath K., et al., conducted an experimental study on the banks of the tube heat exchanger with correspondence to different flow rates for hot and cold fluids. The results for the heat exchanger are studied in terms of heat transfer rate and effectiveness [11]. The design and optimization of a heat exchanger are evaluated accordingly on various bases. The factors responsible for the performance of the heat exchanger have been evaluated accordingly with temperature distribution, pressure drop, velocity variation, etc. [12-21].

2.1. Objective of the work

Heat exchange between two fluids or components or mediums has a huge impact in terms of the transfer of fluid medium whenever needed. So, they are intended to possess a greater impact in all industries. The intent of the present work is to the concept and evaluates an optimal design of baffle for a shell tube heat exchanger. The results are evaluated in terms of the mass flow rate at different temperatures.

2.2. Strategies and conditions

A 2 Dimensional heat exchanger with 100 mm external diameter and 560 mm length containing a helical baffle with 20 degrees inclination was designed and evaluated accordingly. The model of the baffle design was as shown in the Figure 1.

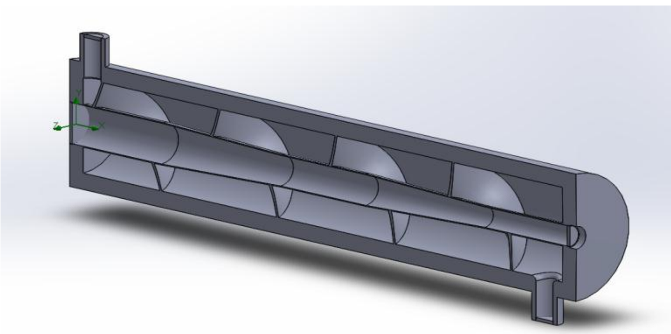

Figure 1. Cross sectional view of baffle.

The conceptual model was applied with two principles and equations namely continuity equation and Momentum equation as they are the key factors of the evaluation and are termed as follows:

Continuity Equation

The constitute of the fluid entering the intended profile must be same as the constitute of the fluid leaving the profile:

$$m_1 = m_2 \tag{1}$$

$$\frac{dm_1}{dt} = \frac{dm_2}{dt} \tag{2}$$

$$\rho_1 A_1 U_1 = \rho_2 A_2 U_2 \tag{3}$$

$$A_1 V_1 = A_2 V_2 \tag{4}$$

Momentum Equation

The rate at which the momentum of a fluid particle changes, must be equal to the forces acting along the flow stream.

Consider a functional sample from the depicted fluid flow.

F - force acting along the flow, N;

dA - cross sectional area of considered functional fluid sample;

dL - length of the functional fluid element;

dW - weight of the functional fluid element;

u - velocity of the functional fluid element;

p - pressure of the functional fluid element.

Assume that the fluid is steady, non-viscous, and in-compressible so that the frictional losses are zero and the density of the fluid is constant. The different forces acting on the fluid are:

a. pressure force acting in the direction of the flow (PdA);

b. pressure force acting in the opposite direction of the flow [$(P+dP)dA$];

c. gravity force acting in the opposite direction of the force ($dw\sin\theta$).

Total force:

$$F = F_G + F_P$$

The pressure force is considered in the direction of low:

$$F_P = PdA - (P + dP)dA \tag{5}$$

The gravity force considered in the direction of flow:

$$F_G = -dw \cdot \sin\theta \tag{6}$$

$$w = \rho dA \cdot dL \cdot g = -\rho g \cdot dA \cdot dL \cdot \sin\theta \tag{7}$$

$$\sin\theta = \frac{dz}{dL} = -\rho g \cdot dA \cdot dz \tag{8}$$

The net force is considered in the direction of flow:

$$F = m \cdot a$$

$$m = \rho dA \cdot dL = \rho dA \cdot dL \cdot a \tag{9}$$

We have:

$$\rho dA \cdot dU = -dP \cdot dA - \rho g \cdot dA \cdot dz \tag{10}$$

$$\frac{dP}{\rho} + u \cdot dU + dz \cdot g = 0 \tag{11}$$

On integrating the Euler's equation, we get the Bernoulli's equation:

$$\int \frac{dP}{\rho} + \int u \cdot dU + \int dz \cdot g = cons\tan t \tag{12}$$

$$\frac{P}{\rho} + \frac{U_2}{2} + z \cdot g = cons\tan t \tag{13}$$

$$\frac{\Delta P}{\rho} + \frac{\Delta U_2}{2} + \Delta z \cdot g = 0 \tag{14}$$

Kappa epsilon model: the k-epsilon model of the energy equation is generally used to analyse the turbulent flow. The final equations for the turbulent flow analysis are:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial t}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + P_K + P_b \rho \in \gamma_K + S_K \tag{15}$$

$$\frac{\partial}{\partial t}(\rho \in) + \frac{\partial}{\partial t}(\rho \in u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial \in}{\partial x_j}\right] + C_1 \frac{\in}{k}(P_K + C_3 P_b) - C_2 \rho \frac{\in^2}{k} + S_K \tag{16}$$

3. Results and Discussions

The shell tube heat exchanger is the most commonly used equipment in terms of industry. The present work discusses the increment in performance in terms of effectiveness and heat transfer flow rate. The model of the baffle was designed helically with 20 degree inclination [1]. The design and evaluation are simulated using the SOLIDWORKS software. The concept model of the baffle was evaluated in terms of the mass flow rate at three different inlet temperatures. The mass flow rate was divided into 8 points ranging from 0.25 Kg/s to 2 Kg/s with 0.25 Kg/s interval for each point. The inlet temperatures are opted to be 363.16 K, 368.16K and 373.16 K. The baffle was evaluated for various outlet values including outlet temperature, pressure drop, heat transfer rate, and heat transfer coefficient. The end results for the baffle are considered based on the outlet temperature i.e. maximum heat transfer. The appreciable results are obtained at a lower inlet mass flow rate i.e. 0.25 Kg/s [2]. The appreciable results are termed at 0.25 Kg/s with an increase of 1.275 % in terms of the outlet temperature concerning the inlet temperature of 368.16 K. Upon comparing with the inlet temperature of 363.16 K, the outlet temperature has been increased by 2.525 %. This depicts the maximum difference for the baffle at a temperature of 373.16 K, thereby suggesting it as the optimum model for the required considerations. From the results displayed in the Table.1, the maximum outlet temperature of 367.74 K is obtained at 373.16 K inlet temperature. Due to the lower inclination angle, the design of the baffle contributed to the increase in the heat transfer values of the heat exchanger [2]. The other contours evaluated at different temperatures for an inlet mass flow rate of 0.25 Kg/s are displayed in the Table.2.

Table 2. Various contour values at different temperatures.

Inlet Temperature	Outlet Temperature	Pressure Drop	Heat transfer Rate
363.16 K	358.45	46.92 Pa	4970.375 W
368.16 K	363.05	47.18 Pa	5432.088 W
373.16 K	367.74	53.45 Pa	5834.042 W

From the Table 2, we can observe that the maximum values for the outlet temperature, pressure drop, and heat transfer rate are 367.74 K, 53.45 Pa, 5834.042 W respectively at an inlet temperature of 373.16 K. The experimental study is conducted at an inlet of hot fluid at 0.25 Kg/s. The temperature variations are 363.16 K, 368.16 K and 373.16 K. The conditions on the cold fluid are kept constant. The conditions of cold fluid are 2 Kg/s and 30 degrees Celsius. The experimental setup is as shown in Figure 2.


Figure 2. Experimental setup.

Table 1. Outlet temperatures at various inlet temperatures.

Inlet temperature (K)	Inlet Mass flow rate, Kg/s	outlet temperature (K)	pressure drop (pa)	heat transfer rate (W)
363.16	0.25	358.45	46.92	4970.375
	0.5	360.02	191.93	6680.955
	0.75	360.75	436.37	7716.419
	1	361.09	780.85	8868.672
	1.25	361.49	1225.91	8988.34
	1.5	361.7	1771.72	9461.239
	1.75	362.55	2418.55	9846.68
	2	362.99	3166.58	10166.1
368.16	0.25	363.05	47.18	5432.088
	0.5	364.75	192.9	7255.014
	0.75	365.55	438.48	8423.724
	1	365.86	784.5	9854.412
	1.25	366.33	1231.58	9854.781
	1.5	366.57	1779.801	10343.54
	1.75	366.75	2429.48	10708.394
	2	366.89	3180.75	11095.592
373.16	0.25	367.74	53.45	5834.042
	0.5	369.34	193.86	8104.835
	0.75	370.32	440.63	9072.372
	1	370.67	788.26	10642.63
	1.25	371.19	1237.4	10614.685
	1.5	371.31	1787.96	11994.717
	1.75	371.54	2440.55	12447.106
	2	371.78	3195.35	11941.784

The values of experimental and simulated data are as follows.

Table 3. Experimental results.

Inlet Temperature	Simulated Data	Experimental Data
363.16 K	358.45	345.54
368.16 K	363.05	350.54
373.16 K	367.74	356.78

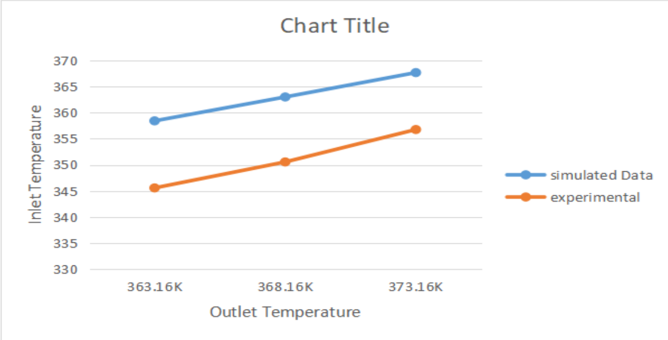

Figure 3. Graph plotted to compare the experimental and simulated data.

4. Conclusions

The shell tube heat exchanger was conceived and evaluated accordingly. The size of the heat exchanger can be reduced by modifying the baffle design which promotes the effectiveness of the heat exchanger. The baffle design was modified by 20 degree internal angle. The results are obtained for various contours including the outlet temperature. The maximum heat transfer was found to be at 373.16 K with an inlet mass flow rate of 0.25 Kg/s. The contours for the other parameters are also exhibited and evaluated accordingly. The outlet temperature was increased by 2.25 % for the heat exchanger when the inlet temperature was intended to be 373.13 K with an inlet mass flow rate of 0.25 Kg/s. Thereby suggesting the heat exchanger be made with the conditions owing to the results obtained.

References

[1] K. Silaipillayarputhur, T. A. Mughanam, A. A. Mojil, M. A. Dhmoush, Analytical and numerical design analysis of concentric tube heat exchangers, IOP Conference Series: Materials Science and Engineering, 272, 012006 2017, 1-11.

[2] Surender Kumar, Experimental analysis of shell and tube heat exchanger by using ABAQUS, International Journal of Engineering Sciences and Research Technology, 7(7), 2018, 340-346.

[3] H. A. A. Bahuruteen, S. G. Daniel, B. Rajasekaran, M. Selvaganesh, Prasad J. Sujin, Performance analysis of single shell four tube heat exchanger, International Journal of Pure and Applied Mathematics, 119(12), 2018, 13729-13734.

[4] V. V. P. Dubey, R. R. Verma, P. S. Verma, A. K. Srivastava, Performance analysis of shell and tube type heat exchanger under the effect of varied operating conditions, IOSR Journal of Mechanical and Civil Engineering, 11(3), 2014, 8-17.

[5] V. Raju and N. Naik, Thermal design and analysis of shell and tube heat exchanger, Journal of Emerging Technologies and Innovative Research, 4(11), 2017, 722-728.

[6] V. Stephenraj and M. K.Sathishkumar, Design and analysis of heat exchanger for maximum heat transfer rate (multi model optimisation technique), International Research Journal of Engineering and Technology, 5(1), 2018, 1421-1426.

[7] D. A. Vijay, and P. Snehalatha, Computer aided design and analysis of a heat exchanger, International Journal of Science, Engineering and Technology Research, 5(7), 2016, 1421-1429.

[8] K. Amarnath, A. Kishore, A. Kathaperumal, K. Karthickraja, K. Karthikeyan, Design and fabrication of bank of tubes counter flow heat exchanger, International Journal of Engineering Research & Technology, ICONEEEA - 2k19 Conference Proceedings, 7(2), 2019, 1-8.

[9] M. Irshad, M. Kaushar, G. Rajmohan, Design and CFD analysis of shell and tube heat exchanger, International Journal of Engineering Science and Computing, 7(4), 2017, 6453-6457.

[10] P. Bichkar, O. Dandgaval, P. Dalvi, R. Godsae, T. Dey, Study of shell and tube heat exchanger with effect of baffles, 2nd International Conference on Materials Manufacturing and Design Engineering, Procedia Manufacturing, 20, 2018, 195-200.

[11] D. De, T. K. Pal, S. Bandyopadhyay, Helical baffle design in shell and tube type heat exchanger with CFD analysis, International Journal of Heat and Tehnology, 35(2), 2017, 378-383.

[12] E. Akpabio, I. Oboh, E. O. Aluyor, The effect of baffles in shell and tube heat exchangers, Advanced Materials Research, 62(64), 2009, 694-699.

[13] S. S. Kamthe and S. B. Barve, Effect of different types of baffles on heat transfer and pressure drop of shell and tube heat exchanger: A review, International Journal of Current Engineering and Technology, Special Issue 7, 2017, 358-362.

[14] R. T. K. Raj and S. Ganne, Shell side numerical analysis of a shell and tube heat exchanger considering the effects of baffle inclination angle on fluid flow, Thermal Science, 16(4), 2012, 1165-1174.

[15] A. Kabir and A. Hafiz, Modeling and Simulation of a Supersonic Convergent Divergent Nozzle Using Computational Fluid Dynamics (CFD), 5 th International Conference on Engineering Research, Innovation and Education School of Applied sciences and Technology, 2018, 1-6.
[16] V. Vekariyamukesh, G. R. Selokar, A. Paul, Optimization and design of heat exchanger with different materials, I JMEMS, 5(1), 2012, 37-42.
[17] R. Aghayari, H. Maddah, M. Zarei, M. Dehghani, S. G. K. Mahalle, Heat transfer of nanofluid in a double pipe heat exchanger, International Scholarly Research Notices, 2014, 736424, 1-7.
[18] J. Albadr, S. Tayal , M. Alasadi, Heat transfer through heat exchanger using Al_2O_3 nanofluid at different concentrations, Case Studies in Thermal Engineering, 1(1), 2013, 38-44.
[19] G. Huminic, A. Huminic, Application of nanofluids in heat exchangers: A review, Renewable and Sustainable Energy Reviews, 16(8), 2012, 5625-5638.
[20] M. S. E. Rao, D. Sreeramulu, D. A. Naidu, Experimental Investigation of Heat transfer rate of Nano fluids using a Shell and Tube Heat exchanger, IOP Conf. Series: Materials Science and Engineering, 149, 012204, 2016, 1-8.
[21] G. J. Tertsinidou, C. M. Tsolakidou, M. Pantzali, M. J. Assael, New measurements of the apparent thermal conductivity of nanofluids and investigation of their heat transfer capabilities, J. Chem. Eng. Data, 62(1), 2017, 491-507.