RESEARCH ARTICLE

SAMRIDDHI Volume 15, Issue 1, 2023 Print ISSN: 2229-7111 Online ISSN: 2454-5767

Investigation of shell and tube heat exchanger with segmental baffles and novel tube arrangements to enhance the performance of Re-boiler - A CFD analysis

Ashutosh Kumar[1], Suman Kumar Saurabh[2*]

[1, 2]Department of Mechanical Engineering, National Institute of Technology Jamshedpur, 831014

Abstract

In this paper, a thermal performance investigation has been conducted on shell and tube heat exchangers used as re-boilers for both with and without baffles. Common steam was employed as the heating medium. A three-dimensional two-fluid numerical model was developed for the re-boiler shell side, aiming to calculate two-phase flow for thermal-hydraulics performances. Four models of shell and tube heat exchangers were created, including two (with and without baffles) with square tube arrangements and triangle tube arrangements. The shell diameters of 297mm and the length of 660 mm. The process fluid for the shell side was n-pentane, while water was selected for the tube side. The highest Nu value for a baffled triangular tube configuration was achieved. The enhancement in Nu number was found to be 58.87% and 55.61% for the baffled triangular and squire tube arrangement respectively when compared with the no baffled tube.

Keywords: Heat exchanger, shell and tube, baffle, Re-boiler.

SAMRIDDHI : A Journal of Physical Sciences, Engineering and Technology (2023); DOI: 10.18090/samriddhi.v15i01.38

Corresponding Author: Suman Kumar Saurabh, Department of Mechanical Engineering, National Institute of Technology Jamshedpur, 831014, e-mail: sksmech2012@gmail.com

How to cite this article: Kumar, A., Saurabh, S.K. (2023). Investigation of shell and tube heat exchanger with segmental baffles and novel tube arrangements to enhance the performance of Re-boiler - A CFD analysis. *SAMRIDDHI : A Journal of Physical Sciences, Engineering and Technology,* 15(1), 213-219.

Source of support: Nil

Conflict of interest: None

Introduction

Re-boilers are the shell and tube heat exchangers that are used to produce vapour supplied to the bottom tray of a distillation column. The liquid in the bottom of the column is partly vaporized in the device, which usually is of a shell-and-tube type of exchanger. The heating medium is generally condensing steam, but other commercial heat transfer fluids can also be used. Boiling takes place either in the shell or in the tube, given the type of shell and tube heat exchanger. Devices that supply vapour for operating other units are termed vaporizers, which are analogous in most respects to re-boilers. Kumar et al. [1] developed a mathematical model to predict tube-row wise heat transfer coefficient, vapour quality, and mass flux in the bundle based on phenomena like recirculation of liquid in the tube bundle and the cascading effect of interaction due to vapours generated at different tubes. It has been found that heat flux and pressure variation have a significant effect on the recirculation rate. Heat transfer coefficient, vapour quality and mass flux increase gradually from the bottom to the top row of bundles keeping heat flux and pressure constant. SCHÜLLER and K. [2] developed a 2-D model which provides local flow directions on a shell-side of a kettle shell and tube heat exchanger. By introducing a velocity-scaling factor local velocities in liquid only regions are obtained. The effect of local heat transfer and drying out in tube bundles helps in achieving a more efficient and safer kettle shell and tube heat exchanger design. Zhang et al [3] developed a numerical model to predict the key performance parameters of a water-logged evaporator while considering a few operating fluids. A shell and tube heat exchanger configuration was considered, and a performance model was developed to account for boiling on individual tubes, merging of vapour bubbles, and movement under gravity. Milada et al. [4] developed a 2-D two-fluid numerical model for the calculation of two-phase flow thermal-hydraulics on the shell side of the shell and tube heat exchanger. The two-phase flow round tubes were designed with the porous media approach. Simulations were performed for existing conditions of performed physical experiments. The obtained numerical outcomes were compared with the existing measured data of void fraction inside the tube. McNeil et al. [5] have the confirmation of the validity of the experimental data and the new understanding of the flow

© The Author(s). 2023 Open Access This article is distributed under the terms of the Creative Commons Attribution 4.0 International License (http://creativecommons.org/licenses/by/4.0/), which permits unrestricted use, distribution, and non-commercial reproduction in any medium, provided you give appropriate credit to the original author(s) and the source, provide a link to the Creative Commons license, and indicate if changes were made. The Creative Commons Public Domain Dedication waiver (http://creativecommons.org/publicdomain/zero/1.0/) applies to the data made available in this article, unless otherwise stated.

processes within the tube bundle creates the possibility of developing a 2D model for industrial design purposes. McNeil et al. [6] revealed that the result of the limited experimental data that are available, the two-fluid model presented does replicate the experimental finding reasonably well and reduces the 1-D model in appropriate circumstances. Failure analysis on unexpected perforation of heat exchanger tube in methacrylic acid heat exchanger of speciality chemical plant. No evidence of corrosion or distortion was detected on the failed tubes. Vortex erosion was the main failure mechanism of the failed tube. The formation of the perforation had undergone three evolution stages in this study. In 1st stage, unreasonable flushing operation caused mechanical indentations at the symmetrical positions on the inner wall of the failed tube. 2nd stage Erosion then happened with a shallow impact angle in the mechanical indentations, leading to the formation of the pits. 3rdstage selective local erosion was accelerated by the occurrence of vortex erosion on the pits, resulting in perforation failure [7]. Saad et al. [8] discovered that the temperatures and flow within the Shell and Tube of both the heat exchangers have been all calculated. The temperature of warm water in the inlet zone will be raised as the speed increases and the temperature of warm water at the socket will be reduced as the speed increases. Miyazaki et al. [9] investigated experimentally the effects of tube arrays, on void fraction and heat transfer characteristics. 2-D void fraction distribution and heat transfer coefficient was compared in in-line and staggered tube bundles with pitch-to-diameter ratio. X-ray radiography was used for the measurement of void fraction distribution. Time-average void-fraction was greater around upstream of a tube than that downstream at bubbly flow conditions for both bundles. Under irregular flow conditions, time-average void fractions at the maximum gap were higher than that around the other points in both tube bundles. For the in-line tube bundle, enhancement of the heat transfer clearly appeared between ±90 to 180°. For the staggered tube bundle, the heat transfer was found to be improved all over the pipe. Multidimensional model of pool boiling with vapour generation on tubes in a bundle is presented by Maslovaric et al. [10]. It is based on the mathematical solution of the two-phase, two-fluid flow model, which consists of the mass and momentum conservation equations for each phase, energy equation for the liquid phase and required closure laws for the calculation of interface transfer procedure. Singh et al. [11] investigated the effectiveness in shell and tube heat exchanger using CFD analysis with the help of ANSYS fluent with 4 cases i.e., without fins, spiral fins, baffle fins and rectangular fins. Out of these four conditions best performance i.e., effectiveness is achieved in heat exchangers with rectangular fins. Anand et al. [12] designed a heat exchanger that used nine tubes. The redesigned heat exchanger was designed using a computational fluid dynamics scheme that considered conditions including tube fluid velocity and pressure drop. Amini et al. [13] numerically studied the thermal performance of a shell-and-tube heat exchanger under various situations using Ansys Fluent software. The impact of the relative tube wall roughness and Reynolds number on total heat transmission and pressure loss are explored. The effect of basic segmented tube fins on heat transmission is investigated using two independent adjustable parameters, fin pitch, and fin height, while surface roughness is kept constant. The impact of these on heat exchanger thermal performance is explored, with findings compared to no-fin and segmented-fin designs. The tube fins were proved the heat exchanger's overall efficiency is imporved. Helical fins and segmented fins were utilized to boost efficiency. Increasing fin height improved the efficiency whereas increasing fin pitch decreases efficiency. When compared to plain tubes, the effectiveness of the helical fins heat exchanger was improved. While increasing surface roughness improved the efficiency. Low surface roughness and helical fins with smaller pitches and larger heights were employed, the heat exchanger's efficiency was significantly boosted. Arani et al. [14] study enhanced the fluid flow and heat transfer of heat exchangers on the shell side. Efficiency evaluation coefficients of the angle elliptical tube bundle were 20.9 and 10.7% higher than the horizontal and vertical tube arrangement respectively.

This study aims to make a comparison in terms of pressure drop, temperature difference, and Nusselt number, among the models where with and without baffle attached to the shell and tube for different tube arrangements.

Process methodology and solution

A numerical solution extracts information regarding pressure distribution, temperature distribution, and flow parameters from a set of mathematical equations (1) to (5) that are solved using finite volume method-based simulations. A computational scheme central difference is used to discretize viscous and thermal diffusion terms. A second-order upwind scheme is employed to discretize the advection terms into small cells and use them for the moment energy and turbulent kinetic energy and dissipation rate equations. The SIMPLE method (Semi-Implicit Method for Pressure Linked Equations) was taken to solve the turbulent problem for pressure-velocity coupling.

The governing equations that will be used in this project are as follows:

Conservation of mass

$$\nabla.(\rho V)=0 \quad (1)$$

Momentum Equation

In x-direction: $$\nabla.(\rho u V) = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{yz}}{\partial z} \quad (2)$$

In y-direction: $$\nabla.(\rho v V) = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} \quad (3)$$

In z-direction: $$\nabla.(\rho w V) = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} + \rho g \quad (4)$$

Table 1: Specification of the modelled heat exchanger

SN	*Specification*	*Arrangement (mm)*
1.	Shell outer diameter ()	297
2.	Shell inner diameter ()	294
3.	Shell length(L)	660
4.	Tube outer diameter ()	37.8
5.	Tube inner diameter ()	36
6.	Tube length(L)	660
7.	Tube pitch length (Pt)	46.875
8.	Baffle pitch(B)	94.25
9.	No of baffle ()	6

Table 2: Properties of the process fluids [15]

Property	*Cold Fluid (shell side) n-pentane (at 244 K)*	*Hot fluid (tube side) Water (at 363 K)*
Specific heat (J/kgK)	2177	4189
Thermal conductivity(W/mK)	0.136	0.66
Prandtl number	3.97	2.62
Density (kg/m^^{3})	626	979.5
Mass flow rate (kg/s)	1.2	2.5
Viscosity (Ns/m)	0.000229	0.000412

Energy Equation: $\nabla.(\rho eV) = -p\nabla.V + \nabla.(k\nabla T) + q + \emptyset$ (5)

Where, $\phi = \left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z}\right)^2 + 2[\left(\frac{\partial u}{\partial x}\right)^2 + \left(\frac{\partial v}{\partial y}\right)^2 + \left(\frac{\partial w}{\partial z}\right)^2]$

Geometry

The table below states the specification of the heat exchanger model for square and triangular tube arrangement with segmental baffle of 25%. The same dimensions will be used to define a heat exchanger for square and triangular tube arrangement without baffle. Specification of different type arrangement are given in table 1.

The model of heat exchanger having different tube arrangement attached with or without baffle was generated for CFD simulations in ANSYS 19.2. Four models were created as shown below based on tube arrangement and with and without baffle.

Mesh

The 3D mesh was generated for the CFD simulation in ANSYS 19.2 software, where the shell was meshed unstructured with overall element size to be 4.5mm and the tubes inside the shell, the tube domains and baffles was meshed structured with overall element size to be 1.5mm for tube and tube domain and 3mm for baffles. Inflation layer was introduced in shell side, tube domain, tube, and baffle. In all of the above sides smooth transition inflation was chosen, where in the transition ratio was taken to be 0.272; maximum inflation layer to be 2 and the growth rate of the layers in all the sides was taken to be 1.2.

Grid independency test

The governing equations (1) to (5) are solved using finite volume method. And computation carried out by 3D non uniform set of mesh with clustering near the solid fluid interface. An extensive grid independence test has been carried out. The relative convergence criterion for residuals was set to 10^{-6}. For shell and tube heat exchanger, five different grid systems have been considered. The average Nusselt number for each grid systems are presented in Table 3. Grid size with 3500000 elements has been considered for the simulation. Figure 3 depicts a comparison of the variation of Nusselt number for different pitch arrangement of tube with increasing number of mesh elements. It is found that the variation in *Nu* for two finer mesh M4 and M5 have less than 2%. Therefore, the final mesh density of 3500000 elements was considered for all computation.

Boundary Conditions

In the model, different initial conditions were applied for different zone. Assume outer face of shell is adiabatic wall. Shell material: Steel, Tube material: Copper, Tube domain: Water (properties given in table 2), Shell Domain: n-pentane (properties given in table 2), Baffle material: Aluminium. The shell side mass flow rate at inlet is 1.2 kg/s at 224K and at an absolute pressure outlet. The tube side mass flow rate

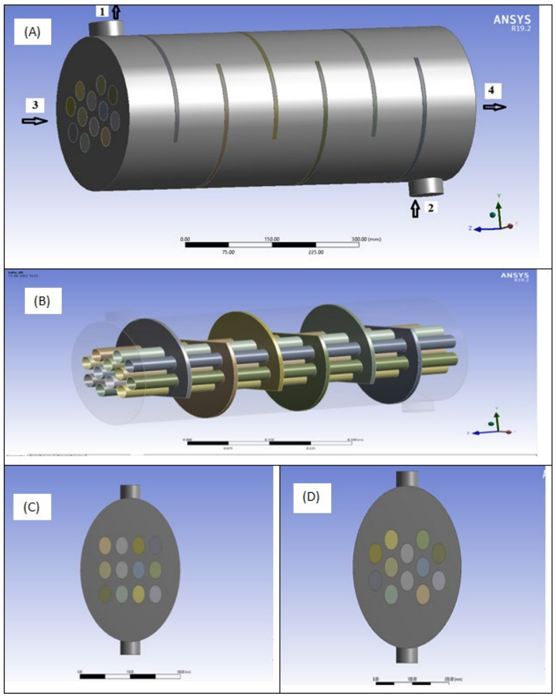


Figure 1: (A). shell and tube heat exchanger – 1. Outlet of cold fluid 2. Inlet of cold fluid, 3. Inlet of hot water 4. Outlet of hot water. (B) Inside arrangement of tube attached segmental baffle. (C) Squire pitch arrangement of tube. (D) Triangular pitch arrangement of tube.

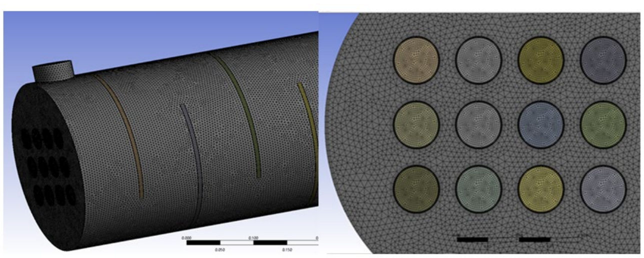


Fig. 2: Isometric view meshing of shell and Meshing of Square tube arrangement

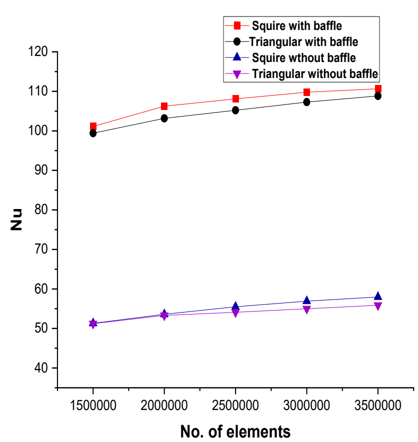


Figure 3: variation of *Nu* with mesh elements

at inlet is taken to be 2.5 kg/s at 363K and at an absolute pressure outlet.

Results and Discussion

After the simulations the following variations can be discussed below, where the parametric study is done based on data generated on the ANSYS 19.2

Effect on pressure in the shell as an investigation domain

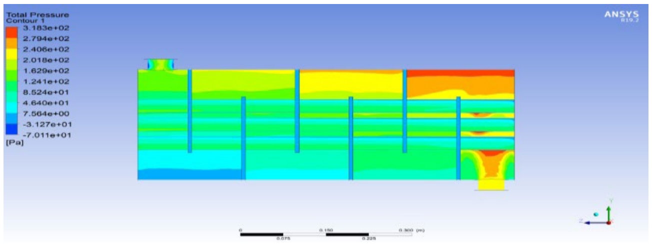
The above figure 5. represents the pressure of heat exchangers with and without baffle with square and triangular tube arrangement. Here in these figures, pressure keeps on decreasing along longitudinal direction. A similar profile is shown in square tube arrangement and triangular tube arrangement may it be with or without baffle. The difference between the profiles for with baffle and without baffle is that in device without baffle the pressure contour along the longitudinal is constant throughout the length of the tube and varies along y-direction (increases along y-direction). But, in device with baffle, the baffles help to direct the fluid and help the fluid to flow along a particular manner. Also, pressure along y- direction is more when it enters the first baffle. Furthermore, when the fluid moves along z-direction, the pressure along longitudinal keeps on decreasing.

The pressure variations are found to be more in baffles are considered in the shell due to a new obstacle are available along its length developed by baffle.

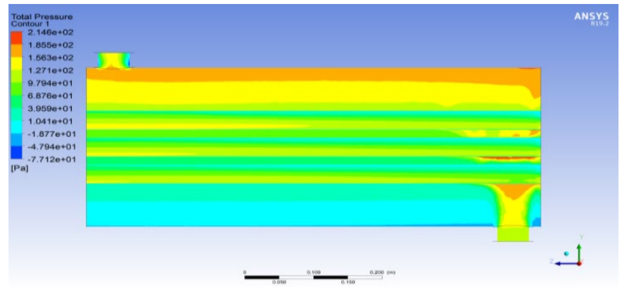
The figure 9. shows for square tube arrangement with baffle has a highest pressure drop of 126.69 Pa followed by triangular tube arrangement with baffle 110.13Pa and 79.34Pa for triangular tube arrangement without baffle. The minimum pressure drop was found in the square tube arrangement without baffle as 73.79Pa. The pressure developed in the baffle attached was 1.59 times the without baffle for triangular tube arrangement. And this value was 1.49 time for squire arrangement of tube. It means that the triangular arrangement creates higher resistance factor and fluid flowing through the shell losses energy more quickly than the squire arrangement of tube.

Temperature Variation

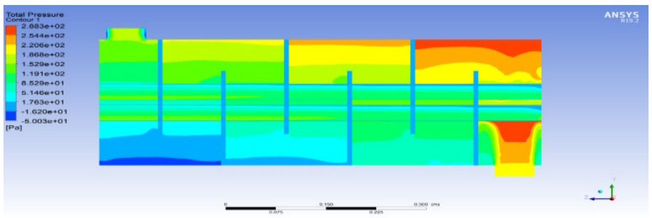
The Figure 7 the temperature variation along with length of shell and tube. The cold fluid enters the shell from below of the shell which is entry gate of cold fluid but hot fluid enters from left entry gate section of the tube to exits from exit gate which is right hand side of the whole test domain. It is clearly seen that the hot fluid without baffle exits with higher temperature but when it is operates with baffle then it getting colder. The triangular tube arrangement is better heat extracted than squire. And cold fluid getting heat then exits from outlet gate of the shell. For cold fluid it is also observed that the triangular tube arrangement is better for heating the fluid. So overall the triangular is very good to enhancing the



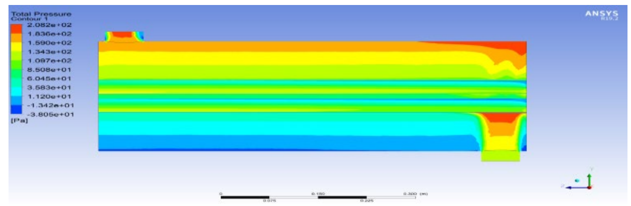
(a) **square tube arrangement with baffle**



(b) **square tube arrangement with baffle**



(c) **triangular tube arrangement with baffle**



(d) **triangular tube arrangement without baffle**

Figure 5: pressure variation across the heat exchanger different tube arrangement

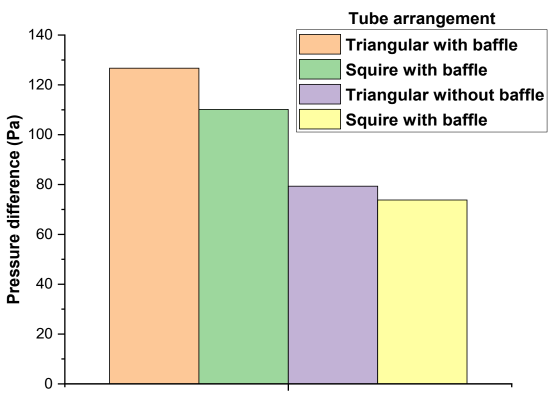
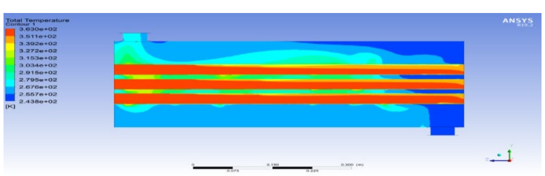


Figure 6: Pressure drops for different sets of models.

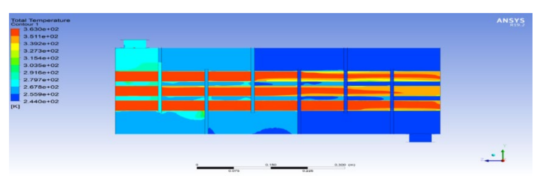
heat transfer for colder fluid.

Here, the above figure 8 shows outlet temperature of hot fluid flowing inside shell domain. The fluid flows longitudinal and as the hot fluid flowing in tubes encounters cold fluid which flows in shell side domain, the hot fluid loses its heat and cools down and the cold fluid gains heat. The outlet temperature getting more hotter in case of triangular with baffle tube arrangement.

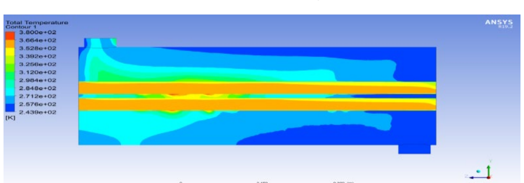
Figure 9 shows the outlet temperature of the cold process fluid flowing over the tubes which is being heated by the hot fluid passing inside the tube. The out temperature of cold fluid was 290.45K, 272.12K, 264.34K and 241.23K for triangular with baffle, squire with baffle, triangular without baffle and squire without baffle respectively. And the same



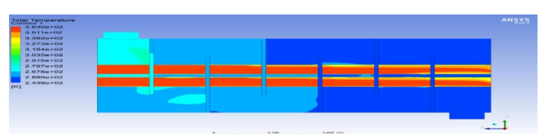
(a) square tube arrangement without baffle



(b) square tube arrangement with baffle

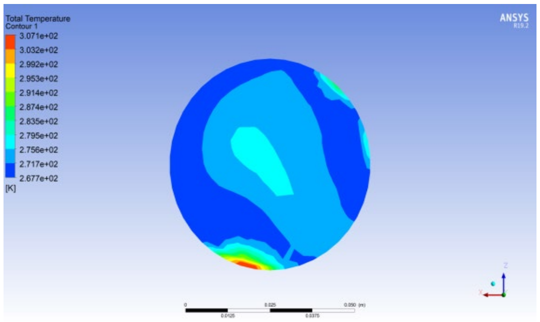


(c) triangular tube arrangement without baffle

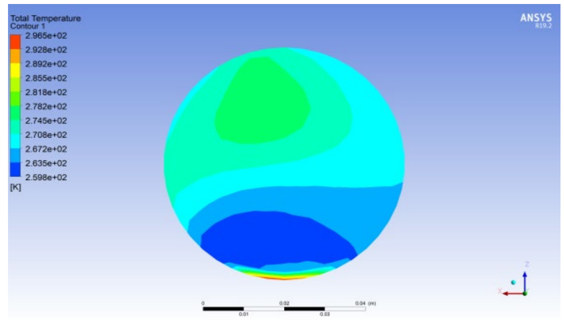


(d) triangular tube arrangement with baffle

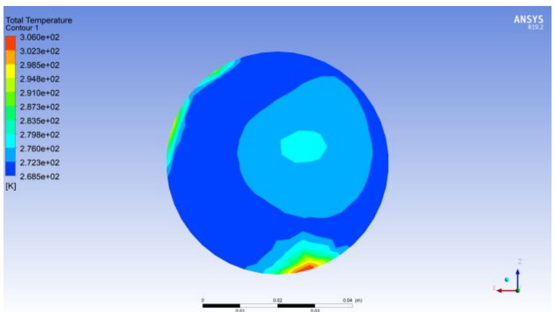
Figure 7: temperature variation across the heat exchanger for different tube arrangement



(a) square tube arrangement for without baffle



(b) square tube arrangement for with baffle



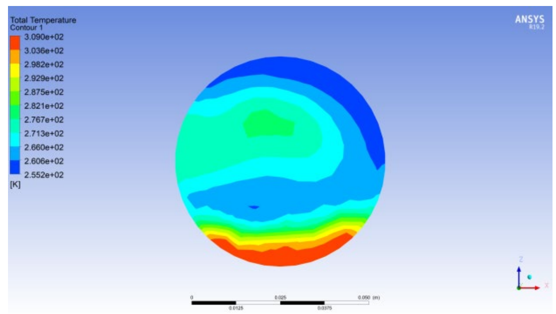


Figure 8 : Temperature variation of cold fluid outlet for different tube arrangement

for hot fluid the outlet temperature were 306.15K, 325.34K 337.56K and 348.7K.

Effect of Nu in the shell

The figure 10 shows the Nusselt number for different tube arrangement with baffle and for without baffle. The highest value of Nusselt number have been observed for triangular

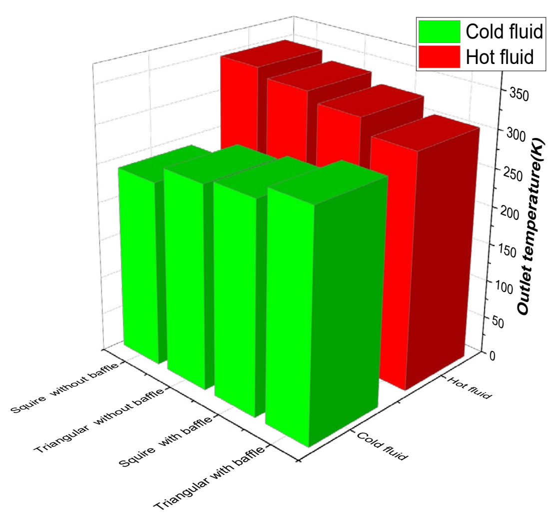


Figure 9: outlet temperature of cold and hot fluid for various tube arrangement

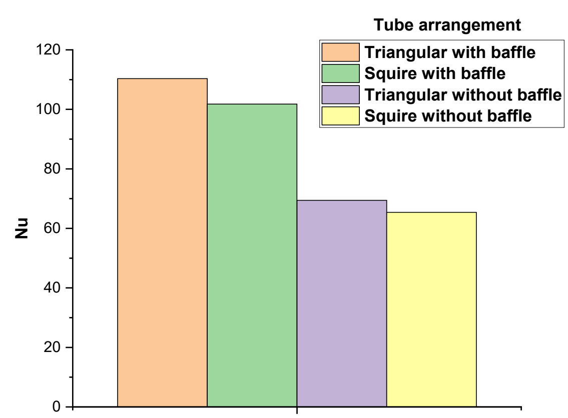


Figure 10: Nusselt number for different sets of models.

Table 3: Nomenclature

	Nomenclature
	Specific heat ()
K	Thermal conductivity(W/mK)
Nu	Nusselt number (---)
ν	Kinematic viscosity ()
Pr	Prandtl number
ρ	Density ()
ṁ	Mass flow rate (kg/s)
	Viscosity (

tube arrangement with baffle was 110.64 and for squire was 101.74. Nusselt number was detected very low when there was baffle used. The corresponding value of Nusselt number were found to be 69.45 and 65.38 for without using baffle.

The Nu for baffle attached with shell was 58.87% higher than the without baffle for triangular tube arrangement. And for squire arrangement of tube, the enhancement in Nu was 55.61%. The result shows that the more ratio of convection to the conduction heat transfer was found in the triangular arrangement of tube than the squire arrangement.

Conclusions

The important conclusions emerged out from the present study are as follows:

- The pressure developed in the baffle attached was 1.59 times and 1.49 times the without baffle for triangular tube and squire arrangement respectively.
- The cold fluid attained a maximum outlet temperature of 290.45K and the hot fluid dropped the minimum outlet temperature to 306.15K.
- The Nu for baffle attached shell was 58.87% and 55.61% higher than the without baffle for triangular and squire tube arrangement respectively.
- In this study, the best result is achieved in the triangular arrangement of tubes with baffles as its desired outcomes have more than other arrangements. The triangular arrangement creates a higher resistance factor and fluid flowing through the shell losses energy more quickly than the squire arrangement of the tube.

Reference

[1] Kumar, S., A. Jain, Bikash Mohanty, and S. C. Gupta. "Recirculation model of kettle reboiler." *International journal of heat and mass transfer* 46, no. 15 (2003): 2899-2909.

[2] SCHÜLLER, RB, and K. CORNWELL. "A THEORETICAL MODEL FOR RECIRCULATING FLOW IN A BOILER." *Chemical Engineering Communications* 13, no. 4-6 (1982): 271-287.

[3] Zhang, Mingkan, Kashif Nawaz, Cheng-Min Yang, Matthew Sandlin, William Asher, Brian Fricke, and Anthony Gehl. "A Numerical Study on the Pool Boiling with Foam Surface Enhancement Using Different Refrigerants." (2021).

[4] Pezo, Milada, Vladimir D. Stevanovic, and Zarko Stevanovic. "A two-dimensional model of the kettle reboiler shell side thermal-hydraulics." *International Journal of Heat and Mass Transfer* 49, no. 7-8 (2006): 1214-1224.

[5] McNeil, David A., Khalid Bamardouf, Bryce M. Burnside, and Mohammed Almeshaal. "Investigation of flow phenomena in a kettle reboiler." *International Journal of Heat and Mass Transfer* 53, no. 5-6 (2010): 836-848.

[6] McNeil, D. A., K. Bamardouf, and B. M. Burnside. "Two-dimensional flow modelling of a thin slice kettle reboiler." *International journal of heat and mass transfer* 54, no. 9-10 (2011): 1907-1923.

[7] Ni, Tong-Wei, Jing-Lu Fei, Sheng-Hui Wang, Yi Gong, and Zhen-Guo Yang. "Failure analysis on unexpected perforation of heat exchanger tube in methacrylic acid reboiler of specialty chemical plant." *Engineering Failure Analysis* 108 (2020): 104267.

[8] Saad, Muhammad, Altamash Munir, and Muhammad Ali Kamran. "Numerical Simulations of Shell and Tube Heat Exchanger with Segmental, Trefoil and Segmented Trefoil Baffles for Performance Comparison." *Heat Transfer Engineering* (2022): 1-18.

[9] Miyazaki, Takeru, Misaki Baba, Hideki Murakawa, Hitoshi Asano, Katsumi Sugimoto, and Daisuke Ito. "Two-Phase Flow Behavior and Heat Transfer Characteristics in Kettle Reboiler." In *ASME Power Conference*, vol. 57601, p. V001T05A011. American Society of Mechanical Engineers, 2017.

[10] Maslovaric, Blazenka, Vladimir D. Stevanovic, and Sanja Milivojevic. "Numerical simulation of two-dimensional kettle reboiler shell side thermal–hydraulics with swell level and liquid mass inventory prediction." *International Journal of Heat and Mass Transfer* 75 (2014): 109-121.

[11] Singh, Gurbir, and Hemant Kumar. "Computational fluid dynamics analysis of shell and tube heat exchanger." *Journal of Civil Engineering and Environmental Technology* 1, no. 3 (2014): 66-70.

[12] Anand, R. S., Shibin David, M. Gajendiran, and Kingston Stanley. "Design and analysis of shell and tube heat exchanger for low temperature applications using CFD." In *IOP Conference Series: Materials Science and Engineering*, vol. 912, no. 4, p. 042056. IOP Publishing, 2020.

[13] Amini, Reza, Mohsen Amini, Alireza Jafarinia, and Mehdi Kashfi. "Numerical investigation on effects of using segmented and helical tube fins on thermal performance and efficiency of a shell and tube heat exchanger." *Applied Thermal Engineering* 138 (2018): 750-760.

[14] Arani, Ali Akbar Abbasian, and Hamed Uosofvand. "Double-pass shell-and-tube heat exchanger performance enhancement with new combined baffle and elliptical tube bundle arrangement." *International Journal of Thermal Sciences* 167 (2021): 106999.

[15] Kothandaraman, C. P. *Heat and mass transfer data book*. New age international, 2004.