Journal of Mines, Metals and Fuels, *71(11): 2132-2142; 2023. DOI: 10.18311/jmmf/2023/36043*

Print ISSN : 0022-2755

Journal of Mines, Metals and Fuels

Contents available at: *www.informaticsjournals.com/index.php/jmmf*

Enhancing Heat Transfer Efficiency: A Numerical Investigation of Shell and Tube Heat Exchanger with Variable Angular Baffles for Minimizing Overall Fuel Consumption

Subhendu Bhattacharyay[1], Kalyan Mitra[2], Prasun Dutta[3]*, Soodipa Chakraborty[3], Himadri Chattopadhyay[4]

[1]*Research Scholar, Indian Maritime University, Kolkata - 700088, India*
[2]*Professor, Indian Maritime University, Kolkata - 700088, India*
[3]*Assistant Profesor, Adamas University, Kolkata - 700126, India; pd.iiest@gmail.com*
[4]*Profesor, Jadavpur University, Kolkata - 700032, India*

Abstract

This study presents a comprehensive numerical investigation on the enhancement of heat transfer efficiency in a shell and tube heat exchanger employing variable angular baffles. The research focuses on minimizing overall fuel consumption through the optimization of heat exchange processes. Various angular configurations of baffles are systematically analyzed to identify the most effective design for improving thermal performance. ANSYS Fluent, a commercial CFD code, resolves shell and tube flow and temperature fields using the finite volume approach. By solving RANS equations, k-ε turbulence models resolve turbulent flow fields. Before running simulations for the current investigation, mesh independency studies cross-check numerical models for grid dependency and validate them with earlier studies. The numerical simulations reveal insights into the impact of different baffle arrangements on heat transfer characteristics, facilitating the development of energy-efficient heat exchangers. The findings contribute valuable knowledge to the field of thermal engineering, offering potential solutions for reducing overall fuel consumption in industrial applications.

Keywords: *k - ε Turbulence Model, Numerical Study, Optimization in Fuel Consumption, Segmental Baffle, Shell and Tube Heat Exchanger*

1.0 Introduction

Fresh water production from sea water for domestic and auxiliary purposes is an essential requirement aboard ships. For generating fresh water, sea water condenser and jacket water evaporator is used. Both are types of heat exchangers. Heat exchangers are, by definition, devices or pieces of equipment in which heat is transferred between mediums of varying temperatures. During this transfer, heat is removed from the medium that is relatively hot, and the medium that is relatively cold gets heat. The influence of the baffle quantity on the average Nusselt number was investigated by means of numerical analysis, which was used to the study of transverse mixed convection[1]. According to the findings, the kind of arrangement as well as the total number of baffles have an influence on the degree to which the average Nusselt number and temperature depart from their respective

*Author for correspondence

values. The laminar flow in enforced convection that takes place through a rectangular conduit that contains a baffle near to the bottom wall was the subject of a computer study that was conducted by Benzenine *et al.*[2]. They investigated the effects of putting in a perforated baffle with a range of different amounts of perforation and looked at the results. The data included in the report includes temperature readings as well as contours of velocity for the two distinct planes that intersect in the middle of the baffle. According to what has been found, the values of the average friction coefficient, the average heat flux, and the average Nusselt number all change depending on the volume of baffle holes and the Reynolds number. In order to do statistical analysis on channels that were equivalent to one another, Yuan *et al.*[3] used disturbance rods rather than fins. It was determined how the distance between the tubes influenced the total quantity of heat that was transported. The Nusselt number of a channel with rough walls can climb to be four times greater than that of a channel with smooth walls under the same conditions; nevertheless, the pressure drop will be significantly larger. It is known knowledge that tubes that are situated in a more immediate proximity to the shell have a more substantial impact on the rate at which heat is transferred. In keeping with this, an additional investigation was carried out on the three-dimensional transmission of heat in a channel that had a simple input zone baffle[4]. According to the findings of a numerical simulation of the thermal behavior of a fluid streaming through a conduit with rows of baffles, the baffle is only partially effective for high blocking report values[5]. An investigation was out in 2005 yielded these results, which were discovered there. A numerical investigation of a turbulent flow field was carried out by Chang and Mills[6] in a conduit that was fitted with segmented baffles. They made the observation that there is a correlation between the change in baffle height and the friction factor. Since shell and tube heat exchangers are so often employed in industrial applications, it is of the highest importance to conduct analyses of fluid motion in cyclic or annular geometries.

Because of the enduring nature of its design, shell and tube heat exchangers are among the most commonly used in a variety of applications. More over 35 percent of all heat exchangers in the world are shell and tube types[7,8]. Shell and tube heat exchangers are also known as shell and tube heat pumps. In their study, Saffarian *et al.*[9] made use of a shell and tube heat exchanger that had a baffle cut that was 25% of its whole area. There were circular and elliptical baffles in the test tubes, each having a cross-section that had an attack angle of ninety degrees. Within the scope of their investigation, a collaborative model of a tube and shell heat exchanger with circular and elliptical tubes and an attack angle of ninety degrees was presented. It has been shown that tubes located closer to the surface of the shell have a greater impact on the pace at which heat is transferred than tubes located deeper into the shell. Tubes and baffles are the two primary components of the design of segmental baffled tube and shell heat exchangers, which have been widely researched and put into use. These heat exchangers have the potential to have an effect on the performance of the heat exchanger. Increasing the rate of heat transfer may be accomplished by a variety of methods, some of which include the installation of baffles inside the flow area, as well as the inclination of the walls or the alteration of the geometrical properties of the walls[10]. Numerous studies, numbering in the tens, have been conducted to investigate the efficiency of the baffled system. Cucumo *et al.*[11] observed that increasing the helix angle increased the heat transfer coefficient and lowered the pressure drop across a variety of segmental and helical baffle types with helix angles of 7°, 20°, 30°, and 40°. This was discovered by a comparative analysis of several baffle types with helix angles of 7, 20, 30, and 40 degrees. The performance of permeable baffles in a conduit is superior to that of solid baffles, according to a comparison of compact and permeable baffles in the conduit12. The existence of obstacles, which not only increase turbulence and heat transmission[12–15], but also make it feasible to distribute the fluid flow in a healthy manner, makes this possibility achievable. Patankar *et al.*[16] were the ones who initially proposed the concept of fully evolved periodic flow. The researchers Kelkar and Patankar[17] found that when the Reynolds number increased, the Nusselt number and friction factor also increased. The investigation that Cheng and Huang[18] carried out on the phenomenon of overlapping baffles indicated that one of the factors that might influence the flow field is the relative positioning of the baffle rows. According to the findings of an experimental investigation[19], an increased level of heat transfer intensity may be achieved by using a conduit that is equipped with inclined baffles and a turbulent flow.

According to the findings of yet another study on shell and tube heat exchangers[20], increasing the baffle spacing leads to an increase in both the heat transfer coefficient and the friction factor. In a heat exchanger, De *et al.*[21] examined the helix angles of 10 degrees, 16 degrees, 22 degrees, and 28 degrees and found that a top tilting of 12 degrees boosted the heat transfer coefficient in the cylinders. This coefficient increases as the speed of the heat exchanger increases.

The Nusselt number increases with the Reynolds number, low- and high-pressure regions are connected to recirculation zones, and smaller baffle spacing leads in increased heat transfer, according to the findings of another study that investigated the connection between baffle spacing and pressure drop[22]. Jadav and Koli[23] conducted a mathematical investigation of the relationship between baffle spacing and cut and how it affects the rate of heat transfer. They contrasted their findings with those of Delaware. Method[24] was used to simulate a number of various flow rates, and then baffle cut designs were tested. Patil and Bhalkikar[25] studied the influence of baffle cut on heat transfer coefficient and pressure drop. They discovered that 30% and 25% baffle reductions lower pressure drops as well as heat transfer coefficients. Patil and Bhalkikar[25] also observed that baffle cuts reduce heat transfer coefficients. Several studies came to the conclusion that an increase in heat transmission may be achieved by using a 25% baffle cut with an appropriate distance between the baffles[26,27]. They also argued, based on the results of a simulation of a shell and tube heat exchanger with baffles, that the spacing between the baffles has no influence on the temperature at the exit; nevertheless, the mass flow velocity does have a significant impact. In the case of a heat exchanger with a rod type and square pitch bundle construction, two more recent studies[20,28] shown that the value of Nu increases in tandem with Re on the tube side. In the case of a shell and tube heat exchanger with a trefoil aperture for creating cross flow, a few experiments[29,30] shown that an expanded heat transfer area increases the rate of heat transfer as well. According to the findings of other tests, the rate of increase in pressure drop is significantly higher than the rate of increase in heat transfer coefficient[31]. Because it is simple to produce and also very inexpensive, the tube and shell heat exchanger with baffles with an angle of attack of ninety degrees is the type of heat exchanger that is most commonly employed in many different industries these days. Simultaneously, the performance of the baffles at different angles is also required to be studied in terms of the improved heat transmission, which is the primary purpose of the current study.

2.0 Problem Description

2.1 Geometry and Details

The model is a shell with a 90 mm diameter (D_s) and a 20 mm diameter (d) circular cross section near the inlet as well as outlet. The shell's length is presumed to be 900 mm. Triangular Tube Bundle of Pitch (P) = 16mm and thickness 2mm total and 7 tubes are also taken into consideration. The central baffle spacing (B) of the shell's two segmental baffles is 86 mm. For the present study, three different angular position of baffles 45 and 135 respectively towards the direction of flow are considered keeping other parameters constant to observe the effect on heat transfer due to different angular position of baffles (Figure 1, Table 1-2).

Table 1. Geometry details

Diameter of the Shell	D_S = 90 mm
Diameter of the Tube	d = 20 mm
Pitch and Geometry of Tube Nest	Trilateral, P = 30 mm
Total tube numbers	Nt=7
Span of the Tube	L= 600 mm
Baffle cut	Bc =36%
Total Baffles	Nb = 2
Baffle Spacing	B= 86 mm
Angular Position 0f Baffles	45

Table 2. Properties of material

	Density [kg/m^3]	Specific Heat [kg/m^3]	Thermal Conductivity [w/m°K]
Water	989.2	4182 0.6	0.6
Aluminum	2719	871	202 .4

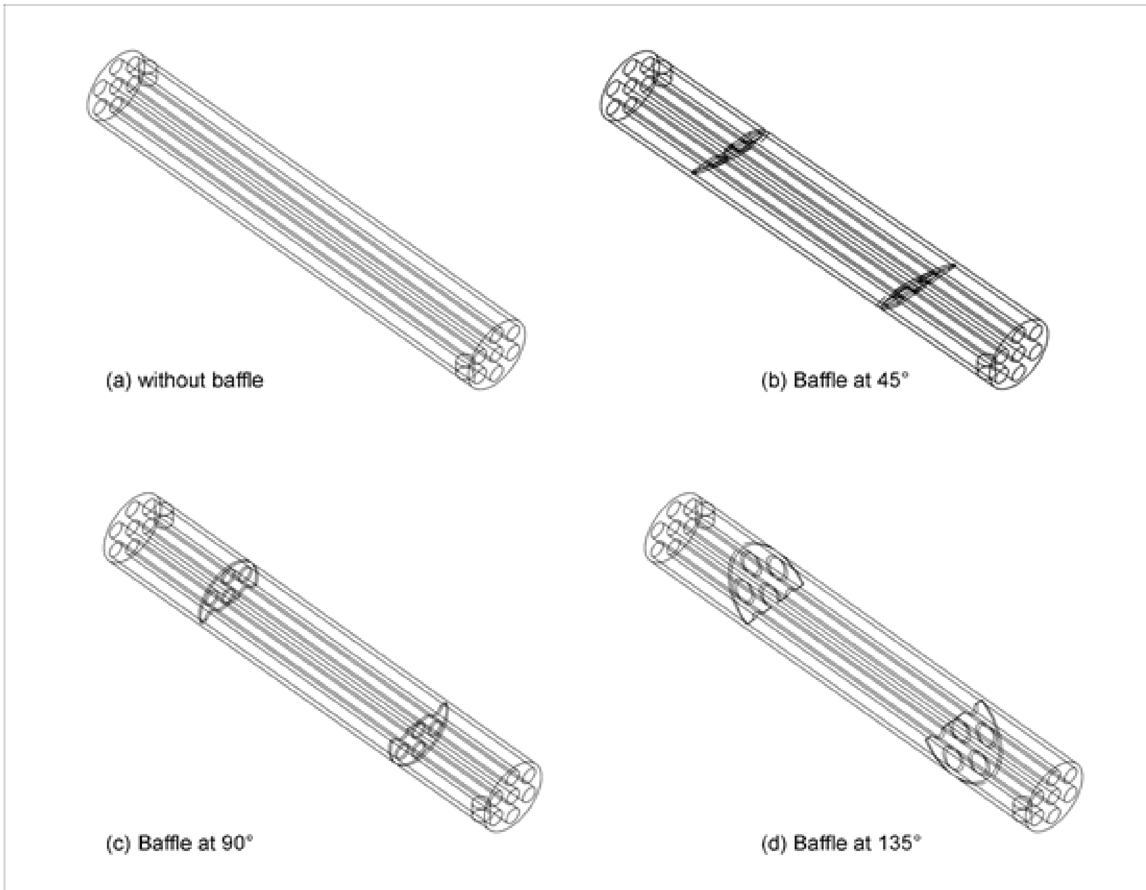

Figure 1. Flow domain with deiffernt baffle condition (flow direction: from upper left to down right side).

2.2 Governing Equation

Equations (1) and (2), with standard nomenclature, show the governing equations of mass and the momentum for the computation of incompressible flow in the heat exchanger

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

$$\rho\left(u_i \frac{\partial u_j}{\partial x_i}\right) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_i}\left[(\mu + \mu_t)\frac{\partial u_i}{\partial x_i}\right], j = 1,2,3 \tag{2}$$

$$\left(u_i \frac{\partial \theta}{\partial x_i}\right) = \frac{\partial}{\partial x_i}\left[\left(\alpha + \frac{\upsilon_t}{Pr_t}\right)\left(\frac{\partial \theta}{\partial x_i}\right)\right] \tag{3}$$

An expression for the turbulent viscosity (μ_t) is required for the closure of the aforementioned equations. The k–ε model, a member of the family of two equation models, is used in this work. To determine its value, which is provided by,

$$\mu_t = C_\mu \frac{k^2}{\varepsilon} \tag{4}$$

Where C_μ is a constant and has been considered in an empirical manner. The current study employs the k-ε turbulence model, which has been shown to perform better in determining turbulent flows. Transport equations for the k-ε model[32–36] is illustrated in Eqs. (4) and (5), the equations also contain adjustable constants.

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + P_k - \rho\varepsilon \tag{5}$$

$$\frac{\partial(\rho \varepsilon)}{\partial t} + \frac{\partial(\rho \varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}P_k - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \tag{6}$$

Where C_μ=0.09; σ_k = 1.00; σ_ε= 1.30; $C_{1\varepsilon}$= 1.44; $C_{2\varepsilon}$= 1.92, and P_k implies the production rate of turbulence kinetic energy and calculated as $P_k = \mu_t \bar{S}^2$ where the time-averaged strain rate tensor (S) is supplementary considered as: $S = \sqrt{2\bar{S}_{ij}\bar{S}_{ij}}$, where $\overline{S_{ij}} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)$.

2.3 Boundary Condition

In this present study, at both inlet of shell and tubes, a same constant velocity profile keeping same Reynolds number according to the mass flow rate with constant different temperature are applied as the Dirichlet condition. The pipe wall was presumed to be in a "no-slip" condition with coupled thermal conditions. At the outlet of both the shell and tubes, the Pressure Outlet condition was presented. Particulars are specified in Table 3.

2.4 Mesh Dependency Analysis and Validation

For the discretization of model, an amorphous mesh is produced with mostly hexahedral elements in the both shell and tubes domain (Figure 2). Close to the walls, where there is a good chance that there will be significant gradients, fine grids are utilized. To certify the individuality of the results regarding the chosen mesh, several tests were conducted. Table 4 displays the results of the three best scenarios, with 835000 cells, 3352587 cells, and 4526000 cells respectively, for the both temperature and the velocity in the middle portion (X = 0.045m).

The findings above demonstrate that the dissimilarity is extremely minor and does not surpass 1%. The scrutiny of the data reveals that the selection of (45,26,000) elements, for the current numerical model, gives a good prediction.

Before proceeding, the model is checked against the previously published data34 for both baffled and unbaffled instances; the results of this validation are summarized in Table 5. It is detected that inclusive Temperature of the

Table 3. Boundary condition

Position	Mass flow rate	Temperature
Shell Inlet Side	$0.5m^3/sec$	300 K
Tube Inlet Side	$0.5m^3/sec$	450 K
Shell Outlet Side	Pressure Outlet	Pressure Outlet
Tube Outlet Side	Pressure Outlet	Pressure Outlet
Wall	No Slip condition	Coupled condition

Table 4. Mesh dependency analysis

Mesh size	Velocity (m/s)	Temperature (K)
2835000	2.0346	333.1256
3352587	2.0434	336.2562
4526000	2.0448	336.3215

Table 5. validation of present model

Parameter	Without baffles		With baffles (at 90°)	
Temparature (K)	Youce and Saim[34]	Present analysis	Youce and Saim[34]	Present analysis
	322.51	325.82	325.15	328.86

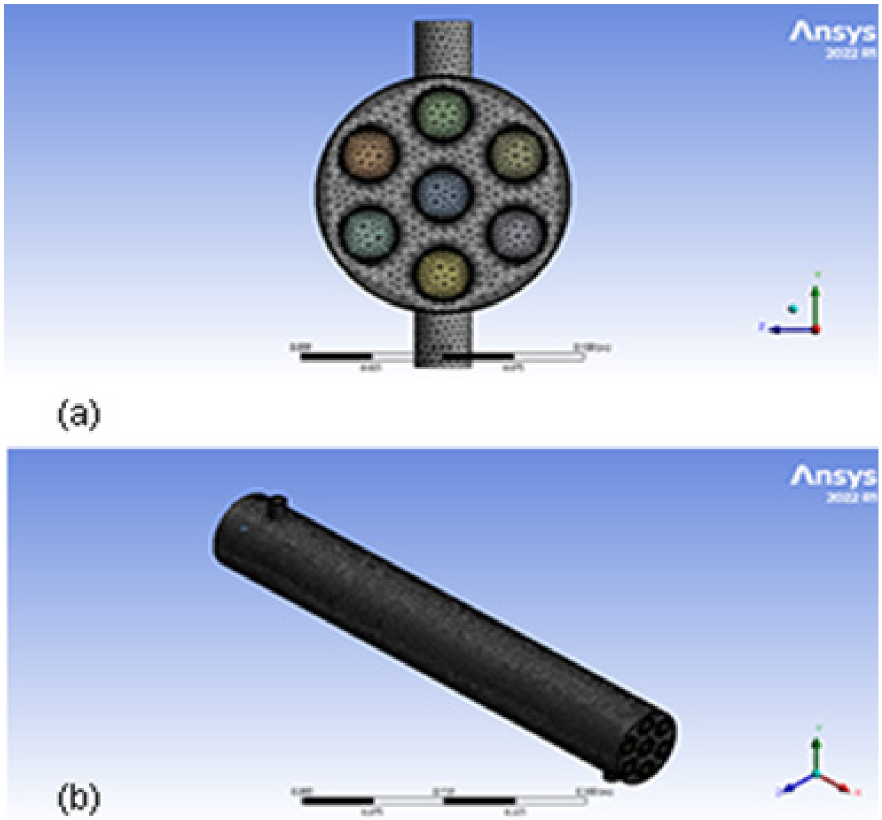

Figure 2. Mesh configuration for this study.

fluid are almost similar parallels to the outcomes of the literature and the variance between the two cases is less than 3%. This agreement between the output of the present model and the previous result is generally satisfactory for present simulations; hereafter, this technique of mesh creation and setup for numerical simulation is used for further investigation.

3.0 Result and Discussions

3.1 Flow Field Analysis

This section presents and discusses the hydrodynamic features of the flow-field of the current study. The characteristic features include the streamline plots. This will help in comprehending the general flow physics of the shell side flow for the arrangements of segmented baffles at different angles in this study.

In Figure 3, the basic nature of the flow is depicted for mass flow rate of 0.5 m^3/s, as obtained from the present simulations. The flow fields are presented in-terms of path lines in the flow domain at central cross section for different baffle configurations. For the very 1st case of the present study (*i.e*, without baffle case), cold fluid enters through the shell nozzle and impinges over the tubes which contains the flow of hot fluid having same mass flowrate. After that, the cold fluid travels through the heat exchanger similar to a parallel flow heat exchanger principle and eventually the cold fluid becomes hot at the outlet of the shell due to the both conductive and competitive heat transfer phenomenon. Whereas in case of 2nd case of present study (baffle at 45° to the flow direction), strong recirculation zone is clearly observed just before and after of the baffles. These recirculation zones increase the local turbulence of the flow field with proper mixing of the flow. This phenomenon can be observed for other two cases also (i.e, baffle with 90° and 135°). The only difference which is observed that, the angles of baffles made a significant contribution to the development of the level of turbulence in the flow field, which is the evident of difference of heat transfer rate and can clearly be seen in the contour temperature distribution plots.

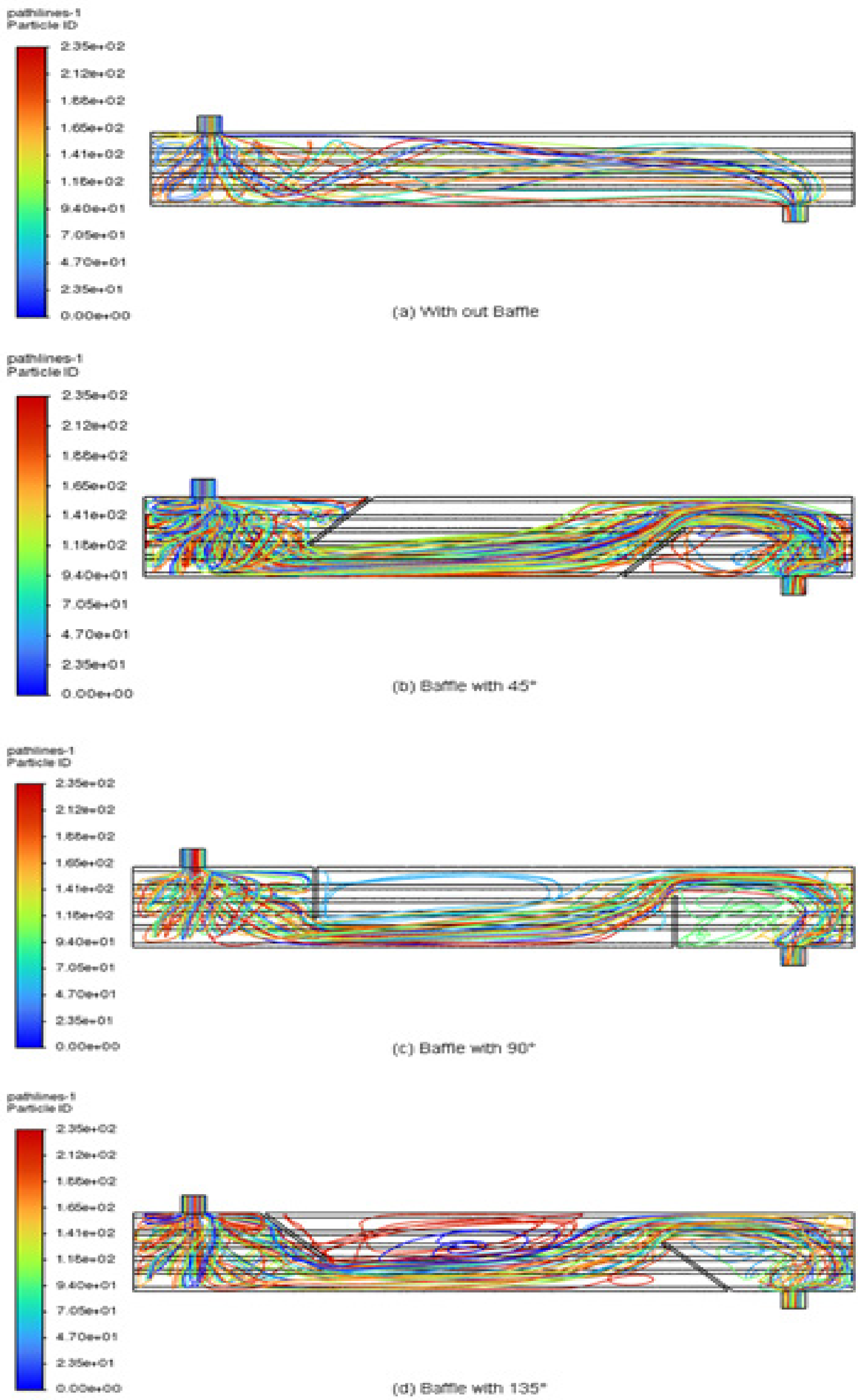

Figure 3. Path lines in the flow domain at central cross section.

3.2 Thermal Field Analysis

Figure 4, displays the temperature distributions in the flow field at the central cross-section for different baffle configurations. For without baffle case, as the cold fluid travels from the inlet to the outlet of the shell, it heated up due to the regular parallel flow heat exchanger principle by both conductive and convective heat transfer process. As the baffles are included in the domain, due to the increases in local turbulence after the baffle position, a local temperature rise is observed in the figures showing the enrichment of the heat transfer proportion. Which finally results in a rise in temperature at the outlet of the shell nozzle.

Table 6 and 7 show a comparative study of both rise of dimensional temperature and heat transfer in terms of non-dimensional Nusselt number. The Nusselt number (Nu), which is calculated from the local temperature gradient as:

$Nu = -\left(\frac{\partial \theta}{\partial y}\right)$, where θ is the nondimensional temperature[37–39]. To obtain the global Nusselt number, the Nusselt number distribution can be averaged over the surface area as: $Nu_{avg} = \frac{1}{A}\iint Nu(y,z)dydz.$

From the above table, it is clearly observed that the baffle with 45° orientation with the flow direction case shows the optimum performance among all other cases in terms of enhancement of heat transfer showing a maximum increase in the rise of temperature.

4.0 Conclusion

The impact of baffles on the stream field and heat transmission in a parallel flow heat exchanger were quantitatively examined in this work at four different

Table 6. Result on change in Temperature

Case	Shell Inlet Temperature (K)	Shell Outlet Temperature (K)	% of change in Temperature
Without Baffle	300	336.15	12.05
Baffle at 45°	300	343.15	14.38
Baffle at 90°	300	341.88	13.96
Baffle at 135	300	342.62	14.21

Table 7. Result on change in $Nu_{averaged}$

Case	Area averaged Nu	% of change in Nu averaged
Without Baffle	48.56	-
Baffle at 45°	65.62	17.06
Baffle at 90°	61.89	13.33
Baffle at 135	63.32	14.76

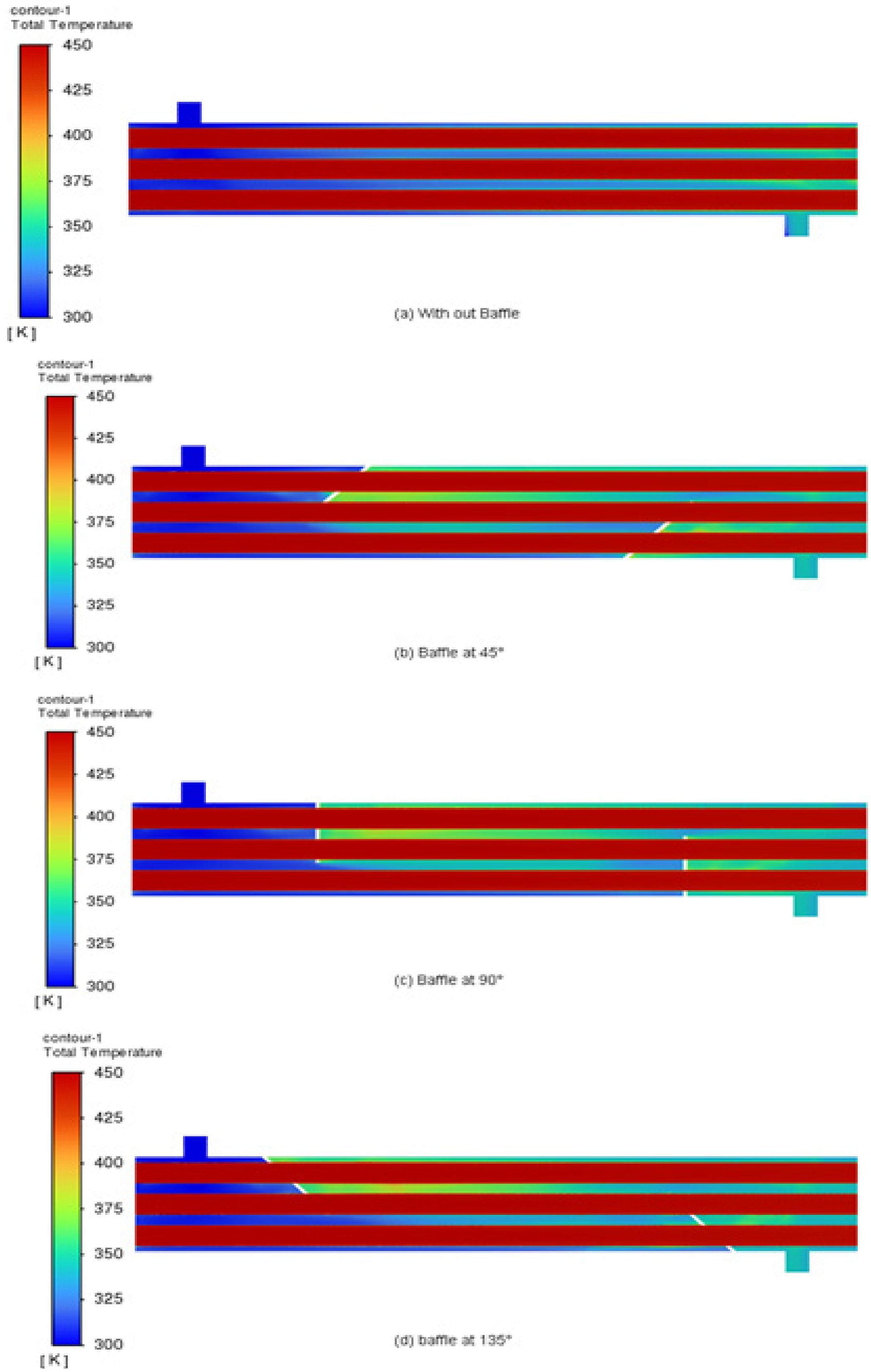

Figure 4. Contour plots of Temperature in the flow domain.

baffle orientations (*i.e*, without baffle, baffles at 45°, 90°, 135°). The detailed stream structures and heat transfer features are presented for a particular mass flow rate condition (*i.e*, 0.5 m^3/s). The results revealed a complex flow structure in the flow domain for the inclusion of baffles. A comparative study was also conducted to check the performance of the baffle angles in terms of heat transfer enrichment. It is found that the baffle with 45° orientation with the flow direction case shows the optimum performance among all other cases in terms of enrichment of heat transfer showing a maximum increase in the rise of temperature. A rigorous study is further needed covering all flow regimes (laminar, transient, and turbulent) to develop correlations between heat transfer characteristics and flow parameters to understand the complete thermal scenario on the impact of baffle angle.

5.0 References

1. Belmiloud MA, Chemloul NES. Effect of baffle number on mixed convection within a ventilated cavity. J Mech Sci Technol. 2015 Nov; 29(11):4719–27.
2. Benzenine H, Saim R, Abboudi S, Imine O, Oztop HF, Abu-Hamdeh N. Numerical study of a three-dimensional forced laminar flow in a channel equipped with a perforated baffle. Numer Heat Transf A. 2018 Jun; 73(12):881–94.
3. Yuan ZX, Tao WQ, Wang QW. Numerical prediction for laminar forced convection heat transfer in parallel-plate channels with streamwise-periodic rod disturbances. Int J Numer Methods Fluids. 1998 Dec; 28(9):1371–87.
4. Guo Z, Anand NK. Three-dimensional heat transfer in a channel with a baffle in the entrance region. Numer Heat Transf A. 1997 Jan; 31(1):21–35.
5. Bazdidi-Tehrani F, Naderi-Abadi M. Numerical analysis of laminar heat transfer in entrance region of a horizontal channel with transverse fins. Int Commun Heat Mass Transf. 2004 Mar; 31(2):211–20.
6. Chang BH, Mills AF. Turbulent flow in a channel with transverse rib heat transfer augmentation. Int J Heat Mass Transf. 1993 Jan; 36(6):1459–69.
7. Pan M, Jamaliniya S, Smith R, Bulatov I, Gough M, Higley T, *et al*. New insights to implement heat transfer intensification for shell and tube heat exchangers. Energy (Oxf). 2013 Aug; 57:208–21.
8. Master BI, Chunangad KS, Pushpanathan V. Fouling mitigation using helixchanger heat exchangers. 2003.
9. Saffarian MR, Fazelpour F, Sham M. Numerical study of shell and tube heat exchanger with different cross-section tubes and combined tubes. Int J Energy Environ Eng. 2019 Mar; 10(1):33–46.
10. Aslam Bhutta MM, Hayat N, Bashir MH, Khan AR, Ahmad KN, Khan S. CFD applications in various heat exchangers design: A review. Appl Therm Eng. 2012 Jan; 32(1):1–12.
11. Cucumo M, Ferraro V, Kaliakatsos D, Mele M, Galloro A, Schimio R, *et al*. Thermohydraulic analysis of a shell-and-tube ``helical baffles” heat exchanger. Heat Technol. 2016 Oct; 34(S2):S255–S262.
12. Yang YT, Hwang CZ. Calculation of turbulent flow and heat transfer in a porous-baffled channel. Int J Heat Mass Transf. 2003 Feb; 46(5):771–80.
13. Youcef-Ali S. Study and optimization of the thermal performances of the offset rectangular plate fin absorber plates, with various glazing. Renew Energy. 2005 Feb; 30(2):271–80.
14. Youcef-Ali S, Desmons JY. Numerical and experimental study of a solar equipped with offset rectangular plate fin absorber plate. Renew Energy. 2006 Oct; 31(13):2063–75.
15. Moummi N, Youcef-Ali S, Moummi A, Desmons JY. Energy analysis of a solar air collector with rows of fins. Renew Energy. 2004 Oct; 29(13):2053–64.
16. Patankar S V, Liu CH, Sparrow EM. Fully developed flow and heat transfer in ducts having streamwise-periodic variations of cross-sectional area. J Heat Transf. 1977 May; 99(2):180–6.
17. Kelkar KM, Patankar S V. Numerical prediction of flow and heat transfer in a parallel plate channel with staggered fins. J Heat Transf. 1987 Feb; 109(1):25–30.
18. Chin-Hsiang C, Wen-Hsiung H. Numerical prediction for laminar forced convection in parallel-plate channels with transverse fin arrays. Int J Heat Mass Transf. 1991 Nov; 34(11):2739–49.
19. Dutta P, Hossain A. Internal cooling augmentation in rectangular channel using two inclined baffles. Int J Heat Fluid Flow. 2005 Apr; 26(2):223–32.
20. Li H, Kottke V. 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 Transf. 1998 May; 41(10):1303–11.
21. De D, Pal TK, Bandyopadhyay S. Helical baffle design in shell and tube type heat exchanger with CFD analysis. Int J heat Technol. 2017; 35(2):378–83.

22. Saim R, Benzenine H, Öztop HF, Al-Salem K. Turbulent flow and heat transfer enhancement of forced convection over heated baffles in a channel. Int J Numer Methods Heat Fluid Flow. 2013 May; 23(4):613–33.
23. Jadhav AD, Koli TA. CFD Analysis of Shell and Tube Heat Exchanger to Study the Effect of Baffle Cut on the Pressure Drop. Int J Res Aeronaut Mech Eng. 2014; 2(7):1–7.
24. Bell KJ. Heat exchanger design for the process industries. J Heat Transf. 2004; 126(6):877–85.
25. Patil CN, Bhalkikar NS. CFD analysis of shell and tube heat exchanger to study the effect of baffle cut on the pressure drop and heat transfer coefficient. Int J Eng Sci & Res Technol. 2014; 2(5):649–54.
26. Kiran K, Umesh CJ, others. Investigation of baffle spacing effect on shell side heat transfer characteristics in shell and tube heat exchanger using computational fluid dynamics. Elixir Therm Eng. 2014; 73:26022–6.
27. Prasanna J, Purushothama HR, Murugeshan DK. A numerical analysis of hydrodynamic and heat transfer effects of shell and tube heat exchanger for different baffle space and cut. Mech Confab. 2013; 2.
28. Rao JBB, Raju VR. Numerical and heat transfer analysis of shell and tube heat exchanger with circular and elliptical tubes. Int J Mech Mater Eng. 2016 Dec; 11(1).
29. You Y, Chen Y, Xie M, Luo X, Jiao L, Huang S. Numerical simulation and performance improvement for a small size shell-and-tube heat exchanger with trefoil-hole baffles. Appl Therm Eng. 2015 Oct; 89:220–8.
30. Zhou GY, Xiao J, Zhu L, Wang J, Tu ST. A numerical study on the shell-side turbulent heat transfer enhancement of shell-and-tube heat exchanger with trefoil-hole baffles. Energy Procedia. 2015 Aug; 75:3174–9.
31. Pon Chit S. Flow analysis in shell side on the effect of baffle spacing of shell and tube heat exchanger. Int J Sci Technol Soc. 2015; 3(5):254.
32. Ozden E, Tari I. Shell side {CFD} analysis of a small shell-and-tube heat exchanger. Energy Convers Manag. 2010 May; 51(5):1004–14.
33. Dutta P, Chattopadhyay H, Nandi N. Numerical Studies On Turbulent Flow Field in a 90° Pipe Bend. J Fluids Eng. 2022;
34. Youcef A, Saim R. Computational analysis of turbulent flow and thermal transfer in a shell and tube heat exchanger. Heat Technol. 2019 Dec; 37(4):1043–51.
35. Dutta P, Nandi N. Numerical analysis on the development of vortex structure in 90° pipe bend. Prog Comput Fluid Dyn An Int J. 2021; 21(5):261.
36. Dutta P, Saha SK, Nandi N, Pal N. Numerical study on flow separation in 90° pipe bend under high Reynolds number by k-ε modelling. Eng Sci Technol an Int J. 2016; 19(2).
37. Dutta P, Chattopadhyay H. Numerical simulation of heat transfer due to array of laminar annular jets impinging on hot moving surface. In: Proceedings book of fifth international conference on advances in mechanical engineering: ICAME. 2019. p. 162–70.
38. Dutta P, Chattopadhyay H. Numerical analysis of transport phenomena under turbulent annular impinging jet. Comput Therm Sci Int J. 2020;
39. Dutta P, Chattopadhyay H. Heat Transfer Due to Annular Jets Impinging On a Moving Surface. J heat Transf ASME. 2022;