

Numerical Simulations For Thermal Analysis Of A Shell And Tube Heat Exchanger

Muhammad Zahid*

Department of Mathematics and Statistics, University of Southern Punjab, Multan
66000, Pakistan Email: muhammadzahid82@gmail.com

Adla Ameen

Department of Mathematics and Statistics, University of Southern Punjab, Multan
66000, Pakistan

ABSTRACT

In this study, numerical simulations are performed for a shell-and-tube heat exchanger in order to investigate the effects of mass flow rate on the heat transfer rate, outlet temperature, pressure drop, and local behavior of the fluid flow and heat transfer. The analysis is carried out for a single shell and seven tubes, one concentric with the shell, while the other six are uniformly distributed around the central tube with turbulent flow conditions. The cold fluid, with a 300 K temperature at the shell inlet, flows in the shell along the positive x-direction, while the hot fluid, with a 340 K temperature at the tubes inlet, flows in the tubes along the negative x-direction. The turbulence model used is the Standard k- ϵ model, which solves two additional equations along with the flow model comprising the three-dimensional continuity, Navier-Stokes, and the energy equations. The results obtained are quite consistent with the physics of the problem. All tasks of the simulations are carried out on ANSYS Workbench.

INTRODUCTION

Among industrial systems, the shell and tube heat exchangers are a widely used for transferring heat between two separate fluids. A cylindrical shell encloses the structured tubes, which are referred to as tube bundles. Two types of fluids flow through the shell and tubes: one moves inside the tubes while the other circulates outside. Its primary design advantage is its large surface area for heat transfer. It works very efficiently when a process requires significant temperature changes. Various-sized shells are frequently utilized in industry. Standards and design recommendations are frequently issued [1]. These methods have been continuously enhanced since the start due to rising operational data, industry experience, and instrumentation advancements. Correlation-based approaches are able to be used for both sizing and iteratively getting the general performance features (rating) of a heat exchanger. If the performance of the proposed scheme is shown to be unsatisfactory at this current iteration, the design parameters can be modified properly to create a better-performing design. A skilled heat exchanger designer knows which way to make modifications. A simple approach is to assume that the flow velocities are too low and, in the next iteration, test a design with a reduced cross-sectional flow area. Though it is quite simple to modify the tube side parameters,

it is exceptionally challenging to find an ideal one. It may be easier to determine the weaknesses and lead the designer in the right pathway. Computational fluid dynamics can be a great tool for learning that skill. Tubes, tube sheets, baffles, and tie rods are some of the components that hold a tube bundle in place.

The shell-side flow in heat exchangers is quite complex due to the presence of multiple objects. In smaller heat exchangers, the streams are typically highly restricted, leading to significant pressure drops. In this study, the model is so compact in size that any leakage and bypass streams become negligible when compared to the main flow stream. However, even with this kind of shell design, the inclusion of baffles creates a more complex flow pattern on the shell side, which affects overall performance. Baffles help directing the flow and ensure proper shell-side fluid circulation, which allows for effective utilization. In this study, the single-segmental baffle has been recognized as the most widely used baffle type. It features a cut-out section that allows fluid to flow in a parallel pattern. The baffle cut, the proportion, the gap between baffles influences the streams structure have a major effect on heat transfer and flow attributes. The baffle cut and the baffle spacing establish the best configuration for the specific shell geometry. When baffles are spaced farther apart than optimal, recirculation zones tend to form behind the baffles, and the main flow moves along to the next baffle. This reduces the effective use of the heat transfer area in those recirculation regions. Studies may be conducted how baffle spacing influences pressure drop and heat transfer by evaluating several baffle cuts values. Among the design tools, the Kern method [2] is commonly used for preliminary estimates, though it tends to be conservative. The Bell Delaware [3] procedure, in contrast, provides more reliable predictions of shell-side pressure drop and heat transfer under practical layouts. While it cannot pinpoint the exact regions where design shortcomings may appear, it does highlight the likelihood of inefficiencies in the shell-side arrangement during performance evaluation. A thorough investigation of the shell-side flow is vital for identifying the underlying reasons behind performance limitations in shell-and-tube heat exchangers. Over the time, researchers have approached this challenge using theoretical models, laboratory experiments, and computational methods, each focusing on different aspects of the system. Some studies, such as those by Gay [4] and collaborators, examined patterns of thermal transport, while others including Halle [5], Pekdemi [6], Gaddis, and Gnielinski [7] concentrated on resistance to fluid motion. Several investigations emphasized the role of the heat exchanger geometry: Li and Kottke [8], along with Karno and Ajib [9], studied the influence of tube arrangements, whereas Sparrow, Reifschneide [10], Eryener [11], and others analyzed how variations in baffle position and spacing affect both pressure losses and heat exchange efficiency. Overall, these contributions demonstrate that the configuration of baffles, particularly their cut and spacing play a decisive role in governing shell-side flow. In continuation of this understanding, the current work removes leakage and bypass flow paths in the exchanger model and applies Computational Fluid Dynamics (CFD) to systematically assess the influence of these two key parameters. Unlike empirical correlation methods, the use of CFD in the design of heat exchangers is still not widespread. Even so, CFD can be applied for both capacity estimation and operational assessment, as it reveals detailed information about fluid motion and heat transfer [12]. During the initial design stage, it reduces reliance

on repeated prototyping and assists in identifying the governing transport processes. Despite these benefits, carrying out a fully resolved numerical simulation of a large-scale exchanger is extremely resource intensive. For example, simulating a shell-and-tube system containing several hundred tubes and multiple baffles could involve on the order of hundreds of millions of computational cells, requiring vast memory and extended run times. Such complexity typically exceeds the limits of standard computing facilities. To overcome these constraints, researchers have frequently turned to simplified strategies, with porous-media models and distributed resistance methods being the most common choices for representing large exchanger geometries in a computationally efficient manner. Prithiviraj and Andrews [13] modeled shell-and-tube exchangers using a distributed resistance method, where several tubes could be represented within a single computational cell, resulting in a relatively coarse mesh on the shell side. Building on this, Sha et al. [14] developed a thermo-hydraulic framework that combined distributed resistance with effective porosity and surface permeability to describe shell-side transport characteristics. He et al. [15] used a modified porous media model and a distributed resistance approach to model three types of shell-and-tube heat exchangers. Stevanovic et al. [16] encounters analytical study of heat transfer and three-dimensional fluid flow using porous media as the baffles and the tube bundle. Two recent studies that use broad CFD models for double-pipe heat exchangers [17] but for shell-and-tube heat exchangers are not found in literature. With today's computers and software, a tiny shell can also be properly developed by accurately examining the geometry. Global facts like the pressure can be estimated by this accurate data and compared to correlation-based ones. Also, by analyzing the temperature and flow fields, the data can be used to find errors in design involving laminarization zones and recirculation.

In the current study, we have performed a parametric numerical simulation for a shell-and-tube heat exchanger in order to investigate the effects of mass flow rate on the heat transfer rate, outlet temperature, pressure drop, and local behavior of the fluid flow and heat transfer.

PROBLEM STATEMENT

A shell-and-tube heat exchanger is a device used to transfer heat between two separate fluids. A set of tubes is used for the warm fluid, while the shell is used for the cold fluid as well as the tubes are immersed in the cold fluid within the shell. In the current study, a 3D shell-and-tube design is simulated for various values of mass flow rate. There are seven tubes in the shell; one is concentric with the shell, while the other six are uniformly distributed around the central tube within the shell. The cold fluid flows along the positive x -direction, while the hot fluid flows along the negative x -direction. The design details of the heat exchanger are given in Table 1.

Online ISSN

3007-3197

Print ISSN

3007-3189

<http://amresearchreview.com/index.php/Journal/about>

Table 1: Geometry details

Parameter	Value
Shell diameter	90 mm
Tube diameter	20 mm
Tube thickness	1 mm
Cold fluid temperature	300 K
Hot fluid temperature	340 K
Number of tubes	7
Central baffle spacing	86 mm
Number of baffles	6
Baffle cut	36%
Shell length	600 mm
Inlet and outlet diameter	30 mm
Inlet mass flow rate	0.4 to 2.0 Kg/s

The computational geometry of the shell-and-tube heat exchanger is given in Figure 1.

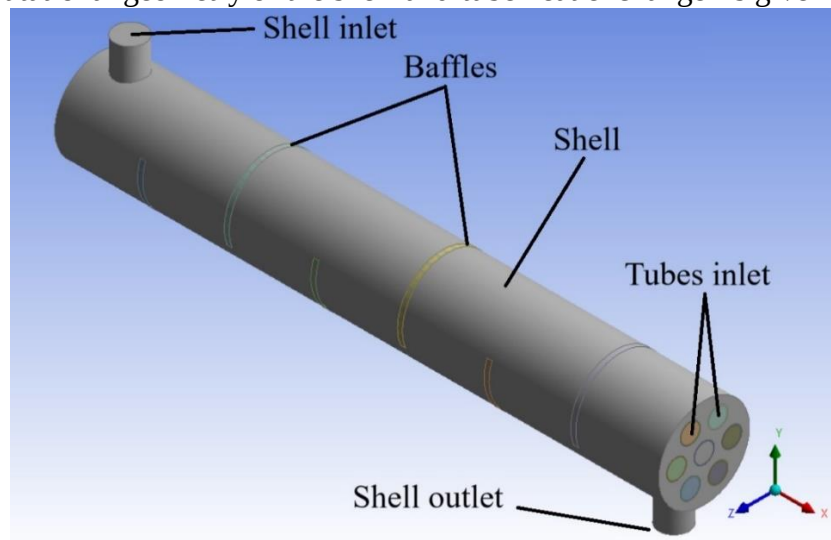


Figure 1. Computational geometry of the shell-and-tube heat exchanger.

In our work, we will perform numerical simulations for investigating the use water as a coolant passing through the shell, while the tubes are set to appropriate thermal boundary conditions. For various values of inlet mass flow rate, we will investigate the performance of the heat exchanger. In this study, the coolant is water and the material of the heat exchanger is aluminum. The properties of these materials are given in Table 2 and Table 3.

Table 2. Thermo-physical properties of the water.

Property	Value	Unit
Density	998.2	Kg/m ³
Specific heat	4182	J/(Kg-K)

Online ISSN

3007-3197

Print ISSN

3007-3189

<http://amresearchreview.com/index.php/Journal/about>

Thermal conductivity	0.6	W/(m-k)
Viscosity	0.001003	Kg/(m-s)

Table 3. Thermo-physical properties of the aluminum.

Property	Value	Unit
Density	2719	Kg/m ³
Specific heat	871	J/(Kg-K)
Thermal conductivity	202.4	W/(m-k)

MATHEMATICAL MODEL

The flow model comprises the Navier-Stokes equations along with the continuity and energy equations. The complete flow model is given as under.

Continuity equation

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0,$$

Momentum equation

$$\begin{aligned} \rho u \frac{\partial u}{\partial x} + \rho v \frac{\partial u}{\partial y} + \rho w \frac{\partial u}{\partial z} &= -\frac{\partial p}{\partial x} + \mu \left[\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right], \\ \rho u \frac{\partial v}{\partial x} + \rho v \frac{\partial v}{\partial y} + \rho w \frac{\partial v}{\partial z} &= -\frac{\partial p}{\partial y} + \mu \left[\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right], \\ \rho u \frac{\partial w}{\partial x} + \rho v \frac{\partial w}{\partial y} + \rho w \frac{\partial w}{\partial z} &= -\frac{\partial p}{\partial z} + \mu \left[\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right], \end{aligned}$$

Energy equation

$$\rho C_p \left(u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} + w \frac{\partial T}{\partial z} \right) = K_f \left[\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \right].$$

For the turbulence modeling, the Standard $k-\epsilon$ turbulence model is used, which solves the following two additional transport equations

$$\begin{aligned} \frac{\partial}{\partial x_i} (\rho k u_i) &= \frac{\partial}{\partial x_i} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_i} \right] + G_k + G_b - \rho \epsilon - Y_M + S_k, \\ \frac{\partial}{\partial x_i} (\rho \epsilon u_i) &= \frac{\partial}{\partial x_i} \left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_i} \right] + C_{1\epsilon} \frac{\epsilon}{k} (G_k + C_{3\epsilon} G_b) - C_{2\epsilon} \rho \frac{\epsilon^2}{k} + S_\epsilon, \end{aligned}$$

where $\mu_t = \rho C_\mu \frac{k^2}{\epsilon}$.

SOLUTION PROCEDURE

In the current study, a conjugate heat transfer problem is solved. The hot fluid flows in the tubes while cold fluid flows in the shell. The two fluids are separated by the tubes. A numerical solution is obtained using the finite volume method. This method is built-in in ANSYS Fluent 16.2, which is used to perform all tasks of the simulations.

Numerical Schemes

Online ISSN

3007-3197

Print ISSN

3007-3189

<http://amresearchreview.com/index.php/Journal/about>

For the solution, the SIMPLE scheme is used for the pressure-velocity coupling, the Green-Gauss Cell-Based method is used for the gradient calculations; the standard scheme is used for the pressure for enhanced accuracy; and the second-order upwind scheme is used for the momentum, energy, and turbulence. The first order upwind is used for the turbulence dissipation.

Convergence Criteria

The convergence of the numerical solution depends on the residuals of all the equations. The threshold for the residuals is set to below 10^{-3} for all the model equations, except the energy equation for which it is 10^{-6} . All the residuals are monitored using plots and console outputs to confirm that the solution remains stable and has converged.

Initialization

Before starting the calculation, the flow field is initialized by setting the reference frame relative to the cell zone and assigning initial values to all field variables. Selecting an appropriate initialization method, such as Hybrid or Standard, helps establish accurate starting conditions, enhances solver stability, and promotes faster convergence with minimal numerical errors. In the current study, we use the Standard Initialization method.

MESHING

The meshing procedure was performed using tetrahedral elements to accurately represent the intricate geometry of the shell-and-tube heat exchanger. Face sizing techniques are applied thoughtfully along curved surfaces and circular interfaces to maintain precise control over element distribution. This approach enabled a higher mesh density in key flow regions, especially near the curved boundaries, thereby enhancing simulation accuracy. Overall, the mesh quality is good enough for convergence and is well-structured to support fluid flow and heat transfer analysis in the subsequent stages. The maximum skewness of the mesh cells is 0.86652.

Online ISSN

3007-3197

Print ISSN

3007-3189

<http://amresearchreview.com/index.php/Journal/about>

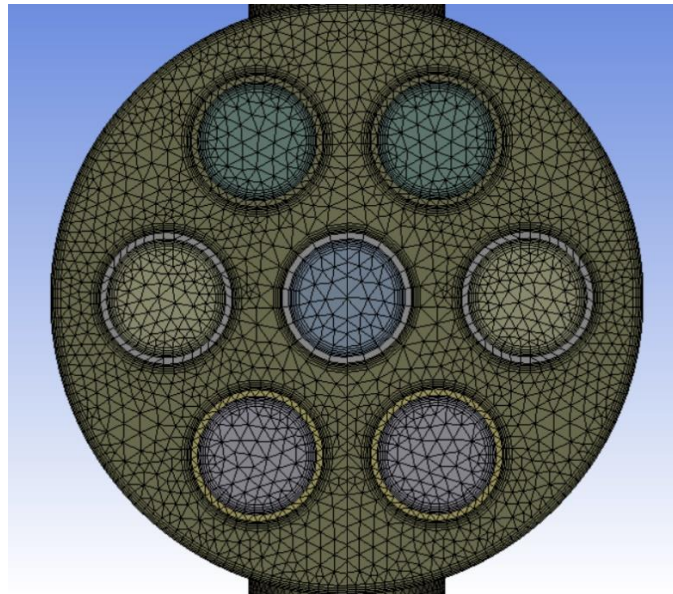


Figure 2. Cross-sectional view of mesh of the shell-and-tube heat exchanger

RESULTS AND DISCUSSION

In this section, we present an overall impact of cold fluid inlet mass flow rate on the field variables and performance measuring parameters for understanding general trends. For the local behavior of the field variables, we will present the contours analysis.

Steady Parametric Thermal and Flow Analysis

In Figure 3, the pressure is plotted against the inlet mass flow rate. It is observed that the pressure drop increases nonlinearly such that initially it rises slowly, and then the rise in pressure drop is rapid.

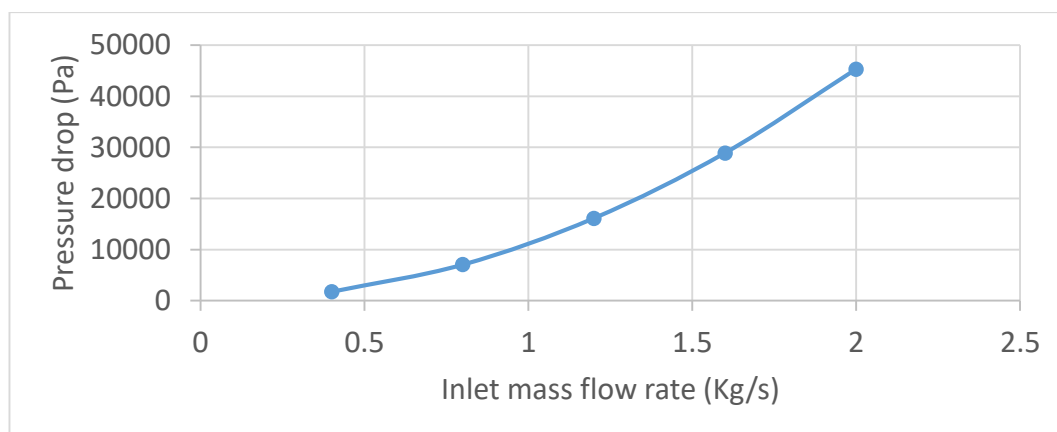


Figure 3. Pressure drop vs inlet mass flow rate.

In Figures 4 and 5, the outlet temperature at the tube outlets and tubes surfaces are plotted against the inlet mass flow rate. The fluid temperature at the tube inlet is 340 K, while it is around 322 K at the tube outlets, for 0.4 Kg/s of cold fluid mass flow rate at

Online ISSN

3007-3197

Print ISSN

3007-3189

<http://amresearchreview.com/index.php/Journal/about>

the shell inlet. This clearly shows a drastic decline in the temperature. The higher values of inlet mass flow rate do not reflect a large impact on the tube outlet temperature and the tube surface temperature.

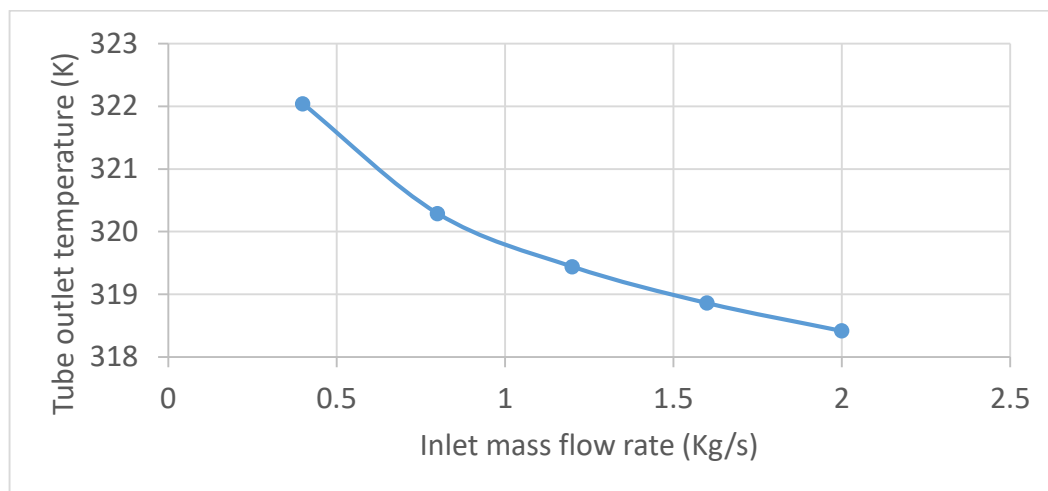


Figure 4. Tube outlet temperature vs inlet mass flow rate.

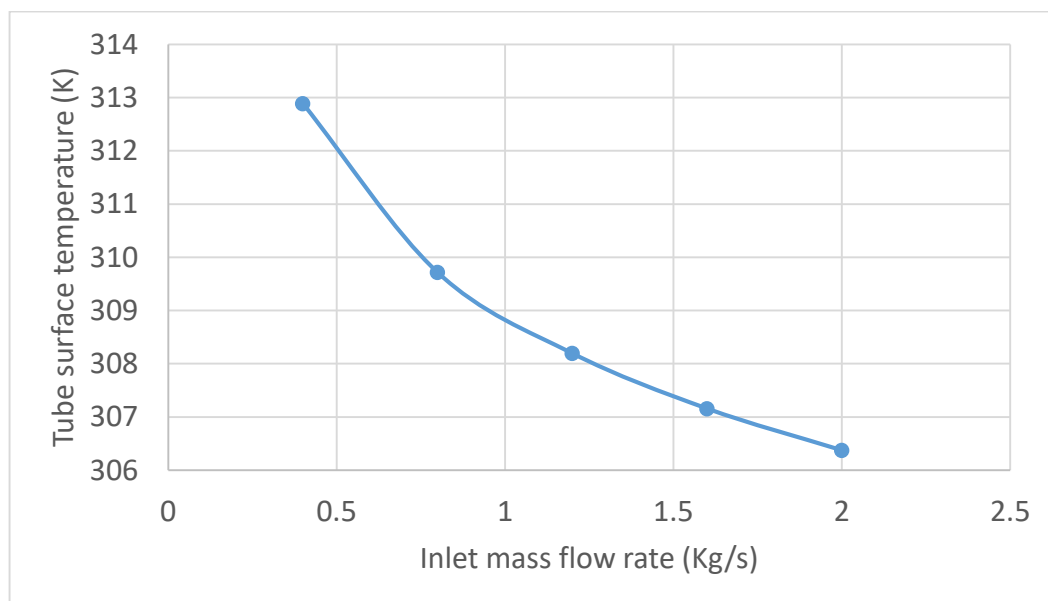


Figure 5. Tube surface temperature vs inlet mass flow rate.

In Figures 6 and 7, the Nusselt number Nu and the total heat transfer rate are plotted against the inlet mass flow rate. Both the curves initially rise rapidly and then slow down, confirming a slow improvement in the heat transfer rate against the increasing inlet mass flow rate of the shell inlet.

Online ISSN

3007-3197

Print ISSN

3007-3189

<http://amresearchreview.com/index.php/Journal/about>

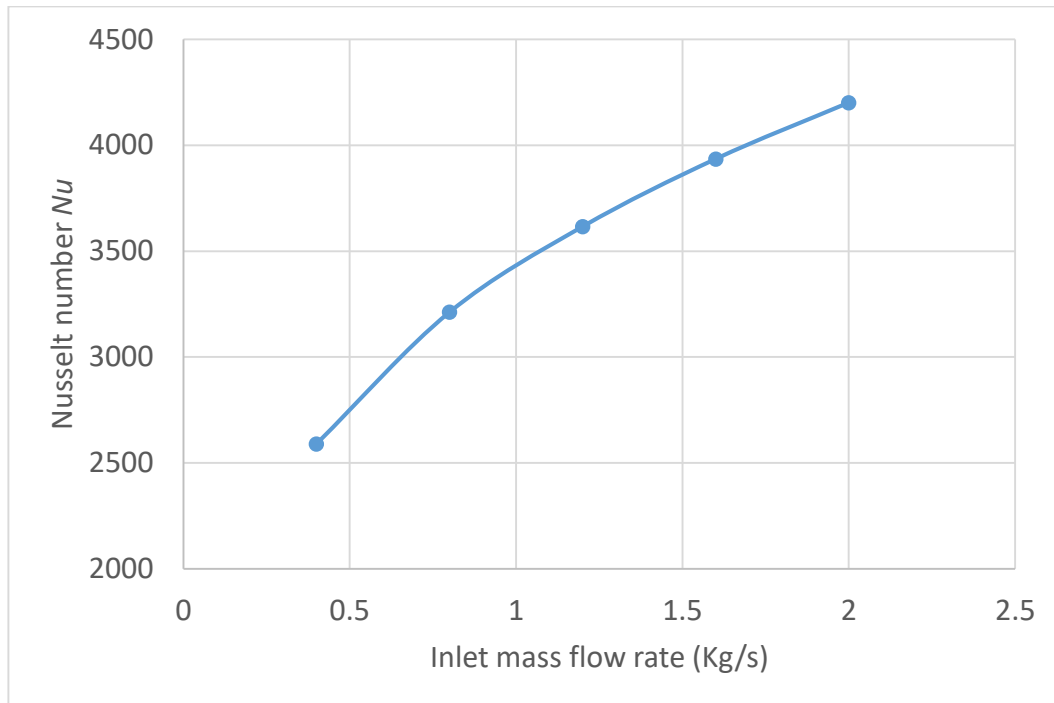


Figure 6. Nusselt number vs inlet mass flow rate.

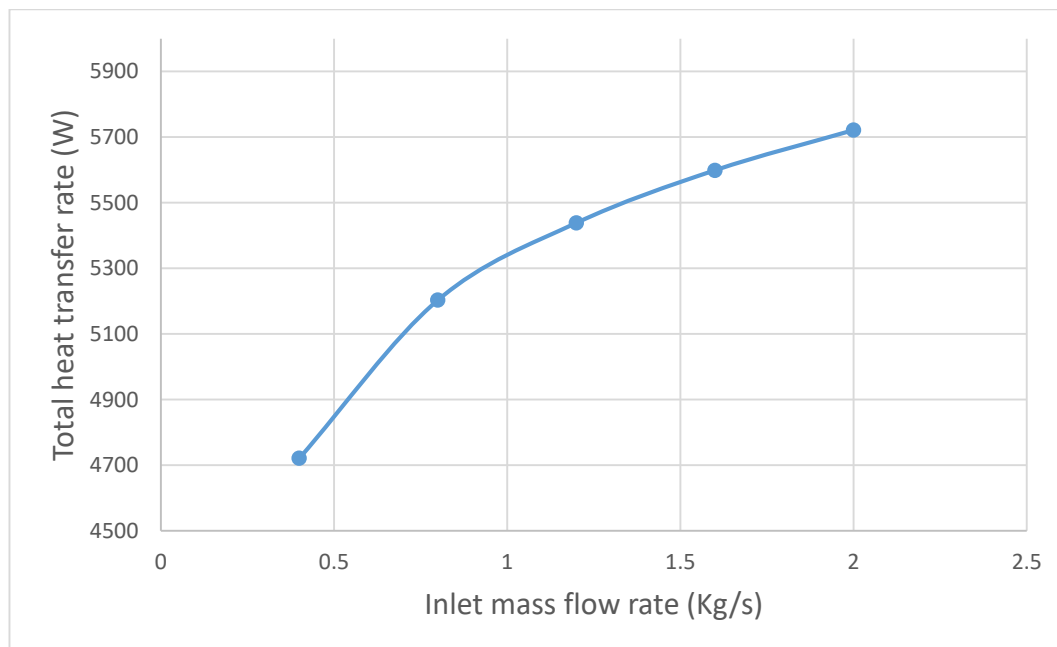


Figure 7. Total heat transfer rate vs inlet mass flow rate.

The overall discussion clarifies that changing the inlet mass flow rate at the shell inlet does not have a big impact on the performance parameters. However, the cost in terms of pressure drop increases nonlinearly.

Online ISSN

3007-3197

Print ISSN

3007-3189

<http://amresearchreview.com/index.php/Journal/about>

Steady Local Thermal and Flow Analysis: Contours Analysis

In Figure 8, pressure contours are plotted in (a) xy-plane, (b) various planes parallel to yz-plane and (c) in the zx-plane. The shell side pressure is higher up to 1900 Pa at the inlet and vicinity which gradually decreases along the flow direction which the positive x-axis. A similar pattern is observed in the tubes, where pressure is higher at the tube inlets and gradually decreases along the flow direction, which is the negative x-axis. The contours also show that the baffles affecting the pressure drop significantly.

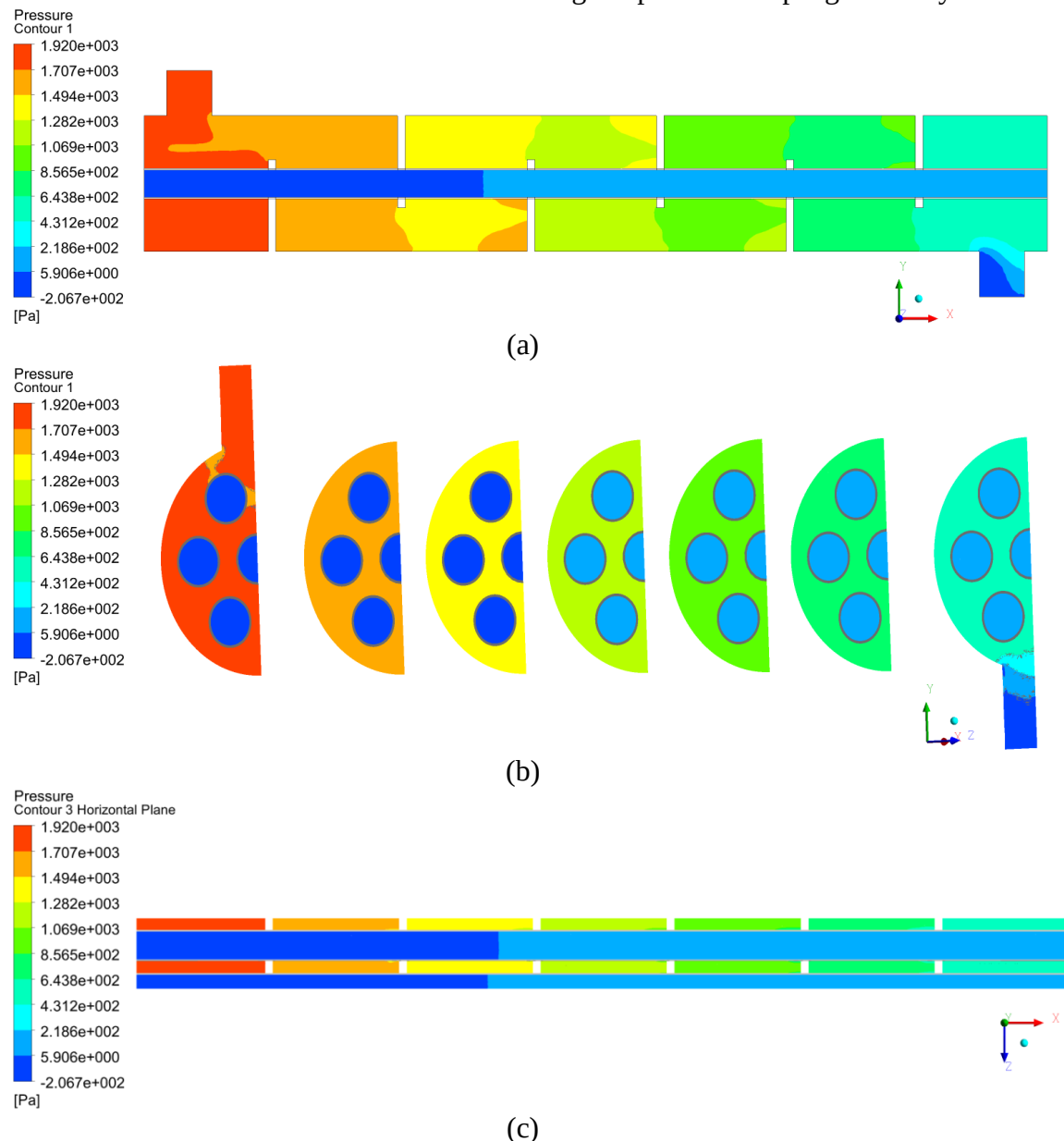


Figure 8. Pressure contours in (a) xy-plane, (b) various planes parallel to yz-plane, and (c) in zx plane.

In Figure 9, temperature contours are plotted in (a) xy-plane, (b) various planes parallel to yz-plane and (c) in zx plane. The contours are highlighting the distribution of heat

Online ISSN

3007-3197

Print ISSN

3007-3189

<http://amresearchreview.com/index.php/Journal/about>

within the tubes and the shell side. Higher temperatures are evident near the tube surfaces, which indicates efficient heat transfer between the working fluids. As the flow progresses downstream, a gradual drop in temperature can be observed in the tubes, while in shell side there is a downstream gradual increase in temperature, demonstrating steady heat exchange performance along the entire length of the exchanger.

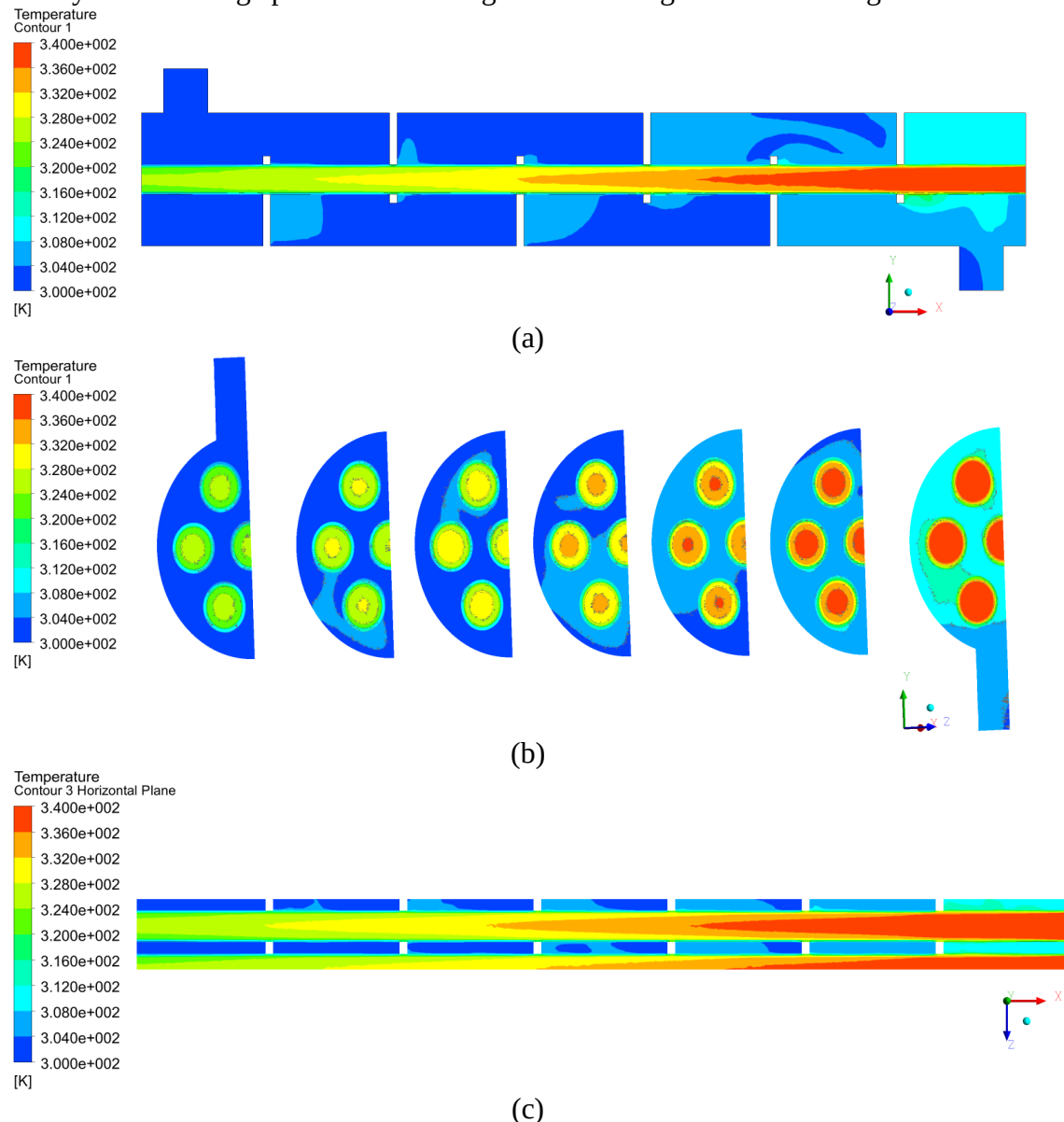


Figure 9. Temperature contours in a) xy-plane, (b) various planes parallel to yz-plane and (c) in zx plane.

In Figures 10, velocity contours are plotted in (a) xy-plane, (b) various planes parallel to yz-plane and (c) in zx plane. The fluid velocity in the shell and tubes are consistent with the physics of the problem.

Online ISSN

3007-3197

Print ISSN

3007-3189

<http://amresearchreview.com/index.php/Journal/about>

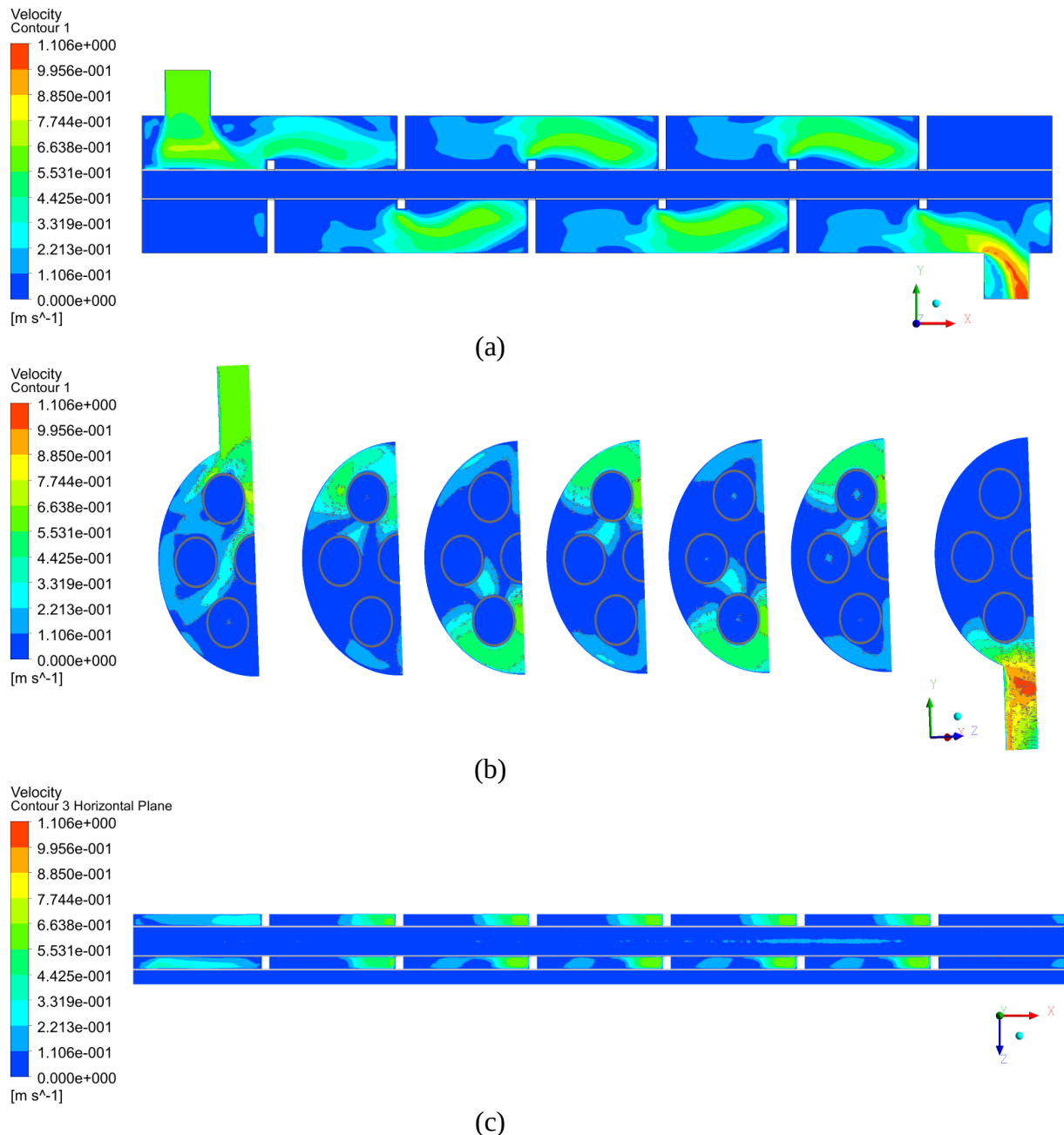


Figure 10. Velocity contours in a) xy-plane, (b) various planes parallel to yz-plane and (c) in zx plane.

In Figure 11, shell-side velocity streamlines are plotted. The fluid strikes each baffle, resulting in a change in its direction, and crosses through the space between the shell and the baffles. There are wakes behind the baffles, resulting in the fluid becoming static behind the baffles and, as a result, heat accumulation in this area, as may be seen in the temperature contours given in Figure 9. There are flow circulations also behind the baffles in the shell.

Online ISSN

3007-3197

Print ISSN

3007-3189

<http://amresearchreview.com/index.php/Journal/about>

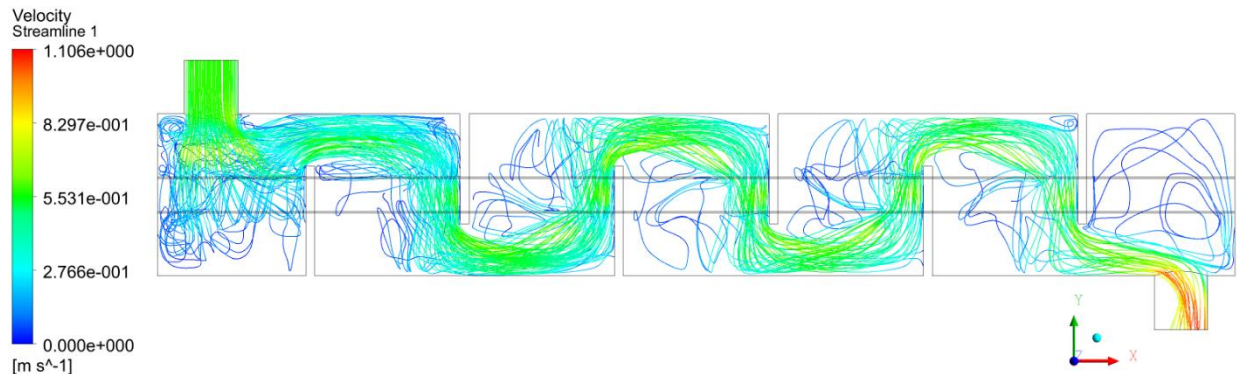


Figure 11. Velocity streamlines computed from cold fluid inlet, i.e., shell side inlet.

CONCLUSIONS

In this research, a shell and tube heat exchanger has been studied under various flow conditions. Increasing the shell side inlet mass flow rate enhanced the heat transfer process and lowered the hot fluid outlet temperature, but it also caused a noticeable rise in pressure drop after a certain point. For the 0.4 Kg/s of the shell side inlet mass flow rate lowered the hot fluid temperature from 340 K to 322 K. On increasing the mass flow rate lowered the temperature further but not significantly because the baffles and turbulence already improved the fluid distribution and boosted thermal performance for lower values of mass flow rate. However, the pressure drop increases nonlinearly. These results highlight the need for a well-balanced design and controlled flow parameters. Overall, managing these factors carefully can lead to better efficiency and lower energy use.

REFERENCES

- [1] Gaddis D, editor. Standards of the Tubular Exchanger Manufacturers Association. Tarrytown (NY): TEMA Inc.; 2007.
- [2] Process heat transfer. New York (NY): McGraw-Hill; 1950.
- [3] Bell KJ. The Delaware method for shell side design. In: Kakac S, Bergles AE, Mayinger F, editors. Heat exchangers: thermal-hydraulic fundamentals and design. New York: Hemisphere; 1981. p. 581–618.
- [4] Gay B, Mackley NV, Jenkins JD. Shell-side heat transfer in baffled cylindrical shell-and-tube exchangers – an electrochemical mass transfer modelling technique. *Int J Heat Mass Transfer* 1976;19:995 1002.
- [5] Halle H, Chenoweth JM, Wabsganss MW. Shell-side water flow pressure drop
- [6] distribution measurements in an industrial-sized test heat exchanger. *J Heat Transfer* 1988;110:60 7.
- [7] Pekdemir T, Davies TW, Haseler LE, Diaper AD. Pressure drop measurements on the shell side of a cylindrical shell-and-tube heat exchanger. *Heat Transfer Eng* 1994;15:42 56.
- [8] Gaddis ES, Gnielinski V. Pressure drop on the shell side of shell-and-tube heat exchangers with segmental baffles. *Chem Eng Process* 1997;36:149 59.

Online ISSN

3007-3197

Print ISSN

3007-3189

<http://amresearchreview.com/index.php/Journal/about>

- [9] Li HD, Kottke V. Visualization and determination of local heat transfer coefficients in shell-and-tube heat exchangers for staggered tube arrangement by mass transfer measurements. *Exp Therm Fluid Sci* 1998;17:210–6.
- [10] Karno A, Ajib S. Effect of tube pitch on heat transfer in shell-and-tube heat exchangers – new simulation software. *Heat Mass Transfer* 2006;42:263–70.
- [11] Sparrow EM, Reifschneider LG. Effect of interbaffle spacing on heat transfer and pressure drop in a shell-and-tube heat exchanger. *Int J Heat Mass Transfer* 1986;29:1617–28.
- [12] Eryener D. Thermoeconomic optimization of baffle spacing for shell and tube heat exchangers. *Energy Convers Manage* 2006;47:1478–89.
- [13] Sunden B. Computational heat transfer in heat exchangers. *Heat Transfer Eng.* 2007; 28:895–7.
- [14] Prithiviraj M, Andrews MJ. Three-dimensional numerical simulation of shell-and-tube heat exchangers. Part 1: foundation and fluid mechanics. *Numerical Heat Transfer. Part A*, Appl 1998;33:799–816.
- [15] Sha WT, Yang CI, Kao TT, Cho SM. Multidimensional numerical modeling of heat exchangers. *J Heat Transfer* 1982;104:417–25.
- [16] He YL, Tao WQ, Deng B, Li X, Wu Y. Numerical simulation and experimental study of flow and heat transfer characteristics of shell side fluid in shell-and-tube heat exchangers. In: *Proceedings of the fifth international conference on enhanced, compact and ultra-compact heat exchangers*, Science, Engineering and Technology, Hoboken, NJ; September 2005. p. 29–42.
- [17] Stevanovic´ Z, Ilic´ G, Radojkovic´ N, Vukic´ M, Stefanovic´ V, Vuc´kovic´ G. Design of shell and tube heat exchangers by using CFD technique – Part one: thermo hydraulic calculation. *Facta Universitatis Series: Mech Eng* 2001;8:1091–105.
- [18] Moawad MA, Ibrahim E, Gomaa A. Thermal performance of a pipe-in-pipe heat exchanger with sinusoidal inner pipe. *Energy Convers Manage* 2008;49:678–86.
- [19] Chen W-L, Dung W-C. Numerical study on heat transfer characteristics of double tube heat exchangers with alternating horizontal or vertical oval cross-section pipes as inner tubes. *Energy Convers Manage* 2008;49:1574–83.
- [20] F. A. McClintock, “The design of heat exchangers for minimum irreversibility,” *Engineering, ASME Pap.*, no. February 1951, p. 108, 1951, doi: 10.4236/eng.2011.39112.
- [21] Heat Exchanger Design Handbook by Thulukkanam Kuppan, MarcelDekker,2000 [22] Kuppan, T. (2000). *Heat Exchanger Design Handbook*. CRC Press.
- [23] Kumar, V. (2018). *Shell and Tube Heat Exchangers: Fundamentals, Design, and Applications*. Wiley.
- [24] Cengel, Y. A. (2015). *Heat Transfer: A Practical Approach*. McGraw-Hill Education.
- [25] Kern, D. Q. (1965). *Process Heat Transfer*. McGraw-Hill Book Company.
- [26] Darcy, H. (1856). *Les Fontaines Publiques de la Ville de Dijon*. Victor Dalmont
- [27] TEMA (2013). *TEMA Standards*. Tubular Exchanger Manufacturer Association

Online ISSN

3007-3197

Print ISSN

3007-3189

<http://amresearchreview.com/index.php/Journal/about>

- [28] Sunden B. Computational fluid dynamics in research and design of heat exchangers. Heat Transfer Eng. 2007; 28:898-910.
- [29] Spalart PR, Allmaras SR. A one-equation turbulence model for aerodynamic flows. AIAA paper 92-0439.
- [30] Wilcox DC. Turbulence modeling for CFD. 2nd ed. California: DCW Industries Inc.; 1998.