INTERNATIONAL JOURNAL OF PROGRESSIVE RESEARCH IN ENGINEERING MANAGEMENT AND SCIENCE (IJPREMS)
(Int Peer Reviewed Journal)
Vol. 05, Issue 08, August 2025, pp : 867-872
www.ijprems.com
editor@ijprems.com
e-ISSN : 2583-1062
Impact Factor : 7.001

PERFORMANCE OPTIMIZATION OF SHELL AND TUBE HEAT EXCHANGERS WITH OSCILLATING FLOW USING NUMERICAL SIMULATION

Raju Kumar Ranjan[1], Surjeet Singh Rajpoot[2], Dr. S.K. Nagpure[3]

[1]M.Tech. Research Scholar, Scope Global Skills University, Bhopal, MP, India.

[2]Assistant Professor, Scope Global Skills University, Bhopal, MP, India.

[3]Associate Professor, Scope Global Skills University, Bhopal, MP, India.

ABSTRACT

Shell-and-tube heat exchangers are widely used in industrial applications for their robustness and high heat transfer capabilities. However, their thermal performance can be limited by boundary layer development and flow distribution. Introducing oscillating (pulsating) flow has emerged as a promising method to enhance heat transfer by periodically disturbing the thermal boundary layer and promoting fluid mixing. In this study, a three-dimensional numerical simulation is conducted to investigate the performance optimization of a shell-and-tube heat exchanger under oscillating flow conditions. The unsteady Navier–Stokes and energy equations are solved using the finite volume method with appropriate turbulence models to capture transient flow features. Key parameters, including oscillation frequency, amplitude, and phase angle, are varied to assess their influence on heat transfer rate, pressure drop, and overall thermal–hydraulic performance. The results indicate that an optimal range of oscillation parameters exists where heat transfer enhancement is maximized with minimal penalty in pumping power. These findings provide valuable insights into the design and operation of oscillation-assisted heat exchangers for improved energy efficiency.

Keywords: Effectiveness, Temperature, Heat Transfer, Heat capacity, Reynolds Number, Nusselt Number, CFD.

1. INTRODUCTION

Shell-and-tube heat exchangers (STHEs) are among the most widely used thermal devices in power plants, petrochemical industries, refrigeration systems, and process engineering due to their high versatility, durability, and ability to handle large temperature and pressure differences. The basic principle of their operation relies on the transfer of thermal energy between two fluids separated by a solid wall, typically with one fluid flowing inside the tubes and the other in the surrounding shell. While conventional designs and steady flow operations are well established, thermal performance in such exchangers is often limited by the development of thick thermal boundary layers and regions of stagnant or low-velocity fluid.

One approach to improving the heat transfer performance is through the introduction of oscillating (pulsating) flow. Oscillating flow periodically accelerates and decelerates the fluid, which disrupts the thermal boundary layer, enhances fluid mixing, and promotes more uniform temperature distribution. Unlike steady flow, oscillating flow can induce secondary vortices and periodic reversal of velocity gradients, resulting in improved convective heat transfer coefficients without necessarily increasing the average flow rate.

Advancements in computational fluid dynamics have made it possible to numerically simulate such unsteady flows with high accuracy. By solving the transient Navier–Stokes and energy equations, CFD enables detailed visualization of velocity fields, temperature contours, and pressure distribution, which are difficult to capture experimentally. Moreover, parametric studies on oscillation amplitude, frequency, and phase angle can be conducted efficiently, allowing optimization for maximum thermal–hydraulic performance.

In this work, a three-dimensional CFD investigation is carried out to analyze the effects of oscillating shell-side flow on the thermal performance of a shell-and-tube heat exchanger. The study aims to identify optimal oscillation parameters that yield maximum heat transfer enhancement with minimal pressure drop penalty, thereby providing practical guidelines for performance optimization in industrial applications.

2. LITERATURE REVIEW

The heat transfer from primary fluid to secondary fluid is more than the without modification system. It permits the temperature of fluid increase for some useful purposes in manufacturing process. Various experiments were done in this area to enhance the rate of heat transfer to improve the efficiency of the system.

K. Vijaya KumarReddy et al [1], CFD Analysis of a Helically Coiled Tube in Tube Heat Exchanger, A helical coil tube heat exchanger is generally applied in industrial applications due to its compact structure, larger heat transfer area and higher heat transfer capability etc,

Karan Ghule et al [2], Numerical Heat Transfer Analysis of Wavy Micro Channels with Different Cross Sections" Among the various heat transfer enhancement techniques employed in micro channels, the use of wavy micro channels has been gaining popularity. In this study, numerical heat transfer analysis of wavy micro channels of different cross sections has been conducted by varying the Reynolds number and the amplitude of waviness.

Eshita Pal et al [3], CFD simulations of shell-side flow in a shell-and-tube type heat exchanger with and without baffles, Shell-and-tube heat exchanger has been extensively used in industrial and research fronts for more than a century. However, most of its design procedures are based on empirical correlations extracted from experimental data of long length shell and tube heat exchanger. In this paper, an attempt has been made to investigate the complex flow and temperature pattern in such a short shell and tube type heat exchanger, with and without baffles in the shell side.

M. M. Bhutt et al [4], Review on CFD use in the various design of heat exchangers, This literature review focuses on the applications of Computational Fluid Dynamics (CFD) in the field of heat exchangers. It has been found that CFD has been employed for the following areas of study in various types of heat exchangers: fluid flow maldistribution, fouling, pressure drop and thermal analysis in the design and optimization phase.

Arezuo Ghadi et al [5], CFD Modelling of Increase Heat Transfer in Tubes by Wire Coil Inserts, In this study has been studied the effect of improving heat transfer coils in heat exchanger in a laboratory by the method of computational fluid dynamics. A shell – tube heat exchanger is used in the laboratory. Difference in temperature and pressure are measured and compared in three different steps of coil, between input and output of each heat exchanger tubes, in the absence and presence coil.In this work the k– and RNG model has been used for representing the effects of the turbulence in tubes by CFD.

Bilal Sungur et al [6], Numerical analysis of the effect of conical turbulators to heat transfer performance of a liquid fuelled boiler, In this study, increasing the efficiency of liquid fuelled smoke tube boilers used for domestic heating was researched. In this context, turbulators with conical geometries placed to smoke tubes of boiler and effects on flame structure and heat transfer were investigated numerically.

D. Kaliakatsos et al [7], CFD Analysis of a Pipe Equipped with Twisted Tape, in this work, a pipe provided with twisted tape inserts is analyzed. This system allows a significant increase of convective heat transfer coefficient by introducing a swirl motion which determines greater heat removal from the solid surface, by improving the fluid mixing. The analysis performed in this paper focuses on the evaluation of the thermal and flow quantities for a pipe of a shell and tube heat exchanger, previously optimized through a design software widely used in the petrochemical industry.

Santosh K. Hulloli et al [8], Numerical Study of Heat Transfer Enhancement in Shell And Tube Heat Exchanger Using CFD, This paper numerically demonstrates the advantage of using different designs of baffles and semicircular turbulators inserted in the shell and tube heat exchangers. In this work, a shell and tube heat exchanger is considered for heat transfer enhancement studies.

3. PROBLEM FORMULATION

Although shell-and-tube heat exchangers are extensively used in industrial applications, their thermal performance is often constrained by limitations in heat transfer due to the formation of stable thermal boundary layers and regions of poor mixing. Conventional enhancement techniques—such as adding baffles, using extended surfaces, or increasing flow velocity—can improve heat transfer but usually at the cost of higher pressure drops and increased pumping power. This creates a trade-off between thermal performance and energy consumption.

Oscillating (pulsating) flow offers an alternative approach for performance enhancement by periodically disturbing the boundary layer, inducing secondary flows, and improving mixing without requiring significant changes in mean flow rate. However, the optimal combination of oscillation parameters—such as frequency, amplitude, and waveform—for maximum performance with minimal pressure penalty is not well established for shell-and-tube configurations, especially when operating under turbulent or transitional flow regimes.

The problem addressed in this study can therefore be formulated as follows:

- **Given:**
  - Geometry of a single-pass shell-and-tube heat exchanger
  - Inlet fluid properties and operating conditions
  - Range of oscillation parameters (frequency, amplitude, phase)
  - CFD-based numerical simulation framework for unsteady flow

INTERNATIONAL JOURNAL OF PROGRESSIVE RESEARCH IN ENGINEERING MANAGEMENT AND SCIENCE (IJPREMS)
(Int Peer Reviewed Journal)
Vol. 05, Issue 08, August 2025, pp : 867-872
www.ijprems.com
editor@ijprems.com
e-ISSN : 2583-1062
Impact Factor : 7.001

- **Find:**

o The influence of oscillating flow parameters on heat transfer rate, pressure drop, and overall thermal–hydraulic performance

o The optimal oscillation parameters that maximize heat transfer enhancement with minimal increase in pumping power

- **Subject to:**

o Governing equations of unsteady incompressible flow:

- Continuity equation
- Navier–Stokes momentum equations (transient form)
- Energy equation for heat transport

o Boundary conditions at tube walls, shell walls, and inlet/outlet

o Realistic industrial operating constraints

This formulation frames the investigation as an optimization problem under physical and operational constraints, making it suitable for CFD-based numerical analysis and validation.

Theoretical Calculation

The amount of heat transferred in any process can be defined as the total amount of transferred energy excluding any macroscopic work that was done and any energy contained in matter transferred. For the precise definition of heat, it is necessary that it occur by a path that does not include transfer of matter. As an amount of energy (being transferred), the SI unit of heat is the joule (J). The conventional symbol used to represent the amount of heat transferred in a thermodynamic process is Q. Heat is measured by its effect on the states of interacting bodies, for example, by the amount of ice melted or a change in temperature.

Heat transfer is a fundamental energy engineering operation. Hot water loops are commonly used to transfer heat in district heating networks and on industrial sites. The capital & operating cost of many hot water loops are higher than they should be. This post will explain why this is happening in the context of the foundational energy engineering equation Q = m * Cp * dT.

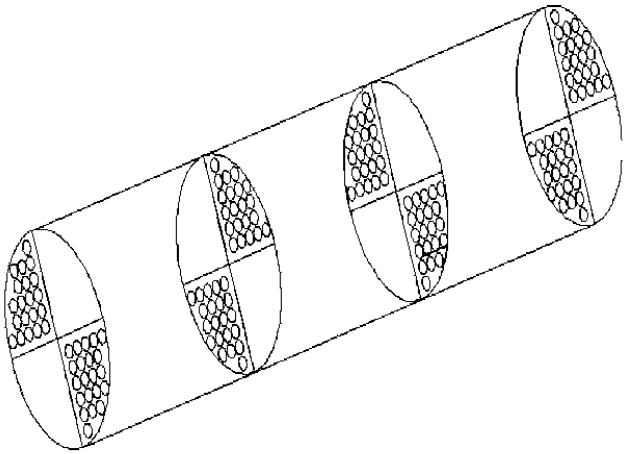

Fig. 1 Interstitial Cross Sections

Above fig shows interstitial cross sections of tube selected for investigation.

This Hex-dominant Parametric (only CFD) mesh is used to generate the mesh for the 3 volumes (1 solid and 2 fluids) followed by creating refinements. Also, the volumes are defined as distinct regions in order to define interfaces at a later point in time. Post the meshing operation, we have 3 different regions viz. Solid Pipes**,** Inner Fluid**,** and Outer Fluid**.**

The image below illustrates the flow of the temperature streamlines in the shell of the boiler. As can be seen in the image, the temperature gradient of the Outer fluid is much steeper at entry and gradually decreases with the furtherance of the fluid across the shell.

www.ijprems.com
editor@ijprems.com

INTERNATIONAL JOURNAL OF PROGRESSIVE RESEARCH IN ENGINEERING MANAGEMENT AND SCIENCE (IJPREMS)
(Int Peer Reviewed Journal)
Vol. 05, Issue 08, August 2025, pp : 867-872

e-ISSN : 2583-1062
Impact Factor : 7.001

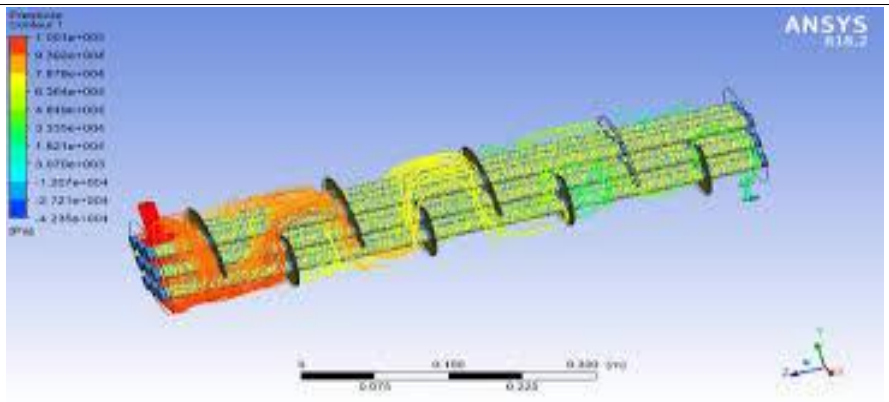

Fig. 2 Temperature plot of shell and tube heat exchager

Above fig shows CFD analysis and result of Temperature plot of hot fluid of shell and tube heat exchanger. Formulae

1. Percentage of difference between T-cfd and T-c2

[(Tc2 - Tc2-CFD)/Tc2]*100]

2. Heat transfer m_h C_{ph} (T_{h1}-T_{h2})

3. Effectiveness

Above table shows inlet, outlet conditions of hot and cold water in heat exchanger for all tube cross sections. % difference column shows difference between Tc2 (CFD) and Tc2

Where,

Th1 – Inlet temperature of the hot fluid

Th2 – Outlet temperature of the hot fluid

Tc1 – Inlet temperature of the Cold fluid

Tc2 (CFD) – Simulation value of Inlet cold fluid

Tc2 – Outlet temperature of the cold fluid

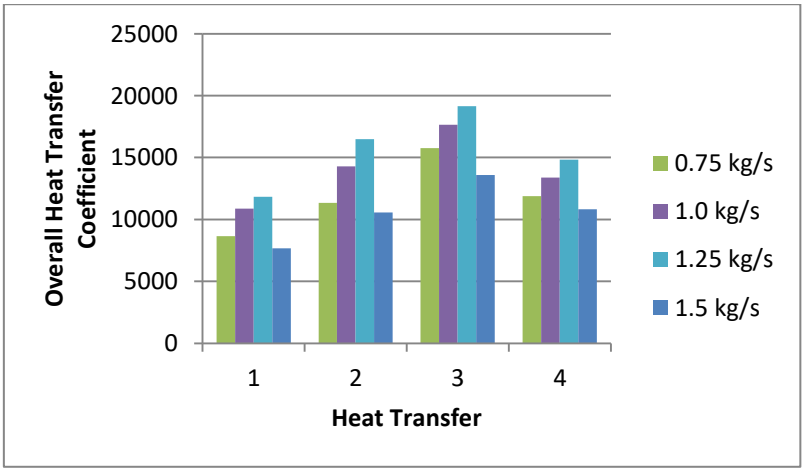

Fig 3 Plot of Overall Heat Transfer Coefficient

Figure presents the variation of the overall heat transfer coefficient for four heat transfer cases at different mass flow rates. The key observations are Case 3 again records the highest U-values across all mass flow rates, peaking at approximately 19,000 W/m²·K for 1.25 kg/s, indicating superior convective heat transfer. Case 2 shows the second-highest U-values (≈ 16,500 W/m²·K at 1.25 kg/s), followed by Case 4, while Case 1 has the lowest U-values. In general, increasing mass flow rate from 0.75 kg/s to 1.25 kg/s enhances the overall heat transfer coefficient due to increased turbulence and mixing. However, at 1.5 kg/s, the U-value drops in most cases, suggesting that the benefit of increased Reynolds number is outweighed by reduced residence time and possible flow distribution in the exchanger. Comparing this with the Effectiveness results, it is clear that Case 3 at 1.25 kg/s delivers the best thermal–hydraulic performance, making it the optimal operating condition in the tested range.

INTERNATIONAL JOURNAL OF PROGRESSIVE RESEARCH IN ENGINEERING MANAGEMENT AND SCIENCE (IJPREMS)
(Int Peer Reviewed Journal)
Vol. 05, Issue 08, August 2025, pp : 867-872
www.ijprems.com
editor@ijprems.com
e-ISSN : 2583-1062
Impact Factor : 7.001

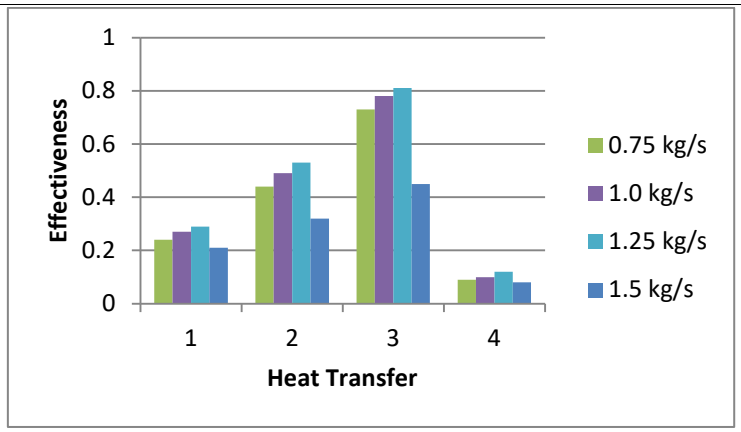

Fig. 4 Graph of Correlation of Effectiveness

The variation of heat exchanger effectiveness for different heat transfer cases and mass flow rates is presented in Figure. It is observed that Case 3 exhibits the highest effectiveness across all mass flow rates, reaching ~0.8 for 1.25 kg/s and ~0.76 for 0.75 kg/s. Case 2 shows moderate effectiveness, with values between 0.45–0.55 for most flow rates. Case 1 records lower effectiveness (0.2–0.25), while Case 4 shows minimal effectiveness (~0.08–0.12) for all tested flow rates. Increasing mass flow rate from 0.75 kg/s to 1.25 kg/s generally increases effectiveness in most cases; however, at 1.5 kg/s, effectiveness tends to drop, likely due to reduced residence time in the exchanger. These results indicate that optimal thermal performance is achieved in Case 3 at 1.25 kg/s, suggesting a balance between convective heat transfer enhancement and sufficient fluid residence time.

4. CONCLUSION

In this the three dimensional model is investigated for the shell and tube heat exchanger and CFD analysis is carried out for the Heat exchanger unit with different designs of cross sections.

- At improving the models' temperature and heat transfer coefficient. Compared to the simple model, the shell and tube model has an improved heat transfer coefficient of 145.
- Compared to the plain model, the tube's effectiveness increased by roughly 0.24, 0.55, 0.85, and 0.16 for varying flow rates, respectively.
- Based on the data and findings from CFD, we have determined that the turbulence model is more appropriate for our simulation. Compared to other variants, the one with the current shell and tube is more effective. A good heat transfer rate is also provided by the heat exchanger with tubes. A heat exchanger that is more effective performs better in its particular application.

5. REFERENCES

[1] Shrikant, R. Sivakumar, N. Anantharaman, M. Vivekenandan, “CFD simulation study of shell and tube heat exchangers with different baffle segment configurations”, Applied Thermal Engineering, Volume 108, Pages 999-1007, 2016.

[2] Mustapha Mellal, Redouane Benzeguir, Djamel Sahel, Houari Ameur, “Hydro-thermal shell-side performance evaluation of a shell and tube exchanger under different baffle arrangement and orientation”, International Journal of Thermal Sciences, volume 121, page no 138-149, 2017.

[3] P. K. Sarma, T. Subramanyam, P. S. Kishore, V. D. Rao and S. Kakac, “Laminar convective heat transfer with twisted tape inserts in a pipe,” International Journal of Thermal Science, vol. 42, pp. 821-828, 2003. DOI: 10.1016/S1290-0729(03)00055-3.

[4] S. W. Hong and A. E. Bergles, “Augmentation of laminar flow heat transfer in tubes by means of twisted tape inserts,” ASME J. Heat Transfer, vol. 98, pp. 251- 256, 1976. DOI: 10.1115/1.3450527.

[5] P. K. Sarma, P. S. Kishore, V. D. Rao and T. Subrahmanyam, “A combined approach to predict friction coefficients and convective heat transfer characteristics in a tube with twisted tape inserts for a wide range of Re and Pr,” International Journal Of Thermal Science, vol. 44, pp. 393-398, 2005. DOI: 10.1016/j.ijthermalsci.2004.12.001.

[6] A. Galloro, R. Schimio and L. Danziani, “Ottimizzazione tecnico-economica di uno scambiatore di calore a fascio tubiero mediante l’utilizzo di inserti meccanici interni”, in Proc. of VII National Congress of AIGE, Rende (CS), Italy, 10-11 June 2013.

[7] Vijaya Kumar Reddy, Sudheer Prem Kumar, Ravi Gugulothu, Kakaraparthi Anuja and Viajaya Rao, "CFD Analysis of a Helically Coiled Tube in Tube Heat Exchanger", Materials Today: Proceedings 4, 2341–2349, 2017.

[8] Karan Ghule, M.S. Soni, "Numerical Heat Transfer Analysis of Wavy Micro Channels with Different Cross Sections", Energy Procedia, vol. 109, pp. 471 – 478, (2017).

[9] Eshita Pal, Inder Kumar, Jyeshtharaj B. Joshi, N.K. Maheshwari, "CFD simulations of shell-side flow in a shell-and-tube type heat exchanger with and without baffles", Chemical Engineering Science, vol. 143, pp. 314–340, 2016.

[10] M. M. Bhutt, Nasir H., M. Bashir, Kanwar N. Ahmad, Sarfaraz Khan, " Review on CFD use in the various design of heat exchangers ", Applied Thermal Engineering, 32, pp.1-12, 2012.

[11] R Andrzejczyk, T Muszynski, "Geometrical and Thermodynamic characteristics of mixed convection heat transfer in the shell and coil tube heat exchanger with a number of baffles", Applied Thermal Engineering, volume 121, pp115-125, 2017.

[12] S.K.Routa, D. N. Thatoia, A.K. Acharya, D. P. Mishra, "CFD supported performance estimation of an internally finned tube heat exchanger under mixed convection flow", Procedia Engineering 38, pp. 585-597, 2012.

[13] N T Anoop.K.S, Deepak.C.S, E..P.Kuriakose, Habeeb Rahman.K.K, Karthik.K.V, "Computational fluid dynamics of the tube in tube heat exchanger with fins", International Research Journal of Engineering and Technology, volume 3, issue 4, 2016

[14] J Johnson, Abdul, A Shani, Harif, H Hameed, Nithin, "Computational fluid dynamics of double pipe heat exchanger", International Journal of Science, Engineering and technology research, volume 4, issue 5, 2015

[15] Tahseen Ahmad Tahseena, M.M. Rahman and M. Ishaka, "Experimental study on Heat transfer and friction factor in laminar forced convection over flat tube over channel flow", International conference on thermal engineering, volume 105, pp. 46-55, 2015