WWW.IJRPR.COM

International Journal of Research Publication and Reviews

Journal homepage: www.ijrpr.com ISSN 2582-7421

Optimization of Thermal and Hydraulic Aspect on Inlet Passage in Shell and Tube Heat Exchangers By CFD

***Rahul Ahirwar*[#1], *Surjeet Singh Rajpoot*[*2]**

#M.Tech. Research Scholar, *Assistant Professor
SCOPE College of College of Engineering, Bhopal, MP, India

ABSTRACT-

The thermal performance of shell and tube heat exchanger configurations, including a conventional tube model and an enhanced modified tube model, was analysed in this research. Simulations were conducted to assess the thermal behaviour of the entire heat exchanger using a numerical approach. The inner components of the enhanced heat exchanger were modelled in three dimensions for greater accuracy. The findings revealed that the use of modified tubes impedes the flow across cross-sections, which impacts the overall efficiency of the heat exchanger. Additionally, the pressure drop was found to be higher in heat exchangers equipped with modified tubes compared to the other configurations. To address efficiency concerns, a protrusion was introduced on the inner wall of the heat exchanger.

Keywords: Effectiveness, Heat capacity, Temperature, Heat Flow rate, CFD

INTRODUCTION

The passage you provided is a detailed explanation of heat exchangers, covering their function, applications, types, and challenges in selection and maintenance. If you're looking for something similar, here's a slightly revised version that retains the core concepts:

A heat exchanger is a device designed to transfer heat between two or more fluids, essential in both heating and cooling processes. These fluids may be separated by a solid barrier to avoid mixing or can be in direct contact. Heat exchangers are integral to various applications, including space heating, refrigeration, air conditioning, power generation, and in industries such as chemical processing, petrochemical refining, natural gas processing, and wastewater treatment.

Despite being the most basic type of heat exchangers used in industries, these devices are valued for their cost-effectiveness in design and maintenance, making them ideal for smaller industrial applications. However, their lower efficiency and the significant space they require at larger scales have driven modern industries toward more efficient alternatives, such as shell and tube or plate heat exchangers.

Selecting the most suitable heat exchanger can be complex due to the numerous variables involved. While hand calculations are feasible, they often require multiple iterations, leading to a preference for computer-aided selection, either by engineers or equipment vendors.

In large-scale cooling water systems, water treatment processes like purification, chemical addition, and regular testing are crucial to reduce fouling in heat exchangers. Similarly, steam systems in power plants use water treatment to prevent fouling and corrosion, ensuring the longevity and efficiency of the heat exchange equipment and other related systems.

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.

PROBLEM FORMULATION

A heat exchanger is a device designed to transfer thermal energy between two or more fluids, between a solid surface and a fluid, or between solid particulates and a fluid, when they are at different temperatures and in thermal contact. Typically, heat exchangers operate without any external heat or work interactions, allowing for efficient heat transfer and significantly accelerating the rate of temperature change.

1. Studying the existing heat exchanger working characteristics, effectiveness, effectiveness, efficiency, losses etc.

2. Generating 3D model of existing heat exchanger using Solid-works software.

3. Theoretical calculations for new models.

4. Selecting of parameters for CFD analysis.

5. Obtaining its CFD model and simulating its working condition.

6. Implementing methods that are ought to improve the performance of heat exchanger.

7. Performing CFD analysis in ANSYS Fluent on new models

8. Comparing the results with the original model.

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 $Q = m * Cp * dT$.

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.

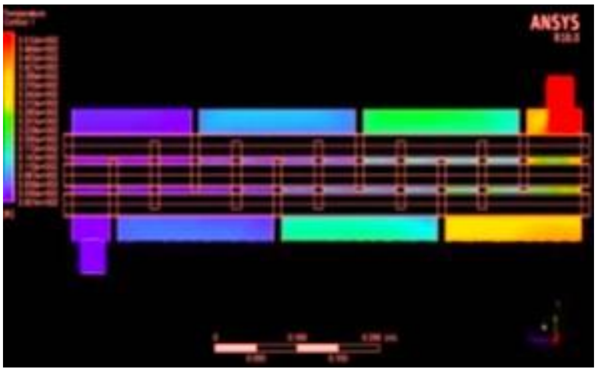

Fig. 2 Temperature plot of hot fluid of Hexagonal cross section tube H.E

The image appears to be thermal analysis visualization from ANSYS, likely related to a computational fluid dynamics (CFD) simulation of a shell and tube heat exchanger. It shows temperature distribution across the heat exchanger, with a colour gradient representing different temperature values.

Above fig shows CFD analysis and result of Temperature plot of hot fluid of Hexagonal cross section 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

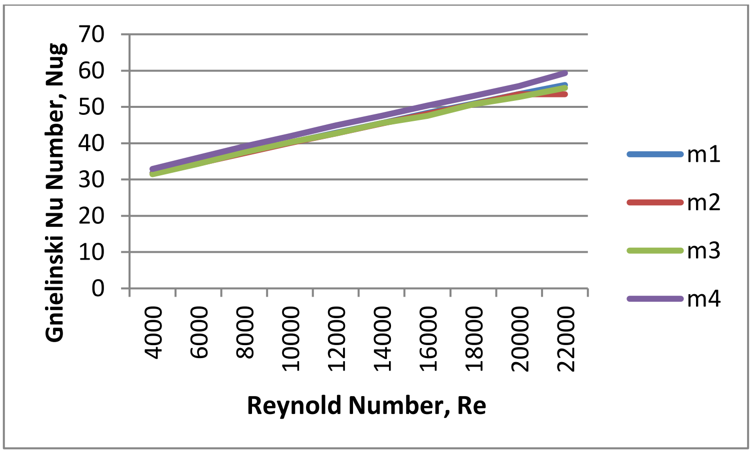

Fig 3 Graph of Gnielinski Number VS Reynolds Number

Above fig shows graph of heat transfer which shows comparison between different mass flow of tube. From graph it is seen that Gnielinski Number VS Reynolds Numbercross section tube is higher than other tube cross sections.

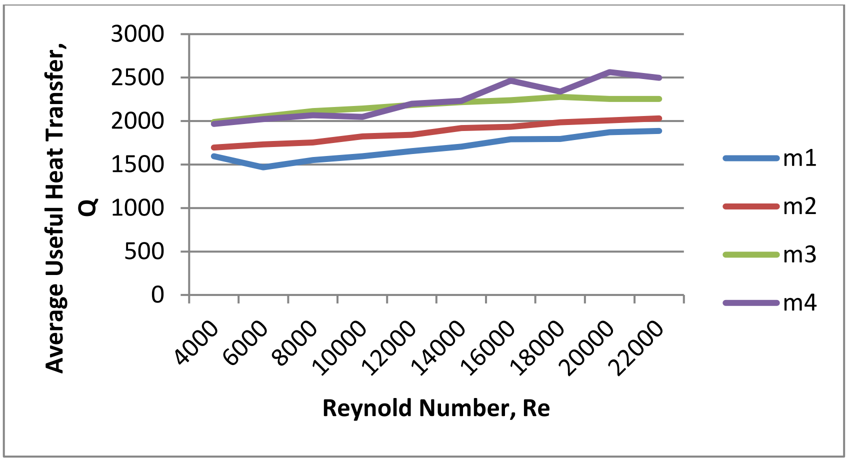

Fig. 4 Graph of Average Heat Transfer VS Reynolds Number

Above fig shows graph of Average Heat Transfer VS Reynolds Number which shows comparison between tubes. From graph it is seen that heat transfer and pressure drop of cross section tube is higher than other tube cross sections.

Conclusion

In this study, a three-dimensional model of a shell and tube heat exchanger is analyzed, and CFD simulations are conducted for the heat exchanger unit using various cross-sectional designs.

- The study focuses on the enhancement of the heat transfer coefficient and temperature across different models.
- The heat transfer coefficient is enhanced to 22.9 for models with varying flow rates and to 144.7 for the hexagonal cross-section model, compared to the plain model.
- The effectiveness of the tube increased by 0.29, 0.53, 0.81, and 0.12, respectively, when using different cross sections compared to the plain model.
- Based on the CFD results, it was concluded that the K-w turbulence model offers better suitability for our simulations. The model with the hexagonal tube provides superior effectiveness compared to other models. Additionally, the heat exchanger with a hexagonal tube design demonstrates a higher heat transfer rate. Enhancing the effectiveness of the heat exchanger directly improves its performance in its intended applications.

REFRENCES

[1] 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.

[2] 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).

[3] 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.

[4] 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.

[6] 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.

[7] 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.

[8] 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

[9] 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

[10] 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

[11] 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.

[12] 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.

[13] 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.

[14] 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.

[15] 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.

[16] 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.