International Journal of Research Publication and Reviews

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

Heat Transfer & Hydraulic Effect of Shell & Tube Heat Exchanger at Inlet Positioning by Using Simulation

Vindheshwari Prasad Mishra[1], Surjeet Singh Rajpoot[2]

[1] M. Tech. Research Scholar, *[2]Assistant Professor
SCOPE College of College of Engineering, Bhopal, MP, India

ABSTRACT-

This study has examined the thermal performance of various shell and tube heat exchanger topologies, including the performed tube model and the improved changed tube model. Using the results of the simulation, a numerical method was used to predict the outcomes for the entire heat exchanger. The internal parts of the upgraded heat exchanger were modelled in three dimensions. The results demonstrate that flow is prevented from crossing the cross-sections when modified tubes are used in place of regular tubes, which lowers heat exchanger efficiency. Additionally, heat exchangers with modified tubes have a greater pressure drop than the other two varieties. An extension was made to the inner wall of the heat exchanger to increase efficiency.

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

INTRODUCTION

A device that helps transfer heat between two or more fluids or between a fluid and a solid surface is called a heat exchanger. Effectively transferring thermal energy between two media without the two fluids coming into direct contact is the aim of a heat exchanger. Heat exchangers are widely used to control temperature, save energy, and speed up heating and cooling processes in a variety of commercial, industrial, and residential settings. The fundamental working principle of a heat exchanger is the movement of two different temperature fluids through independent channels, which permits the transfer of heat from the hot to the cold fluid. This procedure aids in preserving the proper temperatures, enhancing energy economy, and avoiding fluid contamination.

Heat exchangers are used in many different industries, such as refrigeration, chemical processing, HVAC systems, and power plants. They are essential for preserving ideal temperatures, increasing energy economy, and boosting the effectiveness of various industrial processes. One of the most widely used heat exchanger types in a variety of industries is the shell and tube heat exchanger. They are made up of tubes, which are smaller inner tubes through which fluids flow, and a shell, which is a large outer vessel. In the basic design, the tubes are positioned inside the shell, with one fluid flowing through the tubes and the other around their exterior.

A shell and tube heat exchanger works by having a hot fluid enter one set of tubes and a cold fluid enter the area in the shell that surrounds the tubes. The tube walls allow heat to be transferred from the hot fluid to the cold fluid and back again. The shell's internal baffles prevent the fluids from flowing freely, which facilitates effective heat exchange. These heat exchangers are adaptable and have a broad range of uses, such as HVAC systems, chemical processing, power plants, and petrochemical industries. Because of their sturdy design, they are especially well-suited for applications involving high pressure and high temperatures.

Due to the many variables involved, selecting optimal heat exchangers is challenging. Hand calculations are possible, but many iterations are typically needed. As such, heat exchangers are most often selected via computer programs, either by system designers, who are typically engineers, or by equipment vendors.

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 that is used to transfer thermal energy between two or more fluids, between a solid surface and a fluid, or between solid particulates and a fluid, at different temperatures and in thermal contact. External heat and work interactions are typically absent from heat exchangers. This speeds up the rate at which the temperature changes and helps heat transfer.

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.

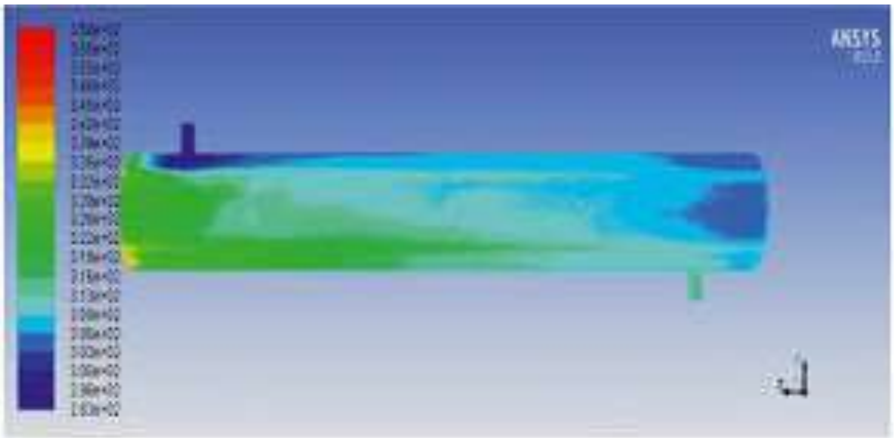

Fig. 2 Temperature plot of hot fluid of cross section

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

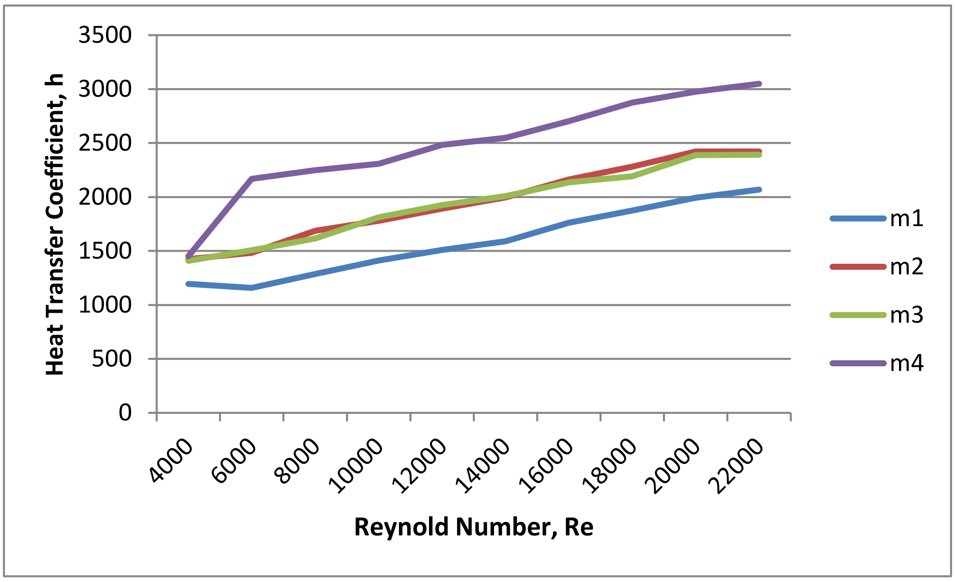

Fig 3 Graph of heat transfer coefficient VS Reyolds number

Above fig shows graph of heat transfer which shows comparison between different mass flow of shell & tube. From graph it is seen that heat transfer and effectiveness of shell & tube is higher than other tube cross sections.

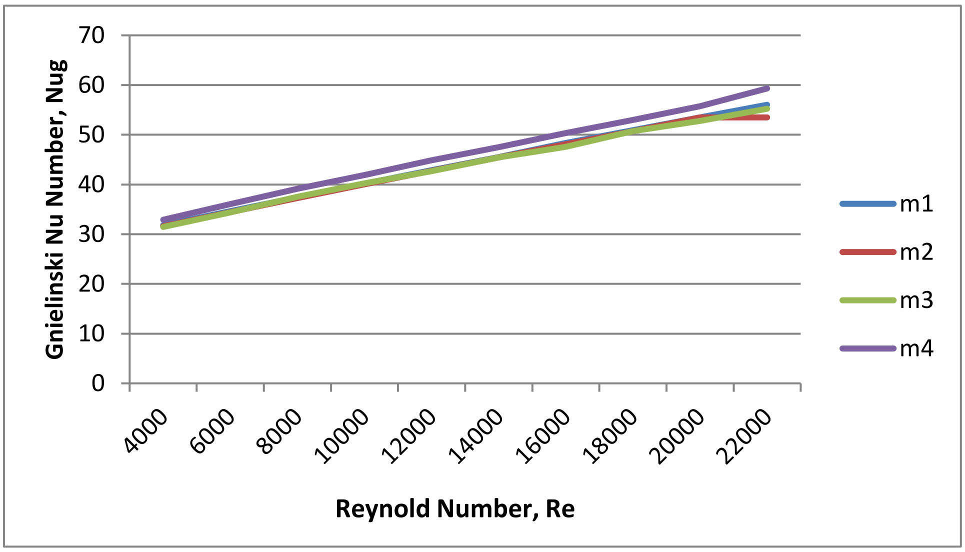

Fig. 4 Gnielinki Number with Reynolds Number

Above fig shows graph of Gnielinki Number with Reynolds Number which shows comparison between shell & tube tube. From graph it is seen Gnielinki Number with Reynolds Number of between shell & tube tube is higher than other tube cross sections.

Conclusion

In this research, the shell and altered tube heat exchanger is explored using a three-dimensional model, and a CFD study is run on a heat exchanger unit with varying mass flow configurations-

- The possibility of increasing the temperature and heat transfer coefficient for the models was looked into.
- Shell and tube model has an improved heat transfer coefficient 2496 W/m2/K at m4 0.20 kg/s mass flow rate when compared to the plain model.
- When compared to the simple model, the tube's effectiveness increased by around m4 for the various flow rates, respectively.
- We have obtained that the turbulence model offers greater fit for our simulation based on the results and data from CFD. More effectively th an other models, the model with the existing shell and tube. The heat transfer rate is also provided by heat exchangers using tubes. The perfo rmance of a heat exchanger in a given application increases as its effectiveness increases.

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 Procedi, 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.