NUMERICAL EVALUTION & HEAT ENHANCEMENT OF INTERSTITIAL OSCILLATING FLOW IN SHELL & TUBE HEAT EXCHANGER BY USING COMPUTATIONAL FLUID DYNAMICS

Gaurav Anand[#1], Surjeet Singh Rajpoot[*2]
#M.Tech. Research Scholar, *Assistant Professor
SCOPE College of College of Engineering, Bhopal, MP, India

Abstract

The Heat exchanger is a device which used to transfer heat from one fluid to another through a solid medium or interface. There is various type of heat exchanger available. In this paper shell and tube type heat exchanger is selected. Our objective was to change the cross section of tube to enhance the efficiency of the heat exchanger. An Interstitial Oscillating Flow in tubes is selected for the investigation. Design of new shell and tube heat exchanger is done using standard designing procedure and 3D modeling is done in Solidworks. Finite Element Analysis software ANSYS Workbench 18.0 is used to perform CFD analysis under a standard working condition to find performance parameter. We found that the Interstitial Oscillating Flow provides more effective heat exchange due to increase in the convective surface. The computational fluid dynamics (CFD) model equations are solved to predict the hydrodynamic and thermal behaviour of the exchanger. The geometry of the problem and meshing of it have been made in and the models have been solved by ANSYS Workbench. CFD models or packages provides the contours and data which predict the performance of the heat exchanger design and are effectively used because it has ability to obtain optimal solutions and has work in difficult and hazardous conditions.

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

INTRODUCTION

A heat exchanger is a device used to transfer heat between two or more fluids. Heat exchangers are used in both cooling and heating processes. The fluids may be separated by a solid wall to prevent mixing or they may be in direct contact. They are widely used in space heating, refrigeration, air conditioning, power stations, chemical plants, petrochemical plants, petroleum refineries, natural-gas processing, and sewage treatment.

Heat exchangers are the simplest exchangers used in industries. On one hand, these heat exchangers are cheap for both design and maintenance, making them a good choice for small industries. On the other hand, their low efficiency coupled with the high space occupied in large scales, has led modern industries to use more efficient heat exchangers like shell and tube or plate.

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.

In large-scale cooling water systems for heat exchangers, water treatment such as purification, addition of chemicals, and testing, is used to minimize fouling of the heat exchange equipment. Other water treatment is also used in steam systems for power plants, etc. to minimize fouling and corrosion of the heat exchange and other equipment.

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. In heat exchangers, there are usually no external heat and work interactions. This facilitates the transfer of heat, and greatly increases the rate of the temperature change.

1. Investigating 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.

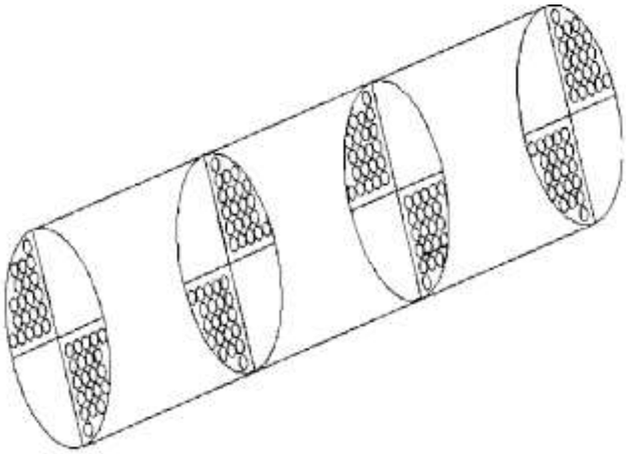

Fig. 1 Interstitial Cross Sections

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.

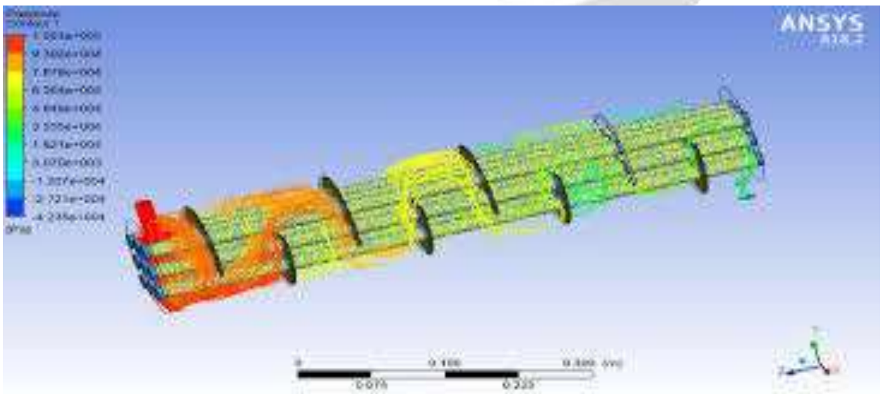

Fig. 2 Temperature plot of shell and tube heat exchager

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

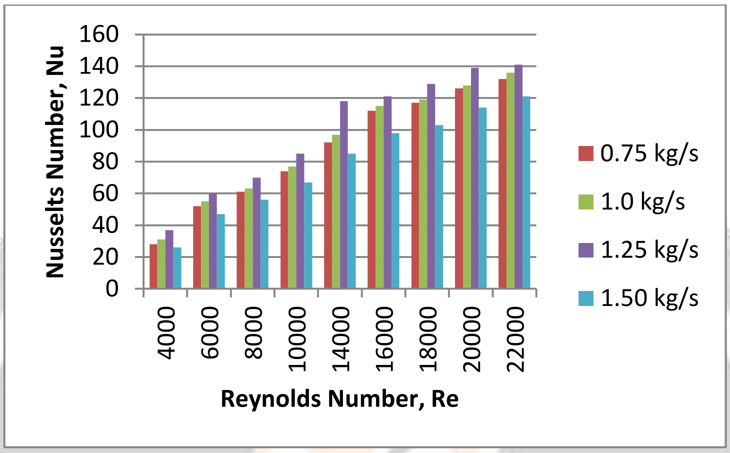

Fig 3 Plot of Nusselt number vs Reynolds Number

Above fig shows obtained numerical results for water are compared with the available Nusselt number correlations in the estimated Reynolds number range. The Nusselt number correlations are used for comparison purpose. The CFD results are plotted and compared with analytical results. The numerical Nusselt number values are in very good agreement when compared with the correlated values.
.

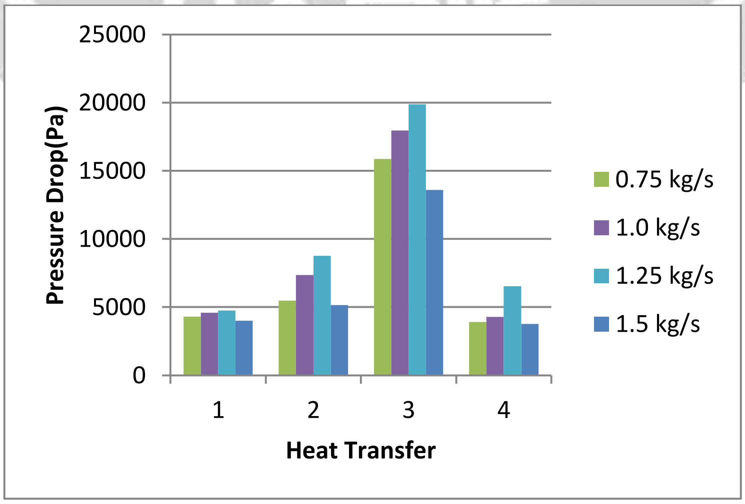

Fig. 4 Graph of Correlation of Pressure Drop

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

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.

- Investigated for augmentation of heat transfer coefficient, and temperature for the models.
- Enhanced heat transfer coefficient of 144.7 for with shell and tube model compare to the plain model.
- Increase in effectiveness of tube of about 0.29, 0.53, 0.81 and 0.12 respectively for with different flow rates compare to the plain model.
- From the results and data obtained from CFD we have concluded that turbulence model provide better suitability to our simulation. Model with existing shell and tube provides better effectiveness than other models. Heat transfer rate in the heat exchanger with tube also provides good heat transfer rate. Increasing effectiveness of heat exchanger increases its performance in its respective application.

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.