JOURNAL OF MECHANICS OF CONTINUA AND MATHEMATICAL SCIENCES
www.journalimcms.org

ISSN (Online) : 2454 -7190 Vol.-17, No.-5, May (2022) pp 56-63 ISSN (Print) 0973-8975

CFD ANALYSIS FOR HEAT TRANSFER AND PRESSURE DROP IN TUBE BUNDLE OF CROSS-FLOW HEAT EXCHANGER

Shrinjoy Sen[1], Tapas Kumar Nandi[2]

[1] Dept. of Mechanical Engineering, Techno International New Town, Kolkata-700156, India

[2] Dept. of Mechanical Engineering, Techno International New Town, Kolkata-700156, India

Correspondent Author : **Tapas Kumar Nandi**

Email: tapasnanditamal@gmail.com

https://doi.org/10.26782/jmcms.2022.05.00005

Abstract

In the present work, the preliminary finding of possibilities of heat transfer and pressure drop is reported across the shell and tube arrangement cross-flow heat exchanger. The heat exchanger consists of cold-water flows through the bundle of circular tubes and hot air across the shell. Like in the conventional arrangement, the flow in adjacent rows of tubes is normal to the fluid flow in the shell in the cross-flow arrangement. The three-dimensional turbulent flow region is modelled by employing ANSYS FLUENT 21.0. The standard k-ε model is used to model the turbulence flow. A SIMPLE algorithm scheme is applied to link the pressure and velocity fields inside the domain for air fluids. The heat transfer in the water inside the tubes is represented by a convective boundary condition. The tube flow Reynolds number was fixed at 2200 and the shell flow Reynolds number was varied from 6000 to 10000 in the turbulent zone. The purpose of this paper is to determine temperature reduction and pressure drop across the tube bundle. The simulation will predict the temperature of the airstream at the heat exchanger exit and the pressure drop. The results indicated that there is a significant amount of temperature drop in the air that releases the heat due to forced convection and temperature drop continues in the turbulent region of the incoming fluid.

Keywords: Cross flow heat exchanger, Temperature drop, Pressure drop, Turbulent flow

I. Introduction

The objective of using a heat exchanger is to transfer as much heat as possible for as small a cost as necessary. It has a wide application in the industry like a petrochemical plant, natural gas processing, wastewater treatment ad also in the home application, Refrigeration – air-conditioning and automobile sector. In many cases involving the sizing or selection of a particular exchanger is dependent on the physical properties of the fluid and its inlet temperature. There are many different types of heat exchangers, including double type, shell and tube type, cross flow and plate and frame. The simulation method presented here is applied to the cross-flow heat exchanger.

The term cross flow means that the fluids flow at right angles to each other as they pass through the exchanger. Each fluid stream can pass through and remain unmixed and mixed. In an unmixed flow situation, the flow channel or passageway would contain internal channels i.e. tubes or walls that restrict lateral fluid motion. In a mixed flow passageway, internal channels are not present and fluid particle mixes and move across the cross-section.

Heat transfer and friction data have been obtained experimentally for gas to liquid cross-flow heat exchangers and catalogued for use in predicting the performance of such exchangers (Kays W.M, London A.L).

Air to air cross-flow heat exchanger is more desirable in the industry than the water type because of their low maintenance and operating cost. The heavy-duty electric motor in the industry is cooled effectively by this type of cross-flow heat exchanger by removing heat from its winding (Paikert, P).

The performance characteristics of a tubular heat exchanger like temperature difference and pressure drop are predicted experimentally. The performance of HE is optimized by modifying the design in terms of the tube diameter and number of tubes (Alok Vyas et al).

Different active and passive techniques are incorporated to increase the heat transfer rate. One such technique is inserting the number of baffles to increase the surface area and enhance the heat transfer rate on the shell side of the tube side which increased pressure drop (Dewan, A et al).

CFD analysis of plate heat exchanger by using Ansys is conducted by Abdul Taufiq (Taufiq. A, Dhakar. P.S).

Two types of material structural steel and copper for optimization of better heat transfer rate studied in Ansys and optimized the best possible value of temperature variations amongst the discussed materials. It is observed that Copper material gives better heat transfer in comparison with structural steel. It is also concluded that increasing the thickness of the plate from 3.5 mm to 5mm enhances the heat transfer rate.

Computational studies of the staggered and double cross-flow heat exchanger are carried out at varying Reynolds numbers. The study reveals that the proposed configuration gives a maximum increase of about 27 per cent in the heat transfer rate per unit pressure drop over the conventional one (Krishnan, A.S, Gowtham, P).

Hence, many researchers have studied numerically and experimentally to enhance the heat transfer characteristic and pressure drop by a different technique. The present paper is an extension of their study of the larger fluid region in the turbulent flow where a three-dimensional simulation model is analysed numerically to find the increase of the temperature reduction and pressure drop at the exit of the shell.

II. Problem Formulation

II.i Model

Fig.1 shows the geometry of the model. The fluid in both the tube and shell are assumed with constant properties. The flow is steady, 3D and incompressible with no viscous dissipation and viscous work.

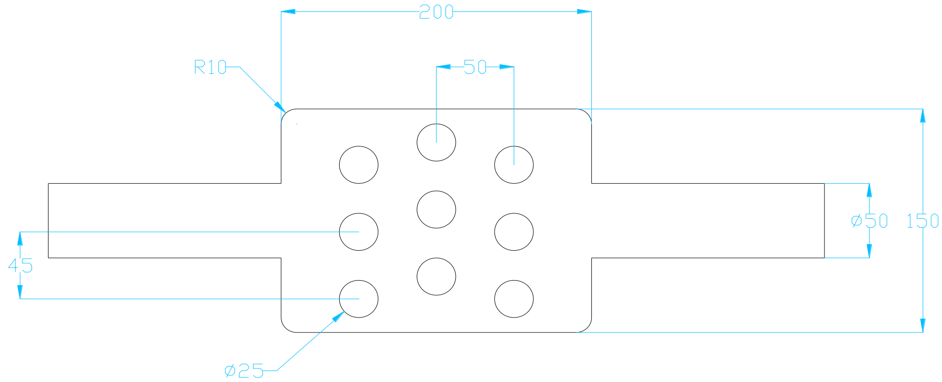

Fig.1. Schematic diagram of cross-flow heat exchanger

II.ii. Governing equation

A structured mesh was developed for the solution domain by using the mesh generation facility. The numerical simulation was performed by solving the time-dependent, continuity momentum and energy equations for a compressible fluid as shown below:

- Continuity Equation: -
  $$\nabla \bar{u} = 0 \tag{1}$$
- Momentum equation: -
  $$\rho_f \frac{D\bar{u}}{Dt} = \nabla P + \mu \nabla^2 \bar{u} \tag{2}$$

Energy Equation:

- Convection: Solid to Fluid Interface: -
- $$\rho_f {C_P}_f \frac{DT}{Dt} = k_s \nabla^2 T \tag{3}$$
- Conduction: - Solid Region
  $$\rho_s c_{ps} \frac{\partial T}{\partial t} = k_s \nabla^2 T \tag{4}$$

II.iii. Boundary condition and Meshing

The usual no-slip boundary conditions are applied at the wall. Both shell and tube side fluid is assumed to be incompressible. The shell side fluid is considered as air which is flowing at 200 l/min and 40^{0}C, in a turbulent region. The Reynolds number of the hot air entering the shell varies from 6000 to 10000. The cool water is flowing on the tube side at 10^0C with a heat transfer coefficient of 400 W/m^{2}K and the velocity of water in the tube is considered constant, in the laminar region. The heat exchanger is divided into two domains solid and fluid and they are discretized to have better control over nodes. The fluid mesh is made finer for better results during the numerical solution process.

III.iv. Numerical Procedure

The governing equations are discretized with an aim to reduce the governing partial differential equation into a set of an algebraic equations. The governing equations were worked out using the SIMPLE algorithm, a finite volume -formulation method in the computational process. The calculation domain is divided into several non-overlapping control volumes such that there is one control volume surrounding each grid point. The control volume integration forms the key step of the control volume method. The differential equations are integrated over each control volume. This method guarantees automatically the integral conservation of quantities such as volume fraction, mass, momentum and energy are exactly satisfied over any group of control volumes and of course, over the whole calculation domain. The fundamental governing equations are continuity, momentum and energy equations which are derived from basic principles of heat and fluid flow. The continuity and momentum equations are used to calculate the velocity vector. The energy equation is used to calculate temperature distribution and tube wall heat transfer coefficient.

III.v. Grid Independence Test

The grid independence test has been carried out and the grid is refined by mixed types of cells i.e. tetra, tetrahedron and triangular cell at tube wall boundaries. In the present study, grid independence has been carried out to ensure that the simulated results are independent of grid spacing. A structured mesh was developed for the solution domain after performing a rigorous grid independence check. Grid independence was ensured by varying the number of cells at different Re within the range. There is no significant change in heat transfer rate with different Re at varying numbers of cells as shown in Fig.2.

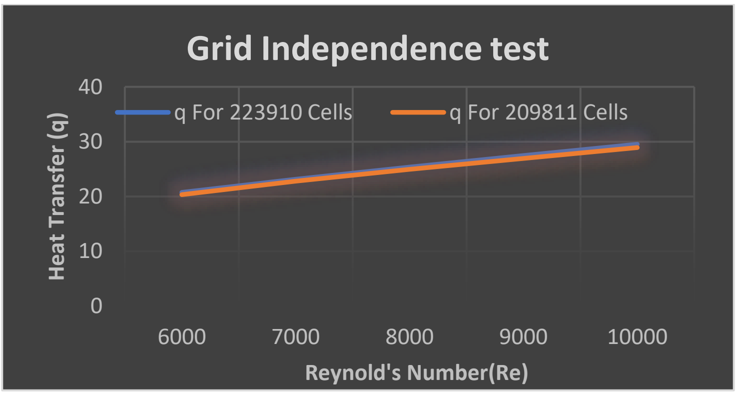

Fig.2. Variation of heat transfer rate with Re at varying number of cells.

III. Results and Discussion

In the present work, investigations were performed within the range of 6000 ≤Re ≤ 10000. The pressure and temperature drop were monitored at several Reynolds numbers. The monitored data is plotted in the below-mentioned figures to establish that heat transfer increases between the hot & cold fluid at a higher value of Re within the range.

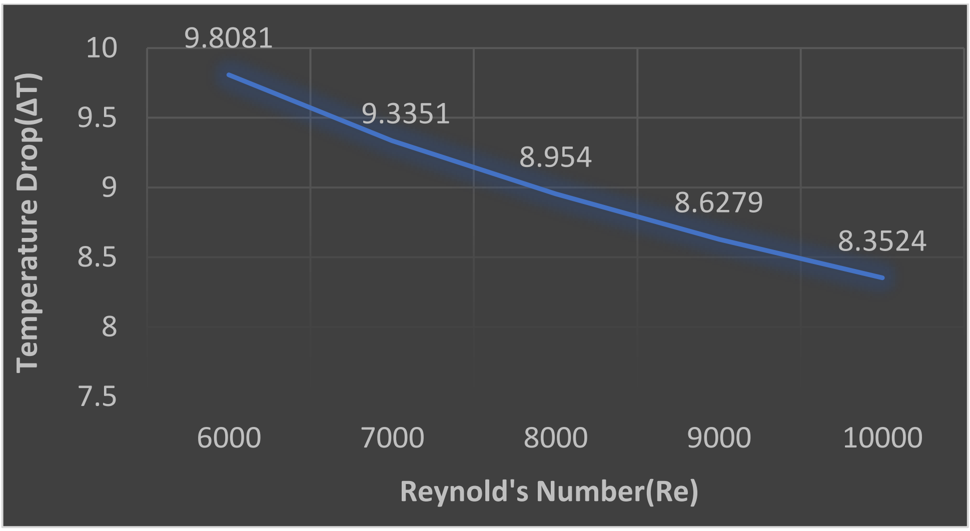

Fig. 3. Variation of Temperature Drop with Reynolds Number

In the turbulent region, it is observed more temperature drop between the inlet and outlet of hot fluid of the heat exchanger. The heat transfer rate increases with the temperature drop across the tube bundle at a constant heat transfer coefficient. When heat transfer almost reaches near-saturated conditions, the temperature drop is observed to decrease with the increase of the Reynolds number as shown in Figure 3. Hence. At the starting of turbulent flow, the heat transfer across the tube bundle is observed more and it goes slow at higher Re, in a turbulent region.

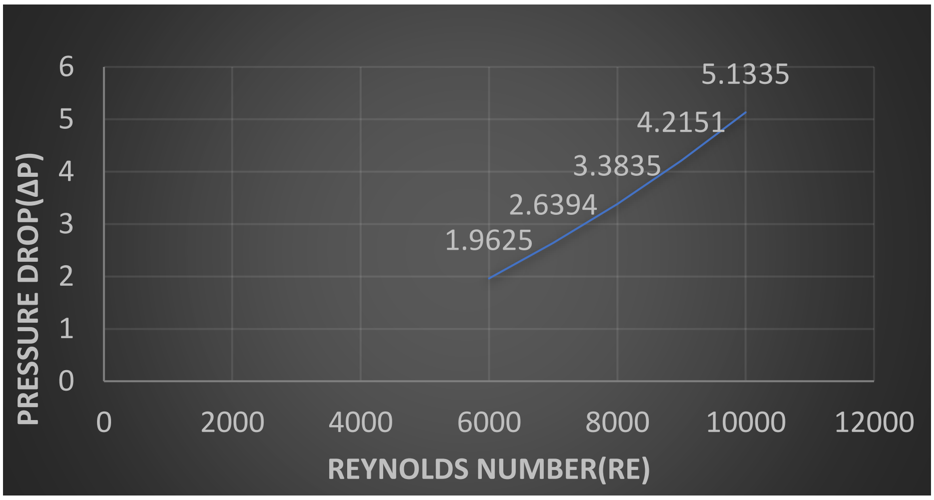

Fig.4. Variation of Pressure Drop with Reynolds Number

Figure 4 shows that Pressure drop(ΔP) increases with the increase of the Reynolds number. As the airflow enters with a zero angle of attack, there is a significant increase in pressure losses. As the flow continues to pass around the tubes, the flow is starting to form downstream large wakes, which increasingly affect the pressure drop through the heat exchanger. The increase of pressure drop between the inlet & outlet of the shell side increases the heat transfer coefficient. With the increase of pressure drop friction coefficient increases which enhances the heat transfer across the tube bundle.

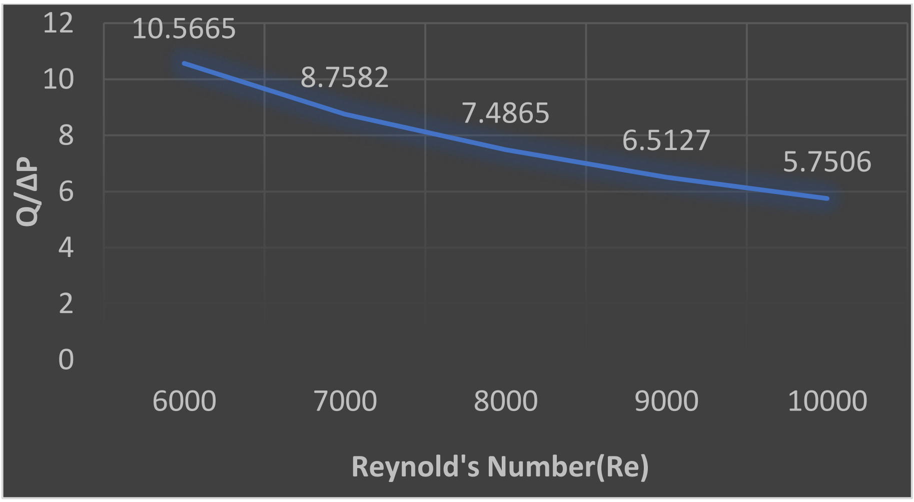

Fig.5. Variation of heat transfer rate per unit Pressure Drop for turbulent flow with Re

Referring to figure 5 it is observed that the heat transfer rate per unit pressure drop decreases with the increase of the Reynolds number. At the lower side of Reynolds numbers, wherein no noticeable vortices are formed, the cross-flow arrangement helps in a better heat transfer rate.

Lesser directional changes lead to reduced pressure drop for the range of Reynolds number considered for the study. On the higher side of Reynolds numbers supporting the formation of vortices, breaking up of vortices could be better achieved by the cross-flow arrangement and more pressure drop is observed as shown in figure 6.

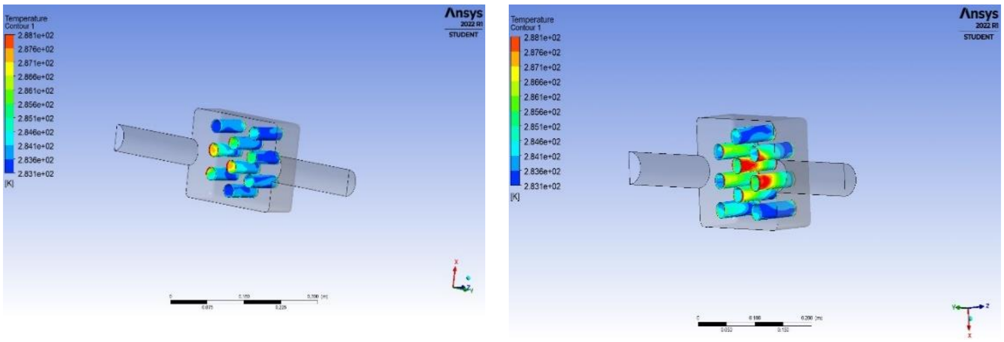

Fig. 6. Temperature Contour Re = 6000 Temperature Contour Re = 10000

At Re=6000, the heat exchange from hot air to cold fluid is not significant. At Re=10,000, the heat exchange from hot air to cold fluid is very much significant.

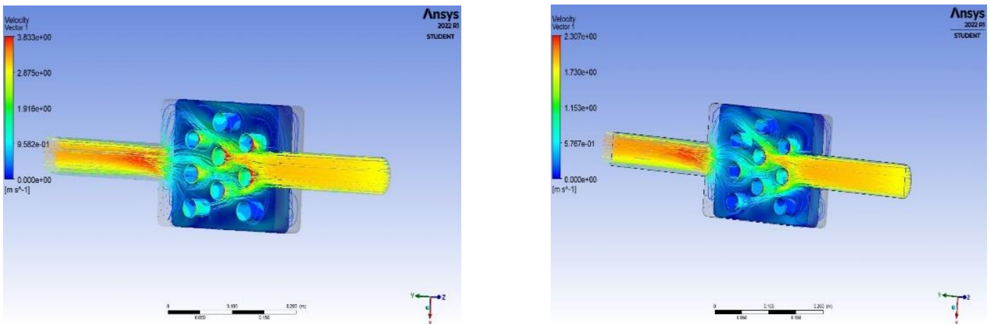

Fig.7. Velocity streamline contour at Re = 6000 Velocity streamline contour at Re = 10000

At the lower side of Re, the intensity of wake formation after the tube is not significant. Low-pressure drop and heat transfer rate are comparatively not marginal. At the higher side of Re, the intensity of wake formation after the tube is significant. Higher pressure drops and heat transfer rates are comparatively marginal as in figure 7.

IV. Conclusion

A numerical investigation of the fluid flow and heat transfer phenomena in cross-flow heat exchangers was conducted for observing the temperature reduction and pressure drop across the tube bundle. The three-dimensional convective heat transfer calculations are performed in a turbulent region with an incoming varying Reynolds number. The heat transfer rate per unit pressure drop between the hot air and cold fluid is calculated which is observed as increasing with the increase of the Reynolds number.

Future work could include that this work can be validated experimental and for more study, the baffles can be designed on the tube wall to enhance the heat transfer effect of the heat exchanger.

Conflicts of Interest:

The authors declare that they have no conflicts of interest regarding this article.

References

I. Alok Vyas et al. (2013). An Experimental Analysis Study to Improve Performance of Tubular Heat Exchanger. Journal of Engineering Research and Applications, Vol. 3, Issue 6, pp.1804-1809.

II. A. Dewan, P. Mahanta, Sumithra, K. Raju, Suresh, P. Kumar. (2004). Review of passive heat transfer augmentation techniques. *Proc. Instn. Mech. Engrs, Part A – J. Power Energy*, **218**(7), 509-526.

III. A.S. Krishnan, P. Gowtham. (2017). Computational study of the staggered and double cross flow heat exchanger, Defence Science Journal, Vol. 67, No. 4, pp. 396-400.

IV. A. Taufiq, P.S. Dhakar. (2020). CFD analysis of plate heat exchanger by using Ansys, International Journal of Research and Analytical Reviews (IJRAR), Volume 7, Issue 3.

V. P. Paikert. (1986). Air cooled heat exchanger: thermal and hydraulic design of heat exchanger. Vol. 3, Hemisphere Publishing Corporation.

VI. W.M. Kays, A.L. London. (1984). Compact Heat Exchanger 3rd edition, McGraw-Hill Book Co.