International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 4 Issue 04, April-2015

Heat Transfer Enhancement in Concentric Tube Heat Exchanger in ANSYS FLUENT

[1]Mayank Bhola
M.Tech scholar: Dept. of Mechanical Engg.
Bipin Tripathi Kumaon Institute of Technology,
Dwarahat, Almora, India

[2]Vinod Kumar
Assistant Professor: Dept. of Mechanical Engg.
Bipin Tripathi Kumaon Institute of Technology,
Dwarahat, Almora, India

[3]Dr. Satyendra Singh
Associate Professor Dept. of Mechanical Engg.
Bipin Tripathi Kumaon Institute of Technology,
Dwarahat, Almora, India

***Abstract*— The study deals with CFD simulation of concentric tube heat exchanger and concentric tube heat exchanger with insert used for heating air using ANSYS FLUENT Software for steel. Design process for heat exchanger and insert has been carried out in SOLIDWORKS, fluid domain is formed in ANSYS workbench, followed by meshing in default mesh tool of ANSYS and solution is developed using ANSYS FLUENT software as FINITE ELEMENT TOOL and the results are compared between the two designs for parallel flow. The Reynolds number of air varied from 21000 to 100000. Inlet temperature of hot water and cold air are 60̊ C and 26̊ C respectively. The work included the determination of Nusselt number, heat transfer coefficient and friction factor for insert in parallel flow and counter flow**.

Keywords— ***Heat Exchanger, Insert, ANSYS FLUENT, CFD , FINITE ELEMENT TOOL***

I. INTRODUCTION

The analysis of heat exchanger is of great significance from engineering point of view due to various engineering applications and implications dealt with it. Considerable significance has been made on the development of various augmented heat transfer surfaces and devices, in recent years. Energy and material saving reconsideration, space considerations as well as economic incentives have led to the increased efforts aimed at producing more efficient and reliable heat exchanger equipment through the augmentation of heat transfer [2][9]. Among many techniques investigated for augmentation of heat transfer rates inside circular tubes, a wide range of inserts have been utilized, particularly when turbulent flow is considered [2]. Enhanced performance of heat exchanger enables the size of the heat exchanger to be decreased [9][3]. In tube heat exchanger design the tube often represents poor performance when handling viscous liquids in laminar flow because near the tube wall, there is thermally inefficient boundary layer with very little mixing. Since heat transfer is controlled principally by the thickness of the boundary layer and its thermal conductivity. A very poor heat transfer coefficient results, so need of augmentation is generated [4]. Inserts continually remove low velocity fluid from the tube wall and replace it with fluid from the centre of the tube. By breaking up the boundary layer at the wall and promoting radial mixing of the tube side fluid, these inserts increases the heat transfer coefficient dramatically for a given pressure drop. These increases could be as large as 20 times for flow at very low Reynolds numbers [4]. The flow of real fluid exhibits viscous effects in pipe flow. In this research paper the effect is identified for turbulent flow. Continuity equation, momentum equation, viscous model and energy equations are used for solving the heat exchanger model.

II. LITERATURE REVIEW

Cylindrical pipes are used very extensively in a lot of heat transfer and engineering applications. They have found extensive use in various types of Heat Exchangers, in Automobile, in thermal power plants. Recently many emphasize has been made to increase the heat transfer characteristics of concentric tube heat exchanger. [1] Giakward et al, (2014) investigated thermal performance of double pipe heat exchanger for laminar flow using twisted wire brush insert which are fabricated by winding a 0.2 mm diameter of the copper wires over a 2 mm diameter two twisted iron core-rods. Author concluded that the Nusselt number for the tube with twisted wire brush insert varied from 1.55 to 2.35 times in comparison of those of the plain tube. [2] Sarada et al, (2010) investigated heat transfer in a horizontal circular tube using mesh insert in turbulent region. Author concluded that maximum Nusselt number obtained at smallest pitch of larger mesh diameter using CFD analysis which is 2.15 times that of plain tube. [8] Jamra et al, (2012) investigated heat transfer enhancement in double pipe heat exchanger using simple pattern of rectangular insert. They observed that the heat transfer coefficient varied from 1.9 times the smooth tube values. [3] Pardhi et al, (2012) investigated performance improvement of double pipe heat exchanger by using two different twisted tape as turbulator. Conclusion of this work was that the heat transfer coefficient increased by 61% for twisted tape 1 and 78% for twisted tape 2.[4] Patil et al, (2011) investigated thermohydraulic performance of tube in tube heat exchanger using twisted tape with winglets. Author concludes that twisted tape insert mixes the bulk flow well and therefore performs better in laminar flow, because in laminar flow the thermal resistant is not limited to a thin region. [5] Omkar et al, (2014) investigated double pipe heat exchanger with helical fins on the inner rotating tube. Author concluded that the Nusselt number increased up to 64% at 100 rpm compared to stationary inner tube with helical fins. [6] Al-Kayim et al, (2011) analytically investigated the thermal performance of double pipe heat exchanger with ribbed inner tube, an enhancement of 4 times in the heat transfer in terms of Stanton number is achieved. [7] Pachegaonkar et al, (2014) investigated the performance of double pipe heat exchanger with annular twisted tape insert. Concluded that swirl flow helps decrease boundary layer thickness of the cold water flow and increase residence time of water in the inner tube.

III. MATHEMATICAL FORMULATION

The system consists of concentric tube heat exchanger with and without insert for heating air through water. The geometric model of the heat exchanger were constructed using design software SOLIDWORKS. In order to numerically establish the heat transfer coefficient of heat exchanger with insert the parameters were assumed to be same that of heat exchanger without insert. Tube diameter was considered to be 0.015 m and length considered was 2.5 m. The three dimensional computational domain is modeled using quad mesh for both models. The flow is assumed to be steady and turbulent. In this numerical investigation, the following hypotheses are adopted.
(i) Physical properties of water are constant.
(ii) Profile of velocity is uniform at the inlet.
(iii) The radiation heat transfer is negligible.
(iv) The flow is assumed to be steady.

IV. GOVERNING EQUATIONS

A. Continuity Equation

Continuity equation also called conservation of mass. Consider fluid moves from point 1 to point 2. The overall mass balance is input – output = accumulation. Assuming that there is no storage the mass input = mass output. However, as long as the flow is steady (time-invariant), within this tube, since, mass cannot be created or destroyed. According to continuity equation, the amount of fluid entering in certain volume leaves that volume or remains there and according to momentum equation tells about the balance of the momentum. The momentum equations are sometimes also referred as Navier-Stokes (NS) equation. They are most commonly used mathematical equation to describe flow. The simulation is done based on the NS equation and then K-Epsilon model.

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}(\rho v_x) + \frac{\partial}{\partial y}(\rho v_y) + \frac{\partial}{\partial z}(\rho v_z) = 0$$

B. Kappa-Epsilon Model

The K-epsilon model is most commonly used to describe the behavior of turbulent flows. It was proposed by A.N Kolmogrov in 1942, then modified by Harlow and Nakayama and produced K-Epsilon model for turbulence. The Transport Equations for K-Epsilon model are for k, Realizable k-epsilon model and RNG k-epsilon model are some other variants of K-epsilon model. K-epsilon model has solution in some special cases. K-epsilon model is only useful in regions with turbulent, high Reynolds number flow.

K – **Equation**

$$\rho[\bar{u}\frac{\partial k}{\partial x} + \bar{v}\frac{\partial k}{\partial r}] = \frac{\partial}{\partial x}[(\mu_l + \frac{\mu_t}{\sigma_k})\frac{\partial k}{\partial x}] + \frac{1}{r}\frac{\partial}{\partial r}[r(\mu_l + \frac{\mu_t}{\sigma_k})\frac{\partial k}{\partial r}] + \rho g - \rho\varepsilon$$

Where, G is the production term and is given by

$$G = \mu_t[2\{(\frac{\partial \bar{v}}{\partial r})^2 + (\frac{\partial \bar{u}}{\partial x})^2 + (\frac{\bar{v}}{r})^2\} + (\frac{\partial \bar{u}}{\partial r} + \frac{\partial \bar{v}}{\partial x})^2]$$

The production term represents the transfer of kinetic energy from the mean flow to the turbulent motion through the interaction between the turbulent fluctuations and the mean flow velocity gradients.

ε - **Equation**

$$\rho[\bar{u}\frac{\partial \varepsilon}{\partial x} + \bar{v}\frac{\partial \varepsilon}{\partial r}] = \frac{\partial}{\partial x}[(\mu_l + \frac{\mu_t}{\sigma_\varepsilon})\frac{\partial \varepsilon}{\partial x}] + \frac{1}{r}\frac{\partial}{\partial r}(r\mu_l + \frac{\mu_t}{\sigma_\varepsilon})\frac{\partial \varepsilon}{\partial r}]$$

$$+ C_{S1}G\frac{\varepsilon}{k} - C_{s2}\frac{\varepsilon^2}{k}$$

Here C_{s1}, C_{s2}, σ_k and σ_ε are the empirical turbulent constant. The values are considered according to the Launder *et al.*, 1974. The values of Cμ, C_{s1}, C_{s2}, σ_k and σ_ε are 0.09, 1.44, 1.92, 1.0 and 1.3 respectively.

C. Energy Equation

The conservation form of energy equation written in terms of total energy is presented below.

$$\rho\frac{DE}{Dt} = -div(pu) + [\frac{\partial(u\tau_{xx})}{\partial x} + \frac{\partial(u\tau_{yx})}{\partial y} + \frac{\partial(u\tau_{zx})}{\partial z} + \frac{\partial(v\tau_{xy})}{\partial x} + \frac{\partial(v\tau_{yy})}{\partial y} + \frac{\partial(v\tau_{zy})}{\partial z} + \frac{\partial(w\tau_{xz})}{\partial x} + \frac{\partial(w\tau_{yz})}{\partial y} + \frac{\partial(w\tau_{zz})}{\partial z} + div(k\, grad\, T) + S_E]$$

D. Boundary Condition

A turbulent flow is considered. The quantities *U, k, ε* are obtained by using numerical calculations based on the *k-ε* model for high Reynolds Number. The boundary conditions are listed below:

1) At the inlet of the channel:

$$u = U_{in,} v = 0$$

$$k_{in} = 0.005 U_{in}^{\ 2}$$

$$\varepsilon_{in} = 0.1 K_{in}^{\ 2}$$

K_{in} stands for the admission condition for turbulent kinetic energy and ε_{in} is the inlet condition for dissipation.

2) At the walls:

$$u = v = 0$$

$$k = \varepsilon = 0$$

3) At the exit:

$$P = P_{atm}$$

The Reynolds number based on circular diameter in case of circular tube and hydraulic diameter D_h in case of rectangular tube.

$$\mathbf{Re} = \frac{\rho.U_0.D_h}{\mu}$$

V. MODELLING AND SIMULATION

The whole analysis is carried out with the help of software "ANSYS FLUENT 14.5". ANSYS FLUENT 14.5 is computational fluid dynamics (CFD) software package to stimulate fluid flow problems. It uses the finite volume method to solve the governing equations for a fluid geometry and grid generation is done using the pre-processor bundle with FLUENT. The three dimensional computational domain is modeled using quad mesh for models. The complete domain of concentric tubes without insert consist of 325239 nodes and 590698 elements and domain with insert consist of 2256549 nodes and 1941317 elements. Grid independence test was performed to check the validity of the quality of the mesh on the solution. Further refinement did not change the result by more than 0.9% which is taken as the appropriate mesh quality for computation.

VI. VALIDATION OF MODEL

In the present paper concentric tube heat exchanger with and without insert was modeled and simulated using computational fluid domain for heating cold air by applying fixed wall temperature boundary conditions. Heat transfer parameters like temperature drop, heat transfer rate, heat transfer coefficient, Nusselt number and pressure drop were calculated. Simulation results were compared with analytical results using the correlations developed by different researchers. Also the simulation results of the concentric tube heat exchanger without insert were compared with the results obtained for concentric tube heat exchanger with insert of equal length and similar operating conditions in order to compare its performance related to heat transfer characteristics.

Table 6.1 Geometric description of concentric tubes and insert

Parameter	Value
Length of the heat exchanger L (m)	2.5
Diameter of inner pipe d (m)	0.025
Diameter of annulus space D (m)	0.022
Diameter of rod fixed with insert d_{in} (m)	0.004
Width of insert w (m)	0.001
Height of insert h (m)	0.006
Length of insert l_{in} (m)	0.005
Length of heat exchanger upon which inserts are acting l (m)	2.4
Inlet water temperature (K)	333
Inlet air temperature (K)	299

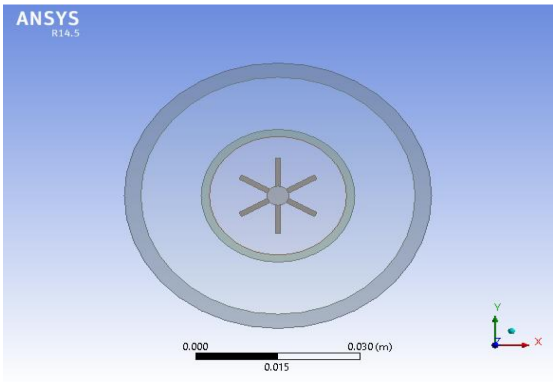

Fig 6.1 concentric tube heat exchanger with insert

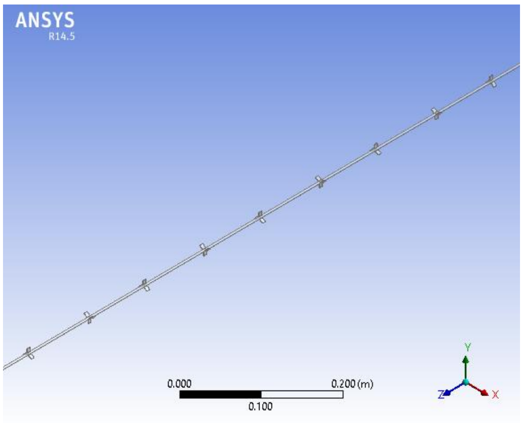

Fig 6.2 Insert pattern

Table 6.2 Properties of air and water

Properties	Air	Water
Density (kg/m^3)	1.155	985
Specific heat C_p (J/kg-k)	1005	4183
Thermal conductivity (W/m-k)	0.02655	0.6153
Viscosity (kg/m-s)	1.845e^{-05}	4.7083e^{-04}

VII. RESULTS AND DISCUSSION

CFD computations were done for three different mass flow rate of air (0.0079, 0.018, 0.036 kg/s) and water (0.0216, 0.36091, 0.6767 kg/s) for both parallel flow and counter flow condition. Parameters adopted for comparison are heat transfer coefficient, Nusselt number and friction factor. In order to validate the CFD results important factor like Nu was calculated by using the correlation for plain tube. Fig 7.1 shows the CFD simulated heat transfer coefficient vs. mass flow rate plot for two cases. Heat transfer coefficient corresponding to the counter flow is higher than that for parallel flow. This is because of the better mixing of fluid particles provided by insert and increase in contact time. In rectangular insert, it was observed that the heat transfer coefficient varied from 1.15 to 1.4 times for parallel flow and 1.2 to 1.5times for counter flow that of the plain tube. Fig 7.2 shows the CFD simulated Nusselt number vs. mass flow rate plot for two cases. The results have shown a good agreement. Fig 7.3 shows the comparison of friction factor for two cases with plain tube. Friction factor for tubes with insert is found to be more than that of plain tube for all mass flow rates. Presence of insert generates secondary flow which dissipates kinetic energy, thus increasing the resistance to flow. For lower mass flow rate pressure drop varies linearly whereas on increasing the mass flow rate pressure drop varies exponentially as seen in the graph.

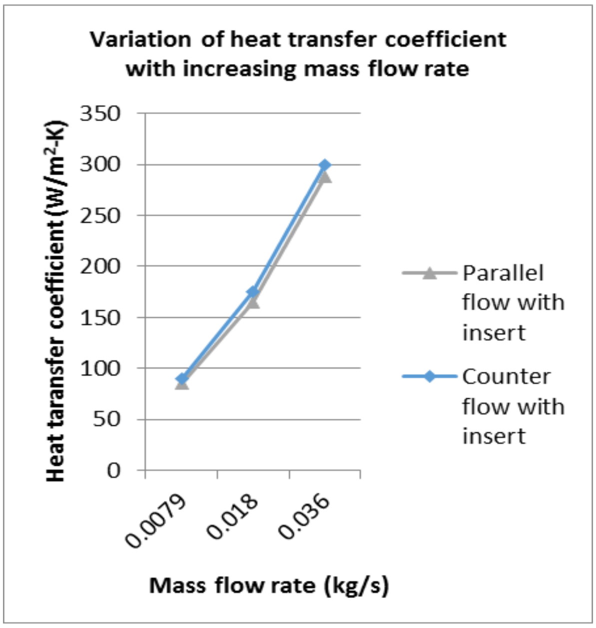

Fig 7.1 Variation of heat transfer coefficient with increasing mass flow rate

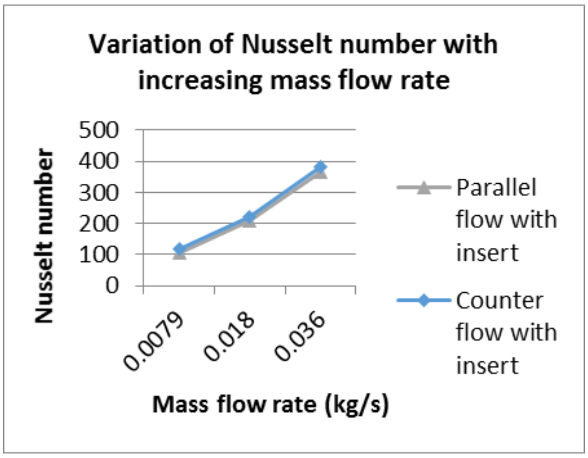

Fig 7.2 Variation of Nusselt number with increasing mass flow rate

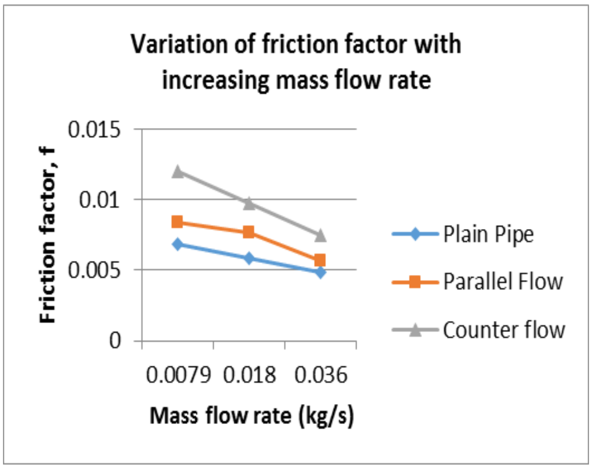

Fig 7.3 Variation of Friction Factor with increasing mass flow rate

VIII. CONCLUSION

The results showed a trend of increase in heat transfer with the provision of insert on the heat exchanger. The heat transfer was found to increase as the Reynolds number was varied over the range.

The results obtained show that the effect of insert on the enhancement of heat transfer depends on both the pattern of insert and the Reynolds number of the flow.

The analytical results obtained by the ANSYS fluent software, are presented to analyze the heat transfer enhancement.

Based on the CFD analysis the following conclusions can be drawn.

- The heat transfer enhancement effect is primarily due to induced turbulence which gives higher heat transfer rates.
- At higher Reynolds number more temperature increment can be attained.
- By using such kind of insert length of Heat exchanger can be minimized, for high Reynolds number applications.
- The inner convective heat transfer coefficient for rectangular insert of this kind is approximately 25% higher than for plain tube.

Counter flow arrangement is more efficient than parallel flow. And show increase in heat transfer coefficient of approximately 27%.

REFERENCES

1. Gaikward, Mali, “Heat Transfer Enhancement for Double Pipe Heat Exchanger Using Twisted Wire Brush Insert” IJIRSET, Vol. 3, Issue 7, ISSN: 2319-8753
2. S.N. Sarada, A.V.S.R. Raju and K.K. Radha, “Experimental and Numerical Analysis of Turbulent Flow Heat Transfer Enhancement in a Horizontal Circular Tube Using Mesh Inserts” Journal of Energy and Power Engineering, ISSN: 1934-8975
3. Pardhi, Baredar, “Performance Improvement of Double Pipe Heat Exchanger by Using Turbulator” IJESAT, Vol. 2, Issue 4, ISSN: 2250-3676
4. A.M.Patil, S.D.Patil, “Analysis of Twisted Tape with Winglets to Improve the Thermohydraulic Performance of Tube in Tube Heat Exchanger” IJMEAR, Vol. 2, Issue 2, ISSN: 2249-6564
5. Shewale Omkar M, Mane Pravin A, Gazge Sajid A Hameed, Pasanna Pradeep A “Experimental Investigation of Double Pipe Heat Exchanger with Helical Fins on the Inner Rotating Tube” IJRET, Vol. 3, Issue 7, ISSN:2319-1163
6. Hussain H.AL-Kayiem, Mohamed F.EL-Rahman “ Ribbed Double Pipe Heat Exchanger: Analytical Analysis” Journal of Engineering Science and Technology, Vol. 6, No. 1
7. Snehal S. Pachegaonkar, Santosh G. Taji, Narayan Sane “Performance Analysis of Double Pipe Heat Exchanger with Annular Twisted Tape Insert,” IJEAT , Vol. 3, Issue 3, ISSN: 2249-8958
8. Sunil Jamra, Pravin Kumar Singh, Pankaj Dubey , “Experimental Analysis of Heat transfer Enhancement in Circular Double Tube Heat Exchanger Using Inserts,” IJMET, Vol. 3, Issue 3, pp. 306-314
9. Bisht Priyanka, Joshi Manish, and Dr Gupta Anirudh, “comparison of heat transfer between a circular and rectangular tube heat exchanger by using ANSYS FLUENT”, International Journal of thermal technologies Vol-3 ISSN: 2277 – 4114