International Journal on Emerging Technologies (Special Issue NCETST-2017) **8**(1): 277-280(2017)
(Published by Research Trend, Website: www.researchtrend.net*)*

ISSN No. (Print) : 0975-8364
ISSN No. (Online) : 2249-3255

Effect on Total Heat Transfer Rate in Concentric Tube Heat Exchanger with Rectangular Insert in ANSYS FLUENT 14.5 for Varying and Constant Flow

Mayank Bhola, Shivasheesh Kaushik, Ashis Saxena
Assistant Professor, Mechanical Engineering Department, Amrapali Group of Instiutes, Haldwani

ABSTRACT: The research deals with CFD simulation of concentric tube heat exchanger with rectangular insert used for heating air using ANSYS FLUENT Software inside steel tube. Design for heat exchanger and insert is carried out in SOLIDWORKS, fluid domain is made 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 mass flow rate of air varied from 0.0079 to 0.036 kg/s and for water it is kept constant at 0.0216 kg/s and then varied from 0.0216 to 0.67 kg/s. Inlet temperature of hot water and cold air are 333 K and 299 K respectively. The work included the determination of total heat transfer rate for insert in parallel flow for constant and varying mass flow rate of hot water.

Keywords: Heat Exchanger, Insert, ANSYS FLUENT, CFD , FINITE ELEMENT TOOL

I. INTRODUCTION

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.[10] Bhola *et al,* (2015) conducted a numerical simulation based on reynolds no. dimensions of heat exchanger used by Jamra et al, and found that heat transfer increase with increase in reynolds no. of both air and water. Present study is carried out to find the effect on total heat transfer rate when flow of hot water is kept constant for varying flow of cold water, to find that whether reynolds no. of hot fluid be varied or not in order to achieve better heat transfer for varying reynolds no. of cold fluid.

II. MATHEMATICAL FORMULATION

The system consists of concentric tube heat exchanger with insert for heating air through water flowing inside steel tubes. The geometric model of the heat exchanger were constructed using SOLIDWORKS design software. Tube thickness 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. The following hypotheses are adopted In this numerical investigation,.

(i) Water has constant physical properties.

(ii) Uniform velocity profile at inlet.

(iii) The radiation heat transfer is negligible.

(iv) The flow is steady.

Governing Equation

1 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.

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

2 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**

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

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

3 Energy Equation

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

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

4 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:

$u = v = 0$

$k = \varepsilon = 0$

2) 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.

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

Modelling and simulation

The whole analysis is carried out with the help of "ANSYS FLUENT 14.5"software. Which is a 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 with insert consist of 2250000 nodes and 1940000 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 1% which is taken as the appropriate mesh quality for computation. Concentric tube heat exchanger with insert was modeled and simulated using computational fluid domain for heating cold air by applying appropriate boundary conditions. Second order upwind scheme was applied for convective and turbulent terms. To evaluate the pressure field, the pressure–velocity coupling algorithm SIMPLE (Semi Implicit Method for Pressure-Linked Equations) was selected. At the inlet, uniform velocity profile has been imposed. The turbulence intensity was kept at 10% at the inlet. The parameters of interest for present case is Total heat transfer rate Simulation results were compared for two conditions which are varying and constant mass flow rate of hot fluid.

Table 1: Geometric description of concentric tubes and insert.

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

Table 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}

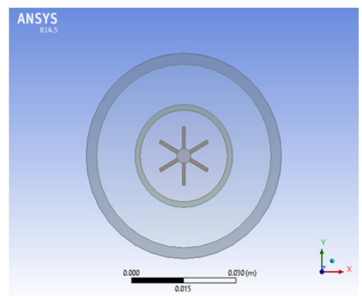

Fig. 1. Concentric tube heat exchanger with insert.

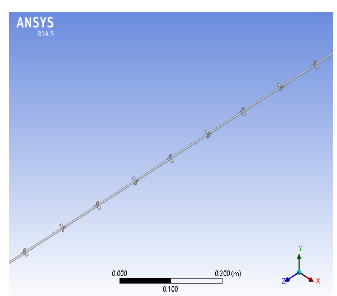

Fig. 2. Insert pattern.

III. 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 concentric tube with rectangular inserts for both constant and varying mass flow rate of hot water. Parameters adopted for comparison is total heat transfer rate.

Fig 3 shows the CFD simulated total heat transfer rate vs. mass flow rate plot for two cases. Total heat transfer rate corresponding to the heat exchanger with constant mass flow rate of hot fluid increases linearly, whereas for varying flow nonlinear increment is seen. It was observed that the difference in total heat transfer is nearly 5 % to 10 % for both the cases. And for higher mass flow rate of cold fluid, total heat transfer rate for constant flow of hot fluid is more than varying flow of hot fluid.

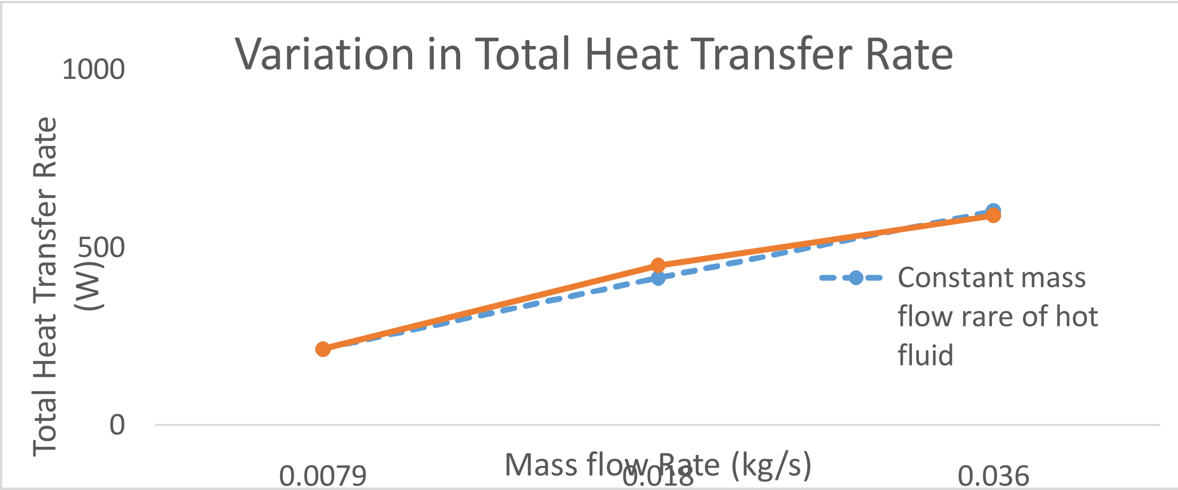

Fig. 3. Variation in total heat transfer rate for constant and varying flow

IV. CONCLUSION

The results showed a trend of increase in total heat transfer rate withboth constant and varying mass flow rate with a difference of approximately 5 % to 10 %. The results obtained show that the enhancement ofheat transfer also depends on the mass flow rate of both the working fluids. The analytical results obtained by the ANSYS fluent software, are presented to analyze the heat transfer enhancement. Based on the CFD analysis the following conclusionis drawn.

- Optimization of concentric tube heat exchanger should also be done according to constant and varying flow, so that best design corresponding to suitable outlet temperature and best total heat transfer rate can be found.

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

[10] Bhola, M; Kumar, V; Dr. Singh, S "Heat Transfer Enhancement in Concentric Tube Heat Exchanger in ANSYS FLUENT 14.5", *International Journal of Innovative Science, Engineering & Technology*, Vol. 2 Issue 3,ISSN 2348 – 7968, pp. 375-379.