Optimization of Heat Transfer Process Parameters of Tube and Tube Heat Exchanger using CFD

M. V. S. Pavan Kumar[1], **J. Vijay Kumar**[2], **Sk. Salman Basha**[3]

[1,2,3] *Asst. Professor, Dept. of Mechanical Engineering, Sasi Institute of Technology and Engineering, AP, India*

***

Abstract - *Heat exchanger is a device which is used to transfer heat from hot fluid to cold fluid. These exchangers are used in industrial purposes like chemical industries, nuclear power plants, refineries, food processing, etc. Effectiveness of heat exchanger is an important parameter while selection of industrial heat exchangers. Sizing of heat exchangers play very significant role for cost optimization. Methods for improvement on heat transfer have been worked upon for many years in order to obtain maximum effectiveness and efficiency with optimum cost. The present work carried out to optimize the process parameters of tube and tube parallel flow heat exchanger experimentally and simulation of CFD without baffle plates by changing the velocity of hot fluid, mass flow rate of cold fluid and inlet temperature of hot fluid. Finally, comparison between Experimental results and simulation results of CFD done.*

Keywords: Heat Exchanger, Effectiveness, Optimization, Parallel Flow, CFD.

1.INTRODUCTION

Heat exchangers are devices that facilitate the exchange of heat between two fluids that are at different temperatures while keeping them from mixing with each other. Heat exchangers are commonly used in practice in a wide range of applications, from heating and air-conditioning systems in a household, to chemical processing and power production in large plants. Heat exchangers differ from mixing chambers in that they do not allow the two fluids involved to mix. In a car radiator, for example, heat is transferred from the hot water flowing through the radiator tubes to the air flowing through the closely spaced thin plates outside attached to the tubes. The heat exchangers can be broadly divided into various types based on the direction of fluid motion, constructional features of the heat exchanger. According to the relative direction of two fluid streams the heat exchangers are classified into three categories. Parallel flow or unidirectional flow, Counter flow and Cross flow. The Heat Exchanger is a device used to exchange the heat energy between the two fluids by which increases the operating efficiency. These efficiencies play a major role for cost effective. Operations in the process industries. While the both fluids flow through the heat exchanger, the temperature of both fluids will exchange. The main objective of this paper is deals with the performance rate of double pipe heat exchanger. By changing the materials which uses the heat input from the waste recovery of steam in refinery process. Final Results are obtained with three different types of materials steel, aluminum and copper. As we increase the fin thickness the temperature of the cold fluid at the outlet of the heat exchanger increases [1]. Heat exchangers are the equipment used in industrial processes to recover heat between two process fluids. They are widely used in space heating, refrigeration, air conditioning, power plants, chemical plants petrochemical plants, petroleum refineries, and natural gas processing. Therefore, the objective of present work involves study of refinery process and applies phenomena of heat transfer to a double pipe heat exchanger. As we increase the fin thickness the temperature of the cold fluid at the outlet of the heat exchanger increases. The simulated outlet temperature is 543k which is very near to design outlet temperature 553k. There is less than 3% variation occurs than design value [2]. Unfortunately, in normal practice the effect of the heat transfer process on the flow patterns cannot be observed as they take place in a nontransparent metal bodies. As such, and because of the fact that direct measurements of flow characteristics can be time consuming. By taking this into consideration we can approximate the flow characteristics like temperatures using simulation software. Now a day's Computational Fluid Dynamics (CFD) is used for simulation due to low research cost, short research period and detailed description to fluid dynamic behavior, computational fluid dynamics have received rapid development in recent decades. In this study, commercial CFD software was used to simulate heat transfer process of tube and tube heat exchanger.

2. EXPERIMENTAL ANALYSIS

2.1 Experimental Setup

The apparatus of a concentric tube heat exchanger, one small diameter tube is fixed in another bigger diameter tube concentrically and outer tube is insulated to reduce the heat losses from its surface to the surroundings. The fluids which are used for transferring the heat are cold water and hot air. The cold water is supplied connecting the inlet to the common tap supply and its flow is controlled by providing inlet and outlet valves. For providing the hot air, a small blower and heater inside the tube placed respectively. The valves arrangement is made in such way, that the flow can be made parallel or counter as desired. The two thermometers are provided to measure the inlet and outlet temperatures of hot air and other two measures the inlet and outlet temperatures of cold water.

Fig: 1 Expermental setup of tube and tube H.E.

The process parameters taken into account here are inlet temperature of hot fluid, velocity of hot fluid and mass flow rate of cold fluid. To optimize the above said parameters full factorial design technique is used. Parameters are three in number and their levels are considered as two. Finally, eight combinations for the above said parameters are obtained as shown below.

Table 1: Process parameters combinations

Run Order	T_{hi} **(K)**	V_h **(m/s)**	m_c **(lpm)**
1	323	2	1
2	323	4	2
3	343	2	2
4	323	4	1
5	343	4	2
6	323	2	2
7	343	4	1
8	343	2	1

Where T_{hi} = Inlet temperature of hot fluid.

V_h = Velocity of hot fluid.

m_c = Mass flow rate of cold fluid

2.2 Mathematical Calculations

For Run-1

Effectiveness (ε)

$$\varepsilon = \frac{Q_{act}}{Q_{\max}} = \frac{m_h c_{ph}(T_{hi} - T_{ho})}{c_{\min}(T_{hi} - T_{ci})}$$

$$= \frac{4.5265 \times 10^{-3}(323 - 306)}{4.5265 \times 10^{-3}(323 - 303)}$$

$$= 0.88$$

3. COMPUTATIONAL DOMAIN

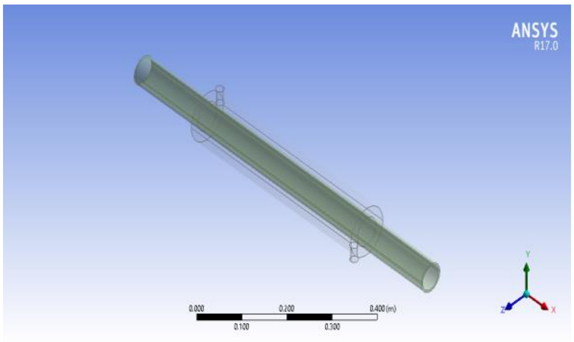


Fig 2: Geometry of tube and tube H.E.

To perform simulations, a tube and tube heat exchanger domain was created using Design Modeler of Ansys work bench. 5 cm diameter of inner tube and 10 cm diameter of outer tube length of 50 cm heat exchanger created using sketching and modeling.

4. MODELLING AND NUMERICAL SIMULATION

The simulations were performed using Multiphase Mixture model available in the commercial CFD software, ANSYS Fluent 14.5. The continuity, momentum equations are solved for phases and coupling between phases is obtained pressure and interphase exchange coefficients in solver.

4.1 Simulation Set up

4.1.1 Mesh

The meshes were constructed in the commercial CFD Software ANSYS MESHING 14.5. Ansys meshing is one of the critical aspects of engineering simulation. Too many cells result in longer solver runs, and too few may lead to inaccurate results. ANSYS meshing technology provides a means to balance these requirements and obtain the right mesh for each simulation in the most automated way possible. Fluid dynamics simulations require very high-quality meshes in both element shape and smoothness of size changes. There are different types of mesh types are available.

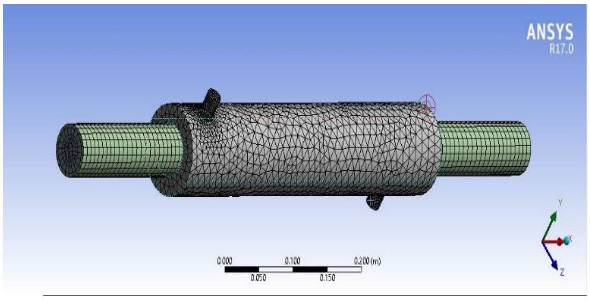


Fig 3: CFD Meshes for tube and tube H.E.

4.1.2 Physical Models and Boundary Conditions

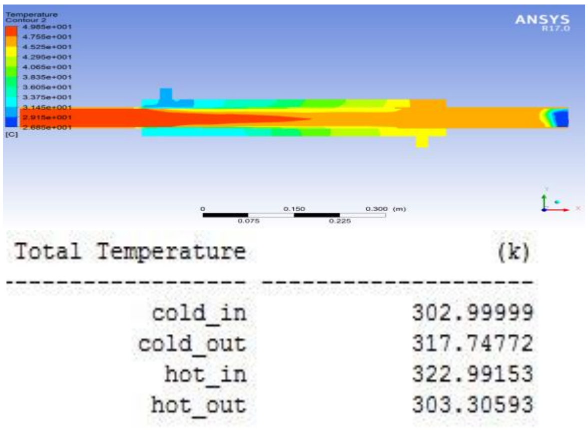
The analysis type is changed to Pressure Based type. The velocity formulation is changed to absolute and time to steady state. Gravity is defined as y = -9.81 m/s^2. Energy is set to ON position. Viscous model is selected as "k-ε model (2 equations). Radiation model is changed to Discrete Ordinates. The create/edit option is clicked to add water-liquid and steel to the list of fluids and solid respectively from the fluent database. Boundary conditions are used according to the need of the model. The inlet and outlet conditions are defined as velocity inlet and temperature inlet and pressure outlet. As this is a parallel-flow with two tubes so there are two inlets and two outlets. The walls are separately specified with respective boundary conditions. No slip condition is considered for each wall. Except the tube walls each wall is set to zero heat flux condition.

4.1.3 Run for Calculation

The number of iteration is set to 500 and the solution is calculated and various contours, vectors and plots are obtained.

For Run-1

Inlet temperature of hot Air = 323 K, Mass Flowrate of hot fluid = 1 lpm, Velocity of Hot Air = 2 m/s



```
Total Temperature                  (k)
-----------------  --------------------
          cold_in           302.99999
         cold_out           317.74772
           hot_in           322.99153
          hot_out           303.30593
```

Fig: 4.5 Temperature Contour of Run 1

$$\varepsilon = \frac{Q_{act}}{Q_{\max}} = \frac{m_h c_{ph}(T_{hi} - T_{ho})}{c_{\min}(T_{hi} - T_{ci})}$$

$$\varepsilon = \frac{4.5265\times10^{-3}(323-303.3)}{4.5265\times10^{-3}(323-303)} = 0.98$$

For remaining seven runs effectiveness is calculated and results.

5. RESULTS

Table 2: Experimental and CFD Effectiveness values

Runs	Expt. (ε)	CFD(ε)
Run 1	0.88	0.98
Run 2	0.55	0.85
Run 3	0.82	0.95
Run 4	0.65	0.83
Run 5	0.7	0.79
Run 6	0.85	0.89
Run 7	0.65	0.77
Run 8	0.7	0.92

Where Expt. = Experimental Values

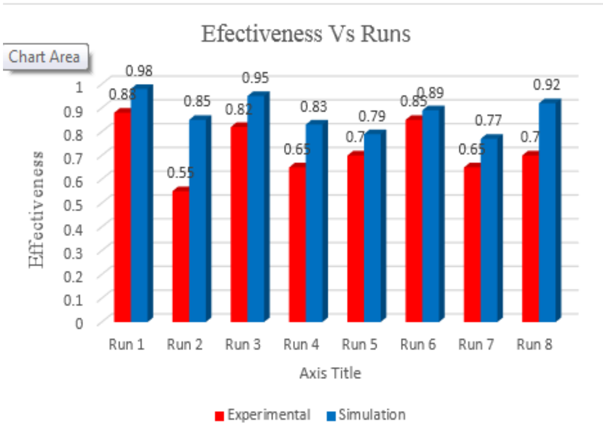


Chart 1: Comparison of Experimental and CFD Effectiveness Values.

From the above graph it is observed that comparison between Experimental values and Simulation values done. The maximum effectiveness for parallel flow tube and tube heat exchanger is observed at run 1 for both experimentation and CFD Simulation. The difference between experimental and simulation values is 10% observed at run 1. Remaining runs are having more than 10% difference between experimental and simulation

6. CONCLUSIONS

Optimization of heat transfer process parameters of tube and tube heat exchanger successfully completed by comparing the experimental and simulation results. To achieve the maximum effectiveness optimized parameters are Velocity of Hot Air = 2 m/s, Hot Air Inlet Temperature = 323 K and Mass Flow rate of water = 1 lpm. The parameters hot air inlet temperature, mass flow rate of water and velocity of air plays a significant role to achieve the maximum effectiveness of Tube and Tube heat exchanger.

REFERENCES

[1] K. Mohan Krishna, D.V.V.S. Bhimesh Reddy "Thermal Analysis of Double Pipe Heat Exchanger by Changing the Materials Using CFD" International Journal of Engineering Trends and Technology (IJETT) – Volume 26 Number 2- August 2015.

[2] KURUVA UMAMAHESH, K. VENUGOPAL [Vol-1, Issue-5, August- 2015] [Vol-1, Issue-5, August- 2015] "Design and Analysis of double pipe heat exchanger using computational method" International Journal of Professional Engineering Studies. Volume V /Issue 2 /JULY 2015.

[3] Bharat Bhushan Verma, Saurabh Kumar "CFD Analysis and Optimization of Heat Transfer in Double Pipe Heat Exchanger with Helical-Tap Inserts at Annulus of Inner Pipe" Volume 13, Issue 3 Ver. VII (May- Jun. 2016).

[4] S. PERUMAL, DR. R. MOHAN "Study on Double Pipe Heat Exchanger Using Different Enhancement Techniques" Imperial Journal of Interdisciplinary Research (IJIR) Vol-2, Issue-2, 2016.

[5] Swathi Juturu, J. Kishore "Comparison of Effectiveness of Two Different Setups of Double Pipe Heat Exchangers" International Journal for Research in Applied Science & Engineering Technology (IJRASET) Volume 3 Issue IX, September 2015.

[6] Gamit Sandip, Gopal Kumar Deshmukh "Numerical and experimental investigation of heat transfer in double pipe heat exchanger with inner and annular twisted tape" International Journal of Engineering Research & Science (IJOER) Vol-1, Issue-5, August- 2015.