CFD Analysis for Performance Enhancement of a Shell and Tube Heat Exchanger using Reduced Width Twisted Tape

[1]Jenifer Abin Valliara, [2]Sandeep M Joshi

[1]Assistant Professor, [2]Associate Professor
Department of Mechanical Engineering,
[1] Don Bosco Institute of Technology, Mumbai, India

***Abstract*—CFD analysis for performance enhancement of the heat exchanger experimental setup using reduced width twisted tapes over full as well as half length twisted tapes inside tubes having twist ratio equal to 1 for laminar flow is carried out in this work. The CFD analysis was done first for plain tube using the same conditions considered for experimental analysis. The results were validated with the experimental results. Then the analysis was done for tube with twisted tape for the same condition as considered for plain tube. The energy balance was carried out for tubes with twisted tape and losses are calculated. It revealed from CFD simulation and analysis of the simulation results that the heat transfer coefficient in counter flow arrangement is enhanced by 10 to 15% using twisted tape insertions. Operating inlet temperature range of hot fluid is being 50-65oC with a mass flow rate of 0.12-0.15kg/sec.**

***Index Terms*—Shell and Tube Heat Exchanger, Overall Heat Transfer Coefficient, Reduced Width Twisted Tape, CFD**

I. INTRODUCTION

In today's scenario there is a lot of concern for the amount of energy losses that takes place during the working of any thermal appliances. Even though these losses cannot be neglected completely, there can be methods to improve the performance keeping into account the various losses present. Methods to improve and enhance the performance of heat exchangers have been the demands for the wide applications of shell and tube heat exchangers. Many correlations and equations for the calculation of heat transfer coefficient, pressure drop, when subjected to different conditions have been reported. The energy balance in a practical application is very often affected by many losses occurring during a heat transfer process. During heat transfer process in a shell and tube heat exchanger, the amount of heat that is lost to the surrounding leads to more energy consumption to maintain the required temperatures. At the same time heat transfer rate is affected during the working of the heat exchanger. This paper focuses on studying effects of twisted tape insertions, using CFD, on thermo-hydraulic performance of the heat exchanger, analyse and compare the performance of heat exchanger with and without twisted tape insertions in the tubes. CFD simulation analysis using aluminum twisted tapes inside tubes whose twist ratio less than 1 is also included in this work

II. LITERATURE REVIEW

3D or 2D axi-symmetric or 2D models could be used for axi-symmetric and planar expansion analysis. 2D axi-symmetric models reduce the number of cells and hence the computational time. Many previous CFD analysis have used axi-symmetric models and found reasonable results. For discretization of the computational domain (for meshing of the domain) structured and unstructured methods are available. Structured methods and quadrilateral cells provide alignment of the flow with mesh which minimizes the numerical diffusion arising from truncation errors that are a consequence of representing the fluid flow equations in discrete form. They also provide better stability and convergence speed, because of good mesh quality in terms of smoothness, skewness, orthogonality index and grid aspect ratio. skewness value of a highly skewed cell in the grid closer to zero is considered better, grid aspect ratio less than 5:1 is better, minimum orthogonal quality index closer to 1 is better.

Types of cells used in the mesh and resolution of the mesh also affect the accuracy, convergence, stability and computing time directly or indirectly. Higher the resolution of the mesh, better the accuracy and lower numerical diffusion but increase in the computational time. Moderately resolved mesh is preferable. Quadrilateral/hexahedral cells offer much larger aspect ratio than triangular/tetrahedral cells without producing higher skewness values that may affect convergence and stability. The alignment of the flow with mesh can never be done by using triangular/tetrahedral cells and hence the high resolution mesh is required implying high computing time.

The equation for conservation of mass or continuity equation must be satisfied at every point in field. The equation applies for a single species fluid, as well as for mixtures in which species diffusion may be occurring.

$$\frac{\partial}{\partial x}(\rho.u) + \frac{\partial}{\partial y}(\rho.v) = \frac{-\partial P}{\partial t} \qquad \text{Eqn 1}$$

with ρ representing the mass density, u for the velocity in mainstream, and v for the normal component.Eq.2.12 represents the momentum equation for the steady state case in which the two terms on the left-hand side represent the net rate of momentum flow, and the terms on the right-hand side account for net viscous and pressure forces, as well as the body force of gravity.

$$\rho_v u \frac{\partial u}{\partial x} + \rho_v v \frac{\partial u}{\partial x} = \rho_v g - \frac{\partial P}{\partial x} + \frac{\partial}{\partial y}\left(\mu_v \frac{\partial u}{\partial y}\right) \qquad \text{Eqn 2}$$

Before deriving the equation of energy conservation, it is necessary to delineate the relevant physical processes. The energy per unit mass of the fluid includes the thermal internal energy and the kinetic energy $v^2/2$:

$$E = h - \left(e + \frac{V^2}{2}\right) \qquad \text{Eqn 3}$$

Where h is the sensible enthalpy for compressible flows defined as:

$$h = \Sigma y_i h_i$$

And for incompressible flows:

$$h = \Sigma y_i h_i + \frac{P}{\rho}$$

With p for the pressure, ρ for the density of the fluid and hi is the enthalpy of the component.

CFD analysis consists of three main steps:-

1 i. Pre-processor: Pre processing includes the following steps:

a. Creating the geometry (domain) based on the design of the system.

b. Meshing-Division of the domain into number of cells (elements).

c. Selection of the phenomena or process that needs to be modeled and defining fluid properties for it.

d. Defining the boundary layer conditions for the domain.

ii. Solver: The solving of the algorithm can be carried out using CFD softwares like ANSYS, FLUENT, CFX and STAR-CD. Following are the sequence in which the algorithm works:

a. Integration of the equations governing the mesh of the domain.

b. Discretization of the resulting integrals into simple algebraic equations.

c. Calculation of the solution of algebraic equation using iterative methods.

iii. Post-processor: Post processing includes the following:

a. Vector plots

b. Contour plots

c. Surface plots

d. Particle tracking

Discretization methods break down any governing differential equations into simple algebraic equations. These equations are then solved iteratively with appropriate softwares.The major challenges in solving these equations are creating a stable approximate equation for the given equation such that the errors during the calculation do not lead to meaningless output. The Finite Element Method is considered to be a better choice specifically when the working domains are complex and the solution lack smoothness.

III. METHODOLOGY

The entire work is divided into following stages:

a. Finding the various losses affecting the performance of a shell and tube heat exchanger.

b. Conducting experiments on the existing setup for different conditions by varying parameters like hot fluid inlet temperature, hot and cold water flow rate.

c. Development of a CFD model of the heat exchanger and carry out the CFD simulation on ANSYS for similar condition as during the experiment. Analysis of simulation results.

d. Carrying out the simulation for the same heat exchanger with a modification of the geometry by introducing reduced width twisted tape inside the tubes of heat exchanger.

e. Comparing the results for both plain and twisted tubes and optimizing the parameters for enhancing the performance of heat exchanger.

IV. CFD ANALYSIS

A 3-D model of the geometry is created in solid works software. A half section cut along the vertical length of the heat exchanger model will be studied for analysis. The actual geometry consists of 55 tubes in the shell. The details of actual geometry is as shown in Table 1. Due to complexity of the actual geometry, the geometry for CFD analysis was modified by reducing the 55 tubes in to one single tube whose cross section area is the sum average of the area of each of the 55 tubes as shown in figure 1 below. For the same geometry CFD analysis was carried out using twisted tape insertion as shown in the figure 2 below.

Table 1 Overall Dimension of Heat Exchanger

No	Description	Unit	Value
1	Shell Diameter	mm	250
2	Tube O.D	mm	19.05
3	Tube I.D	mm	17.02
4	No. Of Tubes	mm	55
5	Tube Length	mm	1470
6	Width of tape	mm	6
7	Thickness of tape	mm	1
8	Twist Ratio	-	1
9	Angle of Twist	degree	19.42

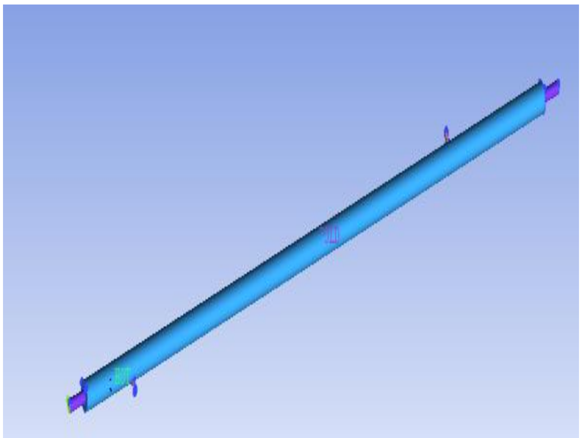

Figure 1 Modified geometry for CFD analysis

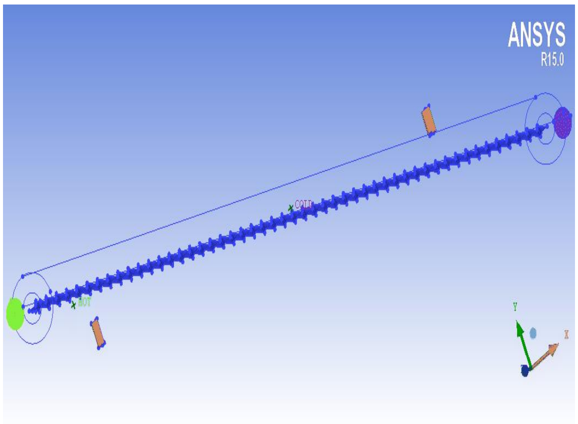

Figure 2 Twisted Tape Insertion in Tube

The meshing was carried out in ICEM CFD 14.5 software which has better access to different types of meshing as per the desired conditions. Initially coarse mesh was generated. The global element scale factor and seed size was selected as 1 and 4 respectively. The surface and the volume mesh type were quadrilateral and tetragonal respectively. Later on, a fine mesh was generated with more number of cells. For this fine mesh, the edges and regions of high temperature and pressure gradients are finely meshed. The number of cells generated was 9 lakhs for heat exchanger with plain tube and 11 lakhs for heat exchanger with twisted tube insertion.

There are many types of discretization schemes. Initially first order upwind scheme was used followed by the second order upwind scheme which is used to have better convergence and to avoid the numerical diffusion. In most of the domain, flow is unidirectional. So, use of second order schemes is beneficial for strong convection. If the convergence criterion is made strict, then the time of solution increases. The advantage of this model is the amount of time saved to converge. Thus a strict criterion is needed to get good results. Hence unscaled residuals are set according to Table 2

Table 2 Residuals

Variable	Residual
x-velocity	10^{-6}
y-velocity	10^{-6}
z-velocity	10^{-6}
Continuity	10^{-6}
Specific dissipation energy	10^{-5}
Turbulent kinetic energy	10^{-5}
Energy	10^{-9}

Standard k - Ɛ model is helpful as it gives a picture of the flow distribution which however is not good in predicting the boundary layer separation. Thus simulation results will deviate from experimental results. The limitations of y+ values at the tube walls results in over-prediction of pressure drop and heat transfer by almost 25%

CFD analysis was carried out for two different conditions. First condition was for heat exchanger with plain tubes inside the shell and second condition was for heat exchanger with twisted tape insertion inside the tube. For each condition, the analysis was carried out for both parallel and counter flow pattern. This analysis was done only for hot fluid mass flow rate = 0.0386 kg/s and cold fluid mass flow rate = 0.1536 kg/s. For this given mass flow rate the only parameter that was varied was hot fluid inlet temperature.

V. RESULTS AND DISCUSSIONS

The temperature of hot fluid was limited to 80 $^{\circ}$C since above this temperature it was unable to attain a steady condition. It was observed that effectiveness is better in counter flow as compared to parallel flow.

Table 1 Performance results from CFD analysis using plain tube for parallel flow arrangement

Hot fluid Inlet temp ($^{\circ}$C)	Cold fluid outlet temp ($^{\circ}$C)	Heat transfer coefficient tube side (W/mK)	Heat transfer coefficient shell side (W/mK)	Overall Heat transfer coefficient (W/mK)
t_{hi}	t_{co}	h_i	h_o	U
65	46	171237.5	15.97516	15.96211
60	36	156502.9	15.63092	15.61825
55	35	156400.5	15.27328	15.26118
50	35	156400.5	15.52174	15.50925

As the hot fluid inlet temperature increases, effectiveness increases provided the mass flow rate of the hot fluid is not high. For a given hot fluid mass flow rate, effectiveness will increase with decrease in cold fluid mass flow rate. 40% decrease in the cold fluid mass flow rate increases the effectiveness by 30%.

Table 2 Performance results from CFD analysis using plain tube for Counter Flow Arrangement

Hot fluid Inlet temp ($^{\circ}$C)	Cold fluid outlet temp ($^{\circ}$C)	Heat transfer coefficient tube side (W/mK)	Heat transfer coefficient shell side (W/mK)	Overall Heat transfer coefficient (W/mK)
t_{hi}	t_{co}	h_i	h_o	U

65	50.7	175421.9	16.16767	16.15435
60	47.3	173460	15.73136	15.71872
55	43.3	170432.8	15.42125	15.40908
50	40	170432.8	15.08298	15.07134

Table 3 Performance results from CFD analysis using twisted tube for parallel flow arrangement				
Hot fluid inlet temp(˚C)	Cold fluid outlet temp (˚C)	Heat transfer coefficient tube side (W/mK)	Heat transfer coefficient shell side (W/mK)	Overall Heat transfer coefficient (W/mK)
t_{hi}	t_{co}	h_i	h_o	U
65	52	194413.4	16.8713	16.8570
60	47	171237.5	16.3730	16.3592
55	46.5	174640.8	15.8972	15.8842
50	41	167607.4	15.6291	15.6165

Table 4 Performance results from CFD analysis using twisted tube for Counter Flow Arrangement				
Hot fluid inlet temp (˚C)	Cold fluid outlet temp (˚C)	Heat transfer coefficient tube side (W/mK)	Heat transfer coefficient shell side (W/mK)	Overall Heat transfer coefficient (W/mK)
t_{hi}	t_{co}	h_i	h_o	U
65	55.7	175421.9	17.6636	17.6477
60	52	220753.5	16.79606	16.7821
55	47	181211.4	16.83075	16.81638
50	42.1	164892.7	15.66788	15.65526

On comparing the heat transfer rate for twisted and plain tube, it was found that both for parallel and counter flow, heat transfer coefficient with tubes having twisted tape insertion is greater than plain tubes.

ACKNOWLEDGMENT

We are grateful to the management of Don Bosco Institute of Technology and Pillai College of Engineering for their valuable time and support to make this work a success .

REFERENCES

[1] Frank P. Incropera, David P. Dewitt, John Willy & Sons, Fourth Edition, textbook of Fundamentals of Heat and Mass Transfer

[2] Yunus A. Cengel, John M. Cimbala, Mc Graw Hill, Second Edition, a textbook of Fluid Mechanics

[3] Jeevraj S., Design and simulation of heat sink for high power electronics using CFD, International Journal of Scientific research and management, Volume 1, Issue 3, pages 145-149, 2013

[4] John D. Anderson, JR, McGraw-Hill International Edition, a textbook of Computational Fluid Dynamics

[5] B.Adrian and K. Allan D. Heat transfer enhancement. In Heat Transfer Handbook, Chapter 14, pg.1033, -1101, Wiley-interscience, 2003.

[6] Bergles, A.E. ―Techniques to augment heat transfer.‖ In Handbook of Heat Transfer Applications (Ed.W.M. Rosenhow), 1985, Ch.3 (McGraw-Hill, New York).
[7] Date, A.W. Prediction of fully developed flow in a tube containing a twisted tape, International Journal Heat Mass Transfer, 17, pp. 845-859, 1974
[8] Date A.W,.Flow in tube containing twisted tape Heat and vent Engr, Journal of Environmental Science,47, pp240-249,1973
[9] S.D.Patil et al. ―Analysis of Twisted Tape with Winglets to Improve the Thermo hydraulic Performance of Tube in Tube Heat Exchanger‖, International Journal of Advanced Engineering Research and Studies E-ISSN2249 – 8974
[10] S. Naga Sarada et. al. ―Enhancement of heat transfer using varying width twisted tape inserts, International Journal of Engineering, Science and Technology Vol. 2, No. 6, pp. 107-118, 2010
[11] Usman Ur Rehman, Heat Transfer Optimization of Shell-And-Tube Heat Exchanger through CFD Studies, Chalmers University of Technology, 2011.