CFD Analysis of Double Pipe Heat Exchanger

Jibin Johnson, Abdul Anzar V M, Abith Shani, Harif Rahiman P, Hashmi Hameed T S, Nithin V S
Department of Mechanical Engineering, IES College of Engineering. Kerala ,India

Abstract— **Heat exchangers are used in industrial processes to recover heat between two process fluids. All the heat exchangers are designed based on the function it fulfills in a process. Although the necessary equations for heat transfer and the pressure drop in a double pipe heat exchanger are available, using these equations the validation of the design is laborious. In this paper the analytical design of the exchanger has been validated based on the results obtained from the CFD analysis. In this paper the CFD analysis is based on the standard k-ε modeling. The solution of the problem yields the optimum values of inner pipe diameter, outer pipe diameter and utility flow rate to be used for a double pipe heat exchanger of a given effective length, when a specified flow rate of process stream is to be treated for a given inlet to outlet temperature.**

Index Terms—**Design, Heat exchanger, Analysis, Pressure drop, Heat transfer, Double pipe, CFD**

I. INTRODUCTION

Heat transfer equipment is defined by the function it fulfills in a process. The objective of any such equipment is to maximize the heat transferred between the two fluids. However, the problem that occurs is that the parameters which increase the heat transfer also increase the pressure drop of the fluid flowing in a pipe. Therefore, a design which increases the heat transferred, but simultaneously could keep the pressure drop of the fluid flowing in the pipes to permissible limits, is very necessary. A common problem in industries is to extract maximum heat from a utility stream coming out of a particular process, and to heat a process stream. This paper is based on a plant study conducted in a chemical processing company of caustic soda (TCCL, Kerala). A solution to extract the maximum heat could have been to increase the heat transfer area or to increase the coolant flow rate. But increasing these parameters without pressure drop considerations is not advisable. Traditional design method of heat exchangers involves the consideration of all the design variables with a laborious procedure of trial and error, taking all possible variations into consideration. This time consuming procedure can be reduced somewhat by making some reasonable assumptions. Also these procedures are not convenient, will have some errors.

In order to validate those designs prepared by trial and error procedures some analyzing methods are needed. Among these the analysis using softwares are more profitable than experimental analysis. We have considered the design of a double pipe heat exchanger by considering three main parameters – the inner and outer diameter of the heat exchanger and the flow rate of the utility. It is assumed that the flow rate, the inlet and the required outlet temperature of the process fluid and the inlet temperature of the utility are known for the specific design of the exchanger.

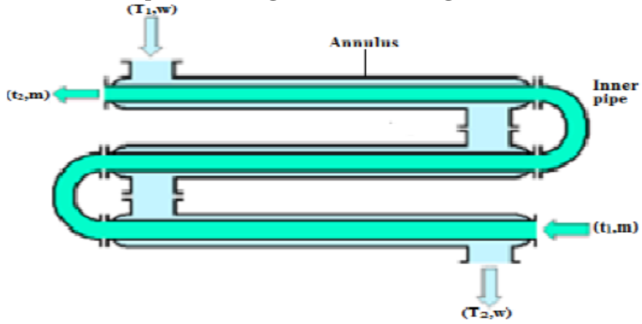

Fig.1. Three double pipe heat exchangers in series

II. ANALYTICAL THEORY

Computational fluid dynamics (CFD) is the science of predicting fluid flow, heat and mass transfer, chemical reactions, and related phenomena by solving numerically the set of governing mathematical equations, such as conservation of mass, conservation of momentum, conservation of energy, conservation of species, effects of body forces, etc. The results of CFD analyses are relevant in, conceptual studies of new designs, detailed product development, troubleshooting, redesign. CFD analysis complements testing and experimentation by reducing total effort and cost required for experimentation and data acquisition.

ANSYS CFD solvers are based on the finite volume method. In this method domain is discretized into a finite set of control volumes. After those general conservation (transport) equations for mass, momentum, energy, species, etc. are solved on this set of control volumes. [5]

$$\underbrace{\frac{\partial}{\partial t}\int_V \rho\phi dV}_{Unsteady} + \underbrace{\oint_A \rho\phi \vec{V}.\vec{dA}}_{Convection} = \underbrace{\oint_A \Gamma_\phi \nabla\phi.\vec{dA}}_{Diffusion} + \underbrace{\int_V S_\phi dV}_{Generation} \quad (1)$$

These Partial differential equations are discretized into a system of algebraic equations. All those algebraic equations are then solved numerically to render the solution field. Where ρ, is the density, Φ, is the temperature difference, V, is the volume, $\vec{V}$, is the velocity vector and A, is the area.

ISSN: 2278 – 7798

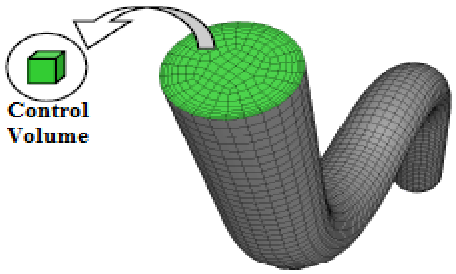

Fig.2. Fluid region in a pipe flow, discretized into a finite set of control volumes

There are mainly nine steps in the CFD analysis under four stages. The first stage is problem identification, which includes defining the analysis goals and identifying the domain that will be analyzed. And in the second stage, pre-processing includes creating a solid geometrical model, designing and generating the mesh, setting up the physics and defining the solver settings. The solution will be computed in the solving stage. Finally in the post processing stage the results will be examined and the model will be updated if revisions are needed.

In the CFD there are several types of models related to different situations. Among these in this paper we adopted the RANS-based turbulence models, specifically standard k-ε model for the design example mentioned. Where k, is the turbulent kinetic energy and ε, is the dissipation rate. The dissipation rate is related to turbulent kinetic energy by,

$$\varepsilon \approx \frac{k^{\frac{3}{2}}}{L_t} \qquad (2)$$

where L_t, is the turbulence length. [5]

III. DESIGN EXAMPLE

This paper is based on the plant study conducted in the Caustic Concentration and Flanking (CCF) plant of TCC Ltd, Kerala, India. Here a plate heat exchanger is under gasket failure as per the operating conditions given below,

TABLE 1
OPERATING CONDITIONS

Operating conditions	Shell side (NaOH)	Tube side (steam)
Mass flow rate (kg/hr)	5501	3722.8
Temperature (in-out) (^{o}C)	66.7-140	170.4-80.1
Pressure (bar)	5	7

The fluid properties of NaOH, specific heat, C_p=3.5 kJ/kgK, viscosity, μ=3.1 mPa.s, thermal conductivity, k=0.75 W/m^{2}K, density, ρ=1355kg/m^3 and the fluid properties of steam, specific heat, C_p=4.2 kJ/kgK, viscosity, μ=0.16 mPa.s, thermal conductivity, k=0.68 W/m^{2}K, density, ρ=935 kg/m^3. We have to design a double pipe heat exchanger as per the same operating conditions of plate heat exchanger and the above mentioned fluid properties. The aim of design is to heat the NaOH using the steam. [2], [3]

Using the above design data the double pipe heat exchanger is designed as per the design procedure mentioned by [1] and [4]. Before starting the design we have to assume the inner pipe and annulus diameters as per the pipe standards. In this design example the nominal pipe sizes and diameters are given below,

TABLE 2
NOMINAL PIPE SIZES

Pipe	Inner diameter (m)	Annulus (m)
Inner pipe, 2”	0.0525	0.0779
Annulus, 3”	0.0603	0.0889

With help all the data mentioned above the designing of the double pipe heat exchanger was completed. And finally the number of pipes and the number of hair pins are obtained. In this example, number of pipes = 8 and number of hairpins = 7.

IV. ANALYTICAL DATA

After the design stage has completed we need to analyze our design for validating our results. In the case of double pipe heat exchanger the flow of fluids through the pipe have to be analyzed. For this Computational Fluid Dynamics (CFD) analysis is necessary. ANSYS is the one of the important software to conduct the CFD analysis. The CFD analysis mainly based on the geometry modeled as per the design results. In the case of above design example the double pipe heat exchanger was modeled in the solid works (Fig.3).

Fig.3. Model of double pipe heat exchanger

The analysis was done in ANSYS fluent. The model prepared in the solid works was simplified in to symmetric fluid sections in the design modeler for the easy computation of analysis.

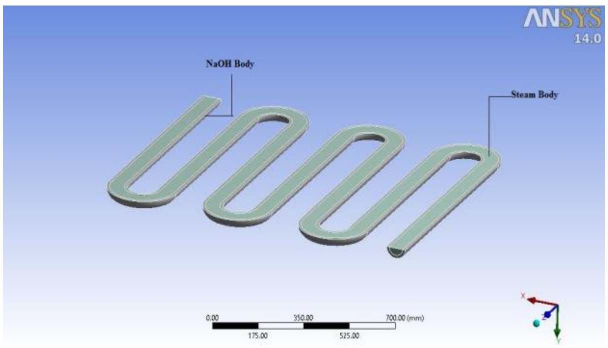

Fig.4. Simplified model of fluid bodies

After the model had simplified, the model was meshed (Fig.5) to discretize the entire structure into finite volumes. In this analysis the boundary conditions are applied at nodes of each finite volume.

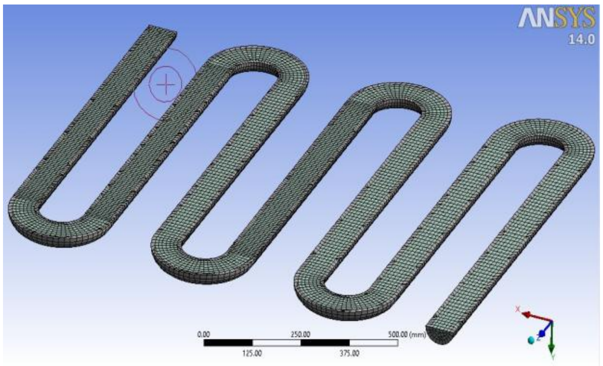

Fig.5. Meshed structure

After meshing is completed the physical setup for calculations including boundary conditions and solver settings were entered. After all the values had entered the CFD computation will be started. The iterations are continued till the convergence of results had obtained.

V. RESULT VALIDATION

The aim of this analysis is to validate the design of double pipe heat exchanger mentioned in the design example. The results obtained from the analysis are given below,

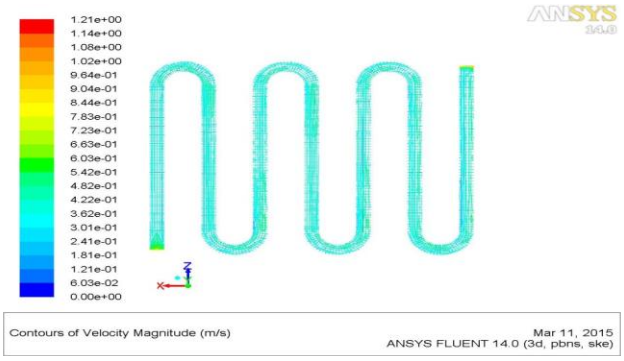

Fig.6. Velocity contour

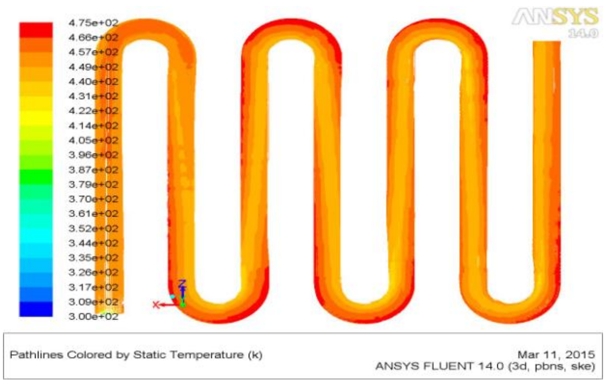

Fig.7. Temperature pathlines by static temperature

From the that we had obtained we can analyze that the design of double pipe heat exchanger had reached a great sucesss. By observing the velocity contours we can infer that the velocity distribution inside the pipes are in the lower range of colour code. That means the velocity ditribution is minimum to enhance the heat transfer rate. Also the temperature pathlines shows that the temperature distribution is in the maximum scale range, which shows the perfect heat transfer rate. From these results we can compare the theoreticaly calculated heat transfer rate with the results obtained from analysis.

TABLE 3
RESULT VALIDATION

Parameter	Theoretical		Analytical	
	Inner pipe	Annulus	Inner pipe	Annulus
Temperature (^{o}C) (in-out)	170.4 - 80.1	66.7 - 140	177 - 112	66.7 - 194
Heat transfer rate (J/hr)	1411.28		2450.97	

VI. CONCLUSION

- ❖ The design and analysis of the double pipe heat exchanger has been a great success.
- ❖ The design stages mentioned in the literature can applicable for all the simple designing of double pipe heat exchangers.
- ❖ The analytical stage in this paper shows the validity of the design procedure mentioned above.
- ❖ Also we can use this type design for solving the problems in industrial processes as mentioned in the design example.
- ❖ Also the validation will show the existence of the design.

ACKNOWLEDGEMENT

The authors would like to thank Mr. E.Sreemohan Chief Engineer, Mechanical Maintenance Department, Travancore Cochin Chemicals Ltd. Ernakulam, Kerala, India for their technical support and experimental assistance throughout the work.

REFERENCES

[1] Donald.Q. Kern, Process Heat Transfer, McGraw Hill Book Company Inc., New York, NY, 1965.
[2] Robert H. Perry, Don W. Green, Chemical Engineer's hand book, McGraw Hill Book Company Inc., New York, NY, 1977.
[3] C.P. Kothandaraman,S. Subramanyam, Heat and Mass Transfer databook, New Age International (P) Ltd., New Delhi, seventh edition, 2010.
[4] Prabhata .K Swamee, Nitin Aggarwal, Vijay Aggarwal, Optimum Design of Double Pipe Heat Exchanger, Int.J. Heat and mass transfer, 51 (2008) 2260–2266.
[5] ANSYS training manual, Fluent 12, first edition, ANSYS. Inc. Proprietary, Canonsburg, 2009.

NOMENCLATURE

T_1 & T_2 - Inlet and outlet temperatures of annulus
T_1 & t_2 - Inlet and outlet temperatures of inner pipe
w - Mass flow rate of annulus
m - Mass flow rate of inner pipe
Pbns - Pressure based navier-stokes
Ske - Standard k-ϵ
RANS - Reynolds Average Navier-Stokes equaiton

ISSN: 2278 – 7798

AUTHORS

First Author -Jibin Johnson, doing graduate in Mechanical Engineering from Calicut University, Student in IESCE, Chittilappilly, Thrissur, Kerala, India.

Second Author -Abdul Anzar V M, doing graduate in Mechanical Engineering from Calicut University, Student in IESCE, Chittilappilly, Thrissur, Kerala, India.

Third Author -Abith Shani, doing graduate in Mechanical Engineering from Calicut University, Student in IESCE, Chittilappilly, Thrissur, Kerala, India.

Fourth Author -Harif Rahiman P, doing graduate in Mechanical Engineering from Calicut University, Student in IESCE, Chittilappilly, Thrissur, Kerala, India.

Fifth Author -Hashmi Hameed T S, doing graduate in Mechanical Engineering from Calicut University, Student in IESCE, Chittilappilly, Thrissur, Kerala, India.

Sixth Author -Nithin V.S, Assistant Professor, Department of Mechanical Engineering, IESCE, Chittilappilly, Thrissur, Kerala, India.