International Journal of Advanced Technology in Engineering and Science
Vol. No.4, Issue No. 04, April 2016
www.ijates.com
ijates
ISSN 2348 - 7550

HEAT TRANSFER OPTIMIZATION OF SHELL AND TUBE HEAT EXCHANGER THROUGH CFD ANALYSIS

Prof. Abhay Bendekar[1], Prof. V. B. Sawant[2]

[1]Asst. Professor, Mechanical Engineering ,

Shree L. R.Tiwari College of Engineerin , Thane (E), (India)

[2]Asst. Professor, Mechanical Engineering ,

Rajiv Gandhi Institute of Technology, Andheri (W), Mumbai, (India))

ABSTRACT

An un-baffled shell-and-tube heat exchanger design with respect to heat transfer coefficient and pressure drop is investigated by numerically modeling. The heat exchanger contained 19 tubes inside a 5.85m long and 108mm diameter shell. The flow and temperature fields inside the shell and tubes are resolved using a commercial CFD package considering the plane symmetry. A set of CFD simulations is performed for a single shell and tube bundle and is compared with the experimental results. The results are found to be sensitive to turbulence model and wall treatment method. It is found that there are regions of low Reynolds number in the core of heat exchanger shell. Thus, $k-\omega$ SST model, with low Reynolds correction, provides better results as compared to other models. The temperature and velocity profiles are examined in detail. It is seen that the flow remains parallel to the tubes thus limiting the heat transfer. Approximately, 2/3rd of the shell side fluid is bypassing the tubes and contributing little to the overall heat transfer. Significant fraction of total shell side pressure drop is found at inlet and outlet regions. Due to the parallel flow and low mass flux in the core of heat exchanger, the tubes are not uniformly heated. Outer tubes fluid tends to leave at a higher temperature compared to inner tubes fluid. Higher heat flux is observed at shell's inlet due to two reasons. Firstly due to the cross-flow and secondly due to higher temperature difference between tubes and shell side fluid. On the basis of these findings, current design needs modifications to improve heat transfer.

Keywords-Heat transfer, Shell-and-Tube Heat exchanger, CFD, Un-baffled.

I. INTRODUCTION

Heat exchangers are one of the mostly used equipment's in the process industries. Heat exchangers are used to transfer heat between two process streams. One can realize their usage that any process which involves cooling, heating, condensation, boiling or evaporation will require a heat exchanger for these purposes. Process fluids, usually areheated or cooled before the process or undergo a phase change. Different heat exchangers are named according to their applications. For example, heat exchangers being used to condense are known as condensers; similarly heat ex- changers for boiling purposes are called boilers. Performance and efficiency of heat exchangers are measured through the amount of heat transferred using least area of heat transfer and pressure drop. A better presentation of its efficiency is done by calculating over all heat transfer coefficient. Pressure drop and area required for a certain amount of heat transfer, provides an in- sight about the capital cost and power requirements (Running cost) of a heat exchanger. Usually, there is lots of literature and theories to design a heat exchanger according to the requirements. A good design is referred to a heat exchanger with least possible area and pressure drop to fulfill the heat transfer requirements [1].

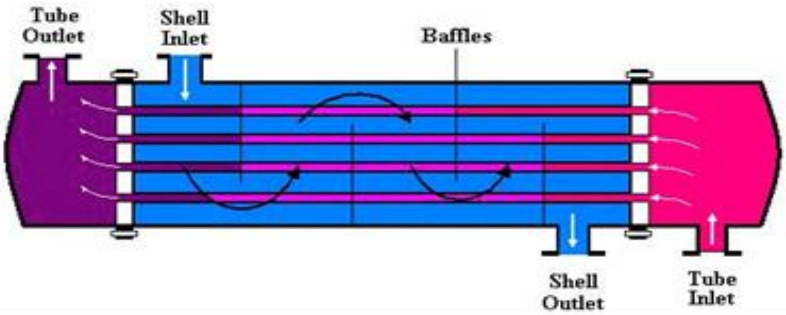


Fig. 1 Counter-current Heat Exchanger Arrangement

A Heat Exchanger Classification

At present heat exchangers are available in many configurations. Depending upon their application, process fluids, and mode of heat transfer and flow, heat exchangers can be classified [2].

Heat exchangers can transfer heat through direct contact with the fluid or through indirect ways. They can also be classified on the basis of shell and tube passes, types of baffles, arrange- ment of tubes (Triangular, square etc.) and smooth or baffled surfaces. These are also classified through flow arrangements as fluids can be flowing in same direction (Parallel), opposite to each other (Counter flow) and normal to each other (Cross flow). The selection of a particular heat exchanger configuration depends on several factors. These factors may include the area requirements, maintenance, flow rates, and fluid phase.

B Applications of Heat exchangers

Applications of heat exchangers are a very vast topic and would require a separate thorough study to cover each aspect. Among the common applications are their use in process industry, mechanical equipment’s industry and home appliances. Heat exchangers can be found employed for heating district systems, largely being used now days. Air conditioners and refrigerators also install the heat exchangers to condense or evaporate the fluid. Moreover, these are also being used in milk processing units for the sake of pasteurization. The more detailed applications of the heat exchangers can be found in the Table 1.1 w.r.t different industries[3].

C Literature Survey

Shell and tube heat exchanger design is normally based on correlations, among these; the Kern method [4] and Bell-Delaware method [5] are the most commonly used correlations. Kern method is mostly used for the preliminary design and provides conservative results. Whereas, the Bell-Delaware method is more accurate method and can provide detailed results. It can predict and estimate pressure drop and heat transfer coefficient with better accuracy. The Bell-Delaware method is actually the rating method and it can suggest the weaknesses in the shell side deign but it cannot indicate where these weaknesses are. Thus in order to figure out these problems, flow distribution must be understood. For this reason, several analytical, experimental and numerical studies have been carried out. Most of this research was concentrated on the certain aspects of the shell and tube heat exchanger design [6]. These correlations are developed for baffled shell and tube heat exchangers generally.

Our study aims at studying simple un-baffled heat exchanger, which is more similar to the double pipe heat exchangers. Almost no study is found for an un-baffled shell and tube heat exchanger. Thus general correlations of heat transfer and pressure drop for straight pipes can be useful to get an idea of the design. Generally there has been lot of work done on heat transfer [7] and pressure drop [8] in heat exchangers. Pressure drop in a heat exchanger can be divided in three parts. Mainly it occurs due to fanning friction along the pipe. In addition to this it also occurs due to geometrical changes in the flow i.e. contraction and expansion at inlet and outlet of heat exchanger [9]. Handbook of hydraulic resistance pro- vides the correlations for the pressure losses in these three regions separately by introducing the pressure loss coefficients.

Compared to correlation based methods, the use of CFD in heat exchanger design is limited. CFD can be used both in the rating, and iteratively in the sizing of heat exchangers. It can be particularly useful in the initial design steps, reducing the number of tested prototypes and providing a good insight in the transport phenomena occurring in the heat exchangers [11]. To be able to run a successful full CFD simulation for a detailed heat exchanger model, large amounts of computing power and computer memory as well as long computation times are required. Without any simplification, an industrial shell and tube heat exchanger with 500 tubes and 10 baffles would require at least 150 million computational elements, to resolve the geometry [12]. It is not possible to model such geometry by using an ordinary computer. To overcome that difficulty, in the previous works, large scale shell-and-tube heat exchangers are modeled by using some simplifications. The commonly used simplifications are the porous medium model and the distributed resistance approach. Shell-and-tube heat exchangers can be modeled using distributed resistance approach [12]. By using this method, a single computational cell may have multiple tubes; therefore, shell side of the heat exchanger can be modeled by relatively coarse grid. Kao et al [13] developed a multidimensional, thermal-hydraulic model in which shell side was modeled using volumetric porosity, surface permeability and distributed resistance methods. In all of these simplified approaches, the shell side pressure drop and heat transfer rate results showed good agreement with experimental data.

With the simplified approaches, one can predict the shell side heat transfer coefficient and pressure drop successfully, however for visualization of the shell side flow and temperature fields in detail, a full CFD model of the shell side is needed. With ever increasing computational capabilities, the number of cells that can be used in a CFD model is increasing. Now it is possible to model an industrial scale shell- and-tube heat exchanger in detail with the available computers and software's. By modeling the geometry as accurately as possible, the flow structure and the temperature distribution inside the shell can be obtained. This detailed data can be used for calculating global parameters such as heat transfer coefficient and pressure drop that can be compared with the correlation based or experimental ones[6]. Moreover, the data can also be used for visualizing the flow and temperature fields which can help to locate the weaknesses in the design such as recirculation and relaminarization zones.

According to a recent review [14], commercial and non-commercialsoftware's are used to model different types of heat exchangers. Normally, for modeling the flow, two equation models are the most commonly used models. $k - \varepsilon$ models are mostly used in industrial designs along with wall functions. Jae et al [15] compared the different near wall treatment methods for high Reynolds number flows. It was found that non-equilibrium wall

ijates
ISSN 2348 - 7550

functions along with k − ε models predicts the reattachment lengths more accurately, but two layer model represents the overall flow domain much better. The use of these near wall treatments is very much dependent upon the choice of turbulence model used.

II. TURBULENCE MODELING

Turbulent flows contain a wide range of length, velocity and time scales and solving all of them makes the costs of simulations large. Therefore, several turbulence models have been developed with different degrees of resolution. All turbulence models have made approximations simplifying the Navier-Stokes equations. There are several turbulence models available in CFD software's including the Large Eddy Simulation (LES) and Reynolds Average Navier- Stokes (RANS). There are several RANS models available depending on the characteristic of flow, e.g., Standard k − ε model, k − ε RNG model, Realizable k − ε, k − ω and RSM (Reynolds Stress Model) models.

III. CFD ANALYSIS

Computational fluid dynamic study of the system starts with building desired geometry and mesh for modeling the domain. Generally, geometry is simplified for the CFD studies. Meshing is the discretization of the domain into small volumes where the equations are solved by the help of iterative methods. Modeling starts with defining the boundary and initial conditions for the domain and leads to modeling the entire system domain. Finally, it is followed by the analysis of the results.

A Geometry

Heat exchanger geometry is built in the ANSYS workbench design module. Geometry is simplified by considering the plane symmetry and is cut half vertically. It is a counter current heat exchanger, and the tube side is built with 11 separate inlets comprising of 8 complete tubes and3 half tubes considering the symmetry. The shell outlet length is also increased to facilitate the modeling program to avoid the reverse flow condition. In the Figure 2 (a) and 2(b)), the original geometry along with the simplified geometry can be seen.

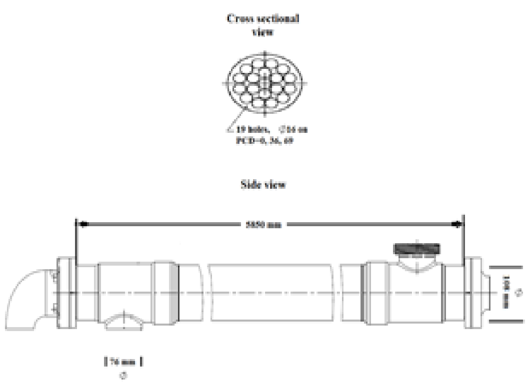


Fig. 2 (a) Original Geometry

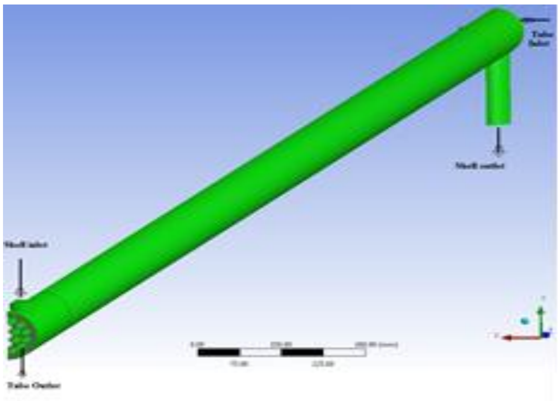


Fig. 2 (b) Simplified Geometry

B Mesh

Initially a relatively coarser mesh is generated with 1.8 Million cells. This mesh contains mixed cells (Tetra and Hexahedral cells) having both triangular and quadrilateral faces at the boundaries. Care is taken to use structured cells (Hexahedral) as much as possible, for this reason the geometry is divided into several parts for using automatic methods available in the ANSYS meshing client. It is meant to reduce numerical diffusion as much as possible by structuring the mesh in a well manner, particularly near the wall region. Later on, for the mesh independent model, a fine mesh is generated with 5.65 Million cells. For this fine mesh, the edges and regions of high temperature and pressure gradients are finely meshed.

C Boundary Conditions

TABLE 1 Heat Exchanger Dimensions

	BC Type	*Shell*	*Tube*
Inlet	*Velocity-inlet*	*1.2 m/s*	*1.8 m/s*
Outlet	*Pressure-outlet*	*0*	*0*
Wall	*No slip condition*	*No heat flux*	*Coupled*
Turbulence	*Turbulence Intensity*	*3.6%*	*4%*
	Length Scale	*0.005*	*0.001*
Temperature	*Inlet temperature*	*317K*	*298K*
Mass flow rate		*20000kg/hr*	*20000kg/hr*

No.	Description	Unit	Value
1	Overall dimensions	mm	54x378x5850
2	Shell diameter	mm	108
3	Tube outer diameter	mm	16
4	Tube inner diameter	mm	14.6
5	Number of tubes		19
6	Shell/ Tube length	mm	5850
7	Inlet length	mm	70
8	Outlet length	mm	200

IV. RESULTS AND DISCUSSION

A Model Comparison

Different turbulence models are evaluated to investigate their application for our case. Each turbulence model along with different wall treatment methods is used with medium mesh (2.2 million cells). A comparison of overall heat transfer coefficient and pressure drop obtained from these models can be seen in the Figures 3 and 4 respectively. Knowing the temperatures from CFD results, Overall heat transfer coefficient is calculated from equations. Due to the available experimental data for comparison, only overall heat transfer coefficient is

ijates
ISSN 2348 - 7550

calculated. Whereas, pressure drop can easily be calculated from CFD and thus, is compared with available experimental data.

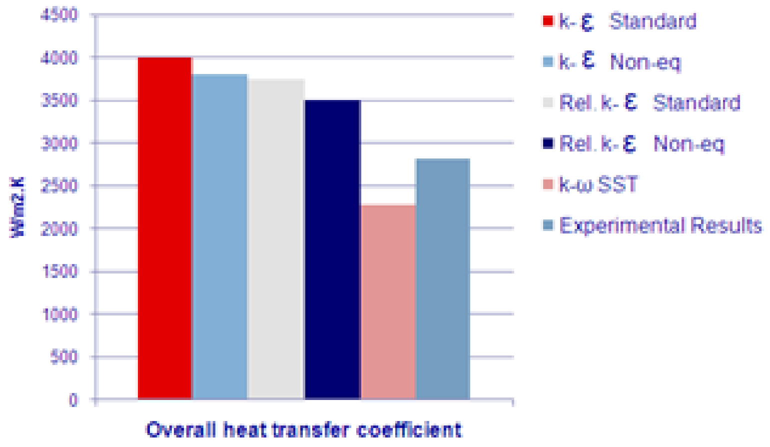


Fig. 3 Overall Heat Transfer Coefficient

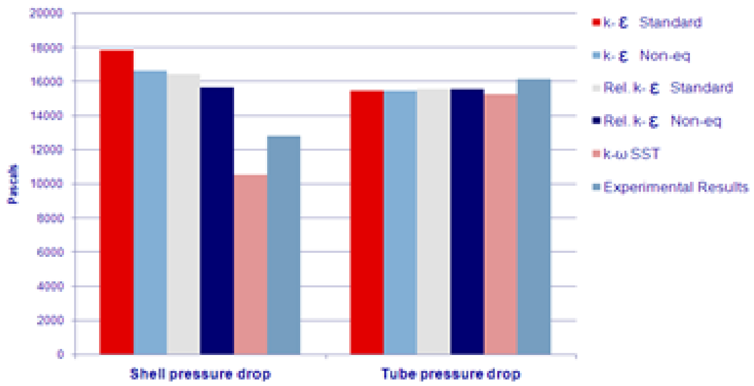


Fig. 4 Pressure Drop

TABLE 2 CFD and Experimental Results

		CFD results			Experimental results			% Difference		
Reynolds number × 10^4		Pressure drop (kPa)		HT coefficient (W/m².K)	Pressure drop (kPa)		HT coefficient (W/m².K)	Pressure drop (kPa)		HT Coefficient(W/m².K)
Shell	Tube	Shell	Tube	Overall	Shell	Tube	Overall	Shell	Tube	Overall
9.2	2.16	4.04	6.01	1544.02	5.6	6.6	1965	27.91	9.0	21.4
10.64	2.52	5.32	7.99	1711.49	7.2	8.7	2196	26.16	8.2	22.1
12.16	2.88	6.8	10.3	1912.16	8.9	10.9	2414	23.63	5.7	20.8
13.68	3.24	8.41	12.7	2097.27	10.8	13.4	2621	22.14	5.3	20.0
15.2	3.6	10.5	15.2	2278.68	12.8	16.1	2819	19.69	5.7	19.2

B CFD Comparison with Experimental Results

On the basis of findings in previous Chapter, SST k − ω model with low Re modification is used with different mass flow rates to compare with experimental results. The results are given in the TABLE 1.

The pressure drop in shell and tube side is shown in Figures 4 and 5 respectively.The pressure drop in the shell is under-predicted by the SST k − ω model by almost 20-27%. Thiscould be due to the several reasons including complicated geometry of the shell side and numerical diffusion. Where as, the pressure drop in tube side

International Journal of Advanced Technology in Engineering and Science
Vol. No.4, Issue No. 04, April 2016
www.ijates.com
ijates
ISSN 2348 - 7550
(straight tubes) is predicted with an average error between 5-9%. It can be due to small baffles in the tubes used in the experimental setup.

Overall heat transfer coefficient comparison with experiments can also be seen in the Figure6. It is also been under-predicted by this model but still better than other models with an average error of 19-20%. The good thing about these results is the constant difference from experimental results and consistency with the real systems, i.e. with higher pressure drop, higher heat transfer is achieved.

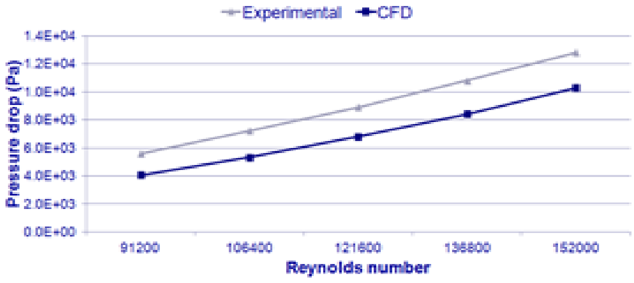


Fig. 4 Comparison of Shell Side Pressure Drop

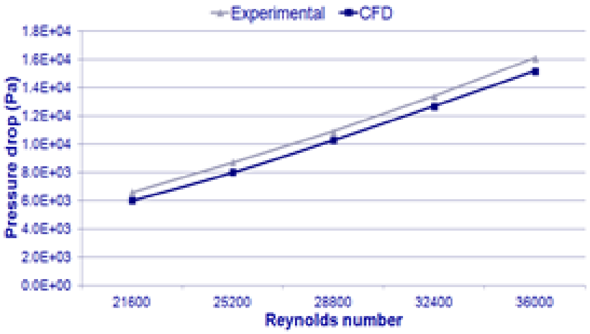


Fig. 5 Comparison of Tube Side Pressure Drop

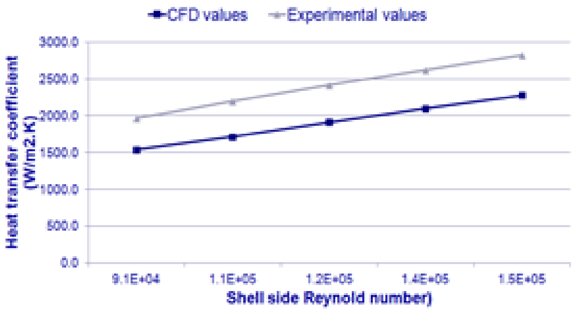


Fig. 6 Comparison of Overall Heat Transfer Coefficient

C Contour Plots

The temperature and velocity distribution along the heat exchanger can be seen through side view on the plane of symmetry. The contour plots in Figure 7 and 8 show the whole length of heatexchanger. The whole length is too much to be displayed on a single page with understandable resolution, thus it is cut into 4 parts to see it closely. The top most part is the inlet region and lowest part is the outlet.

As the heat exchanger is almost 6 meters long, the velocity and temperature contour plots across the cross section at different position along the length of heat exchanger will give an idea of the flow in detail. For convenience the plots are taken at 5 different positions and the details of the temperature distribution in comparison to the velocity distribution can be observed in the Table 4.1.

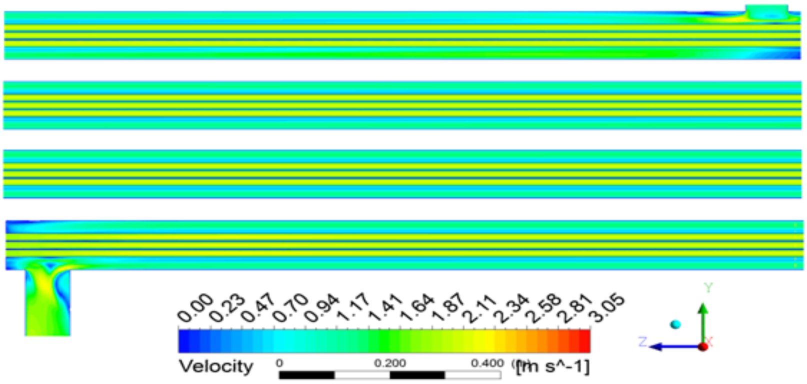


Fig. 7 Velocity Contour Plot at Symmetrical Plane

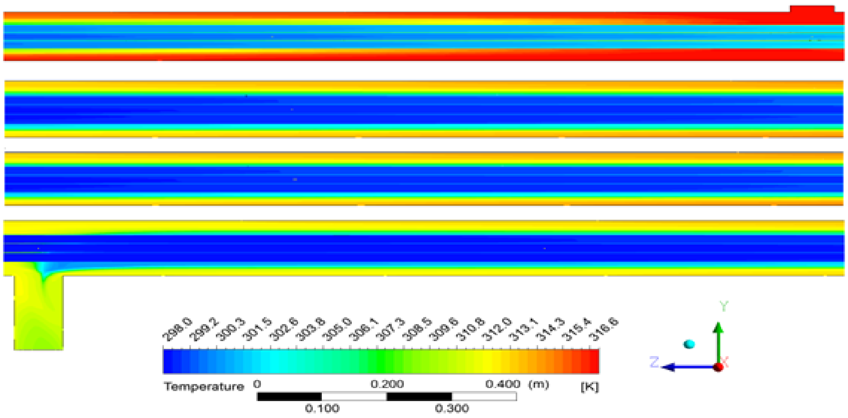
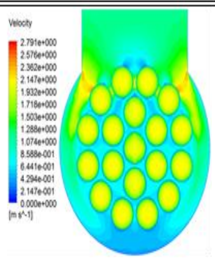
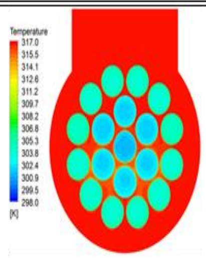
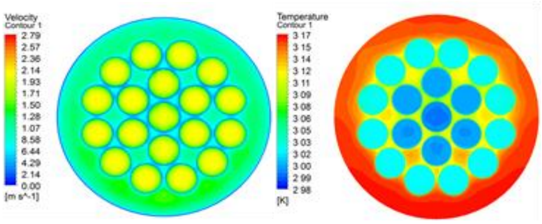


Fig. 8 Temperature Contour Plot at Symmetrical Plane

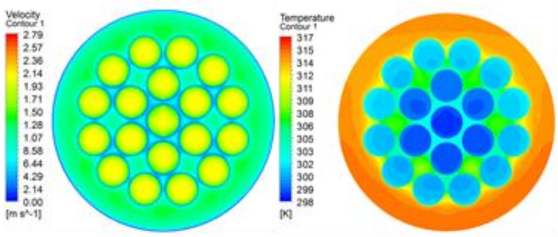
TABLE3 Velocity and Temperature Contour Plots

Velocity	Temperature
Inlet	
	
1 M	

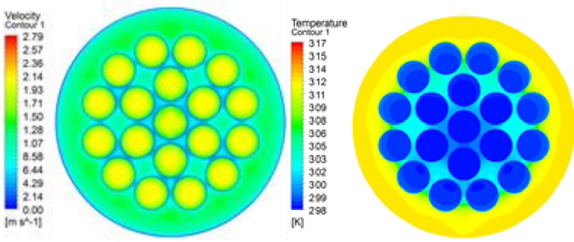
ijates
ISSN 2348 - 7550



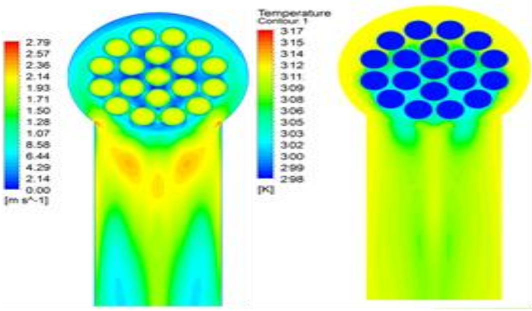
3M



5M



Outlet



D Vector Plots

Velocity vector plots can be seen below in Figures 9 and10. These plots give an idea of flow separation at inlet region and the impingement of the fluid on the tubes. The major portion of the fluid tends to move around the tube bundle, and part of the fluid enters the tube bundle through the tube spacing as seen in Figure 9. This region is a major reason of pressure drop due to impingement on the tube bundle. At the outlet, boundary layer separation takes place and the flow from the shell tends to mix with each other. This could be a non-symmetric region due to mixing of fluid from all sides.

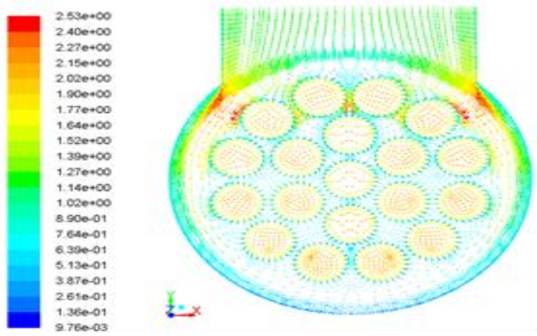


Fig. 9 Vector Plot of Velocity at Inlet

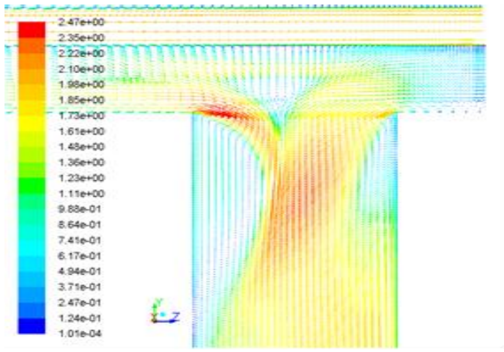
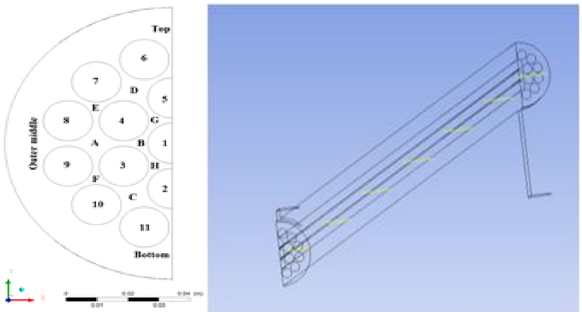


Fig. 10 Vector Plot of Velocity at Outlet

E Profiles

Temperature and velocity profiles are very useful to understand the heat transfer along with the flow distribution. The temperature profiles are drawn across the cross section and along the lengthof heat exchanger at different positions. Whereas, the velocity profiles are drawn only across the cross section. In order to understand the profiles, following Figures 11(a) and 11(b) must be understood first.



(a) Cross-Section **(b) Length**

Fig. 11 Profiling(a) and (b)

F Velocity Profile

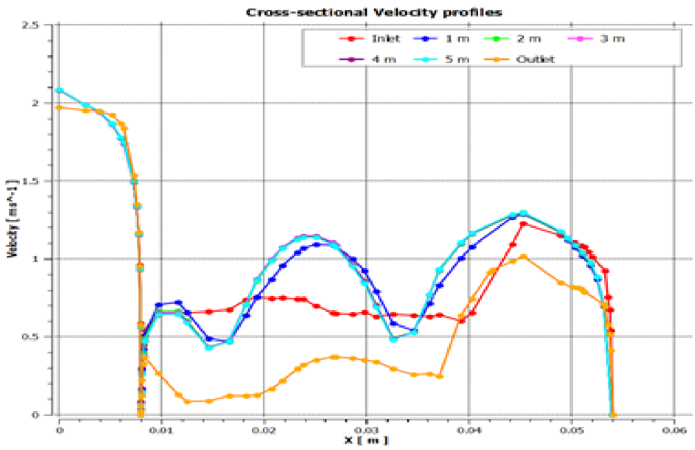


Fig. 12 Velocity Profiles Across the Cross-section at Different Positions in the Heat Exchanger

G Temperature Profiles

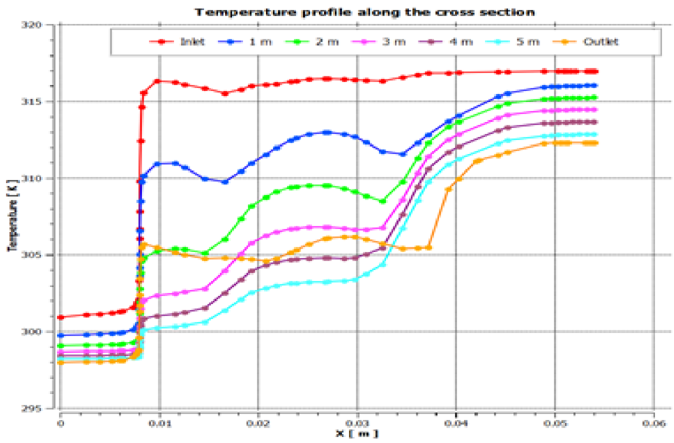


Fig. 13 Temperature Profiles Across the Cross-section of the shell at Different Positions in the Heat Exchanger

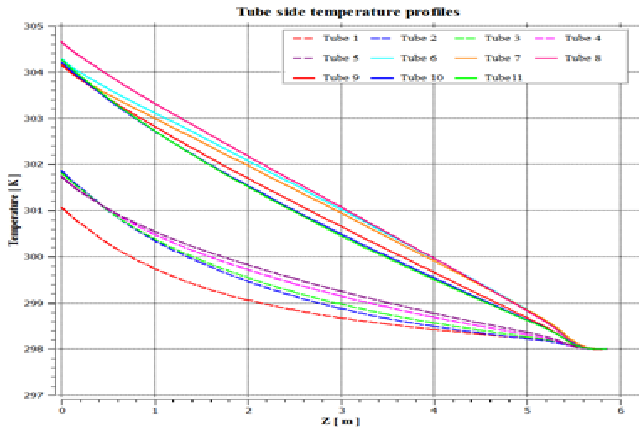


Fig. 14 Tube side Temperature Profiles along the Length of Heat Exchanger

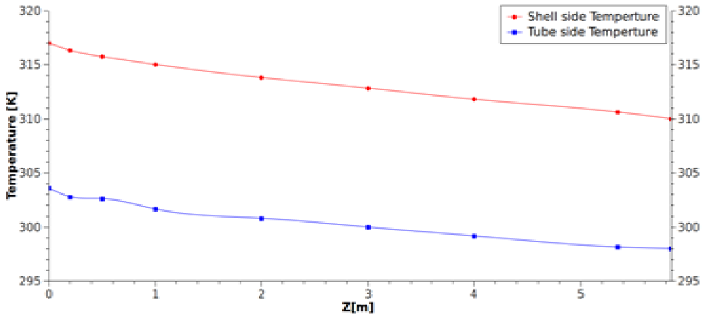


Fig15 Mass Averaged Shell and Tube side Temperatures

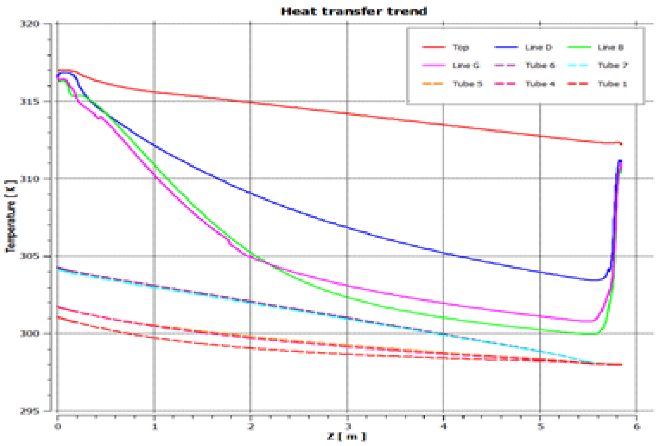


Fig. 16 Shell and Tube Side Temperature Profiles along the Length of Heat Exchanger

H Pressure Drop and Heat Transfer

Pressure drop along the length of heat exchanger can be seen in the Figure 18. It depicts the static pressure at inlet and outlet regions and along the length of tubes at different inlet velocities. The steeper inclination at the beginning and end of the graph shows the higher pressure drops atinlet and outlet regions. As described earlier, this happens due to cross-flow and impingement of the flow at inlet and outlet of the heat exchanger. Subsequently, heat transfer at these regions is higher as compared to the rest of heat exchanger. It can be seen in the Figure 19 that local heat transfer coefficient is very high at the inlet. This is due to several reasons, mainly being the cross flow at inlet. In addition, the temperature difference between the shell side and tube side fluid is much higher as observed in Figure16.

The study of both Figures 18 and 19 gives an idea of effect of cross-flow over the pressure drop and heat transfer. Certainly, creating cross-flow regions enhances the heat transfer at the cost of higher pressure drop. Thus it provides an insight about installing baffles for higher heat transfer in this thin and long heat exchanger.

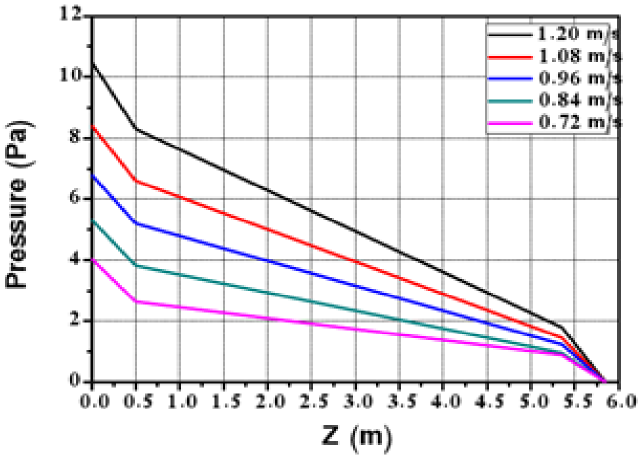


Fig.18 Shell Side Pressure Drop along the Length of Heat Exchanger

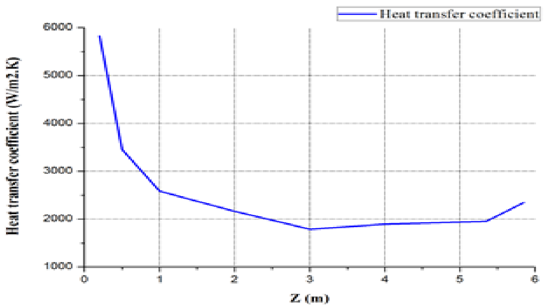


Fig.19: Heat Transfer Coefficient along the Length of Heat Exchanger

V. CONCLUSION

The heat transfer and flow distribution is discussed in detail and proposed model is compared with the experimental results as well. The model predicts the heat transfer and pressure drop with an average error of 20%. Thus the model still can be improved. The assumption of plane symmetry works well for most of the length of heat exchanger except the outlet and inlet regions where the rapid mixing and change in flow direction takes place. Thus improvement is expected if complete geometry is modeled. Moreover, SST $k-\omega$ model has provided the reliable results given the y+ limitations, but this model over predicts the turbulence in regions with large normal strain (i.e. stagnation region at inlet of the shell). Thus the modeling can also be improved by using Reynolds Stress Models, but with higher computational costs. Furthermore, the enhanced wall functions are not used in this project due to convergence issues, but they can be very useful with $k-\varepsilon$ models.

The heat transfer is found to be poor because the most of the shell side fluid by-passes the tube bundle without interaction. Thus the design can be modified in order to achieve the better heat transfer in two ways. Either, the shell diameter is reduced to keep the outer fluid mass flux lower or tube spacing can be increased to enhance the inner fluid mass flux. Just doing this might not be enough, because it is seen that the shell side fluid after 3m doesn't transfer heat efficiently. It is because the heat transfer area is not utilized efficiently. Thus the design can further be improved by creating cross-flow regions in such a way that flow doesn't remain parallel to the

ijates
ISSN 2348 - 7550

tubes. It will allow the outer shell fluid to mix with the inner shell fluid and will automatically increase the heat transfer.

REFERENCES

[1] "Aheat exchanger applications general." Heat_exchanger/heat_exchanger_application.htm, 2011.

[2] S. Kakac and H. Liu, "Heat exchangers: Selection, rating, and thermal performance," 1998.

[3] http://www.wcr-regasketing.com,"Heat exchanger applications." Http://Www.wcr-regasketing.com/heat-exchanger-applications.htm, 2010.

[4] D. Kern, Process Heat Transfer. Mcgraw-Hill, 1950.

[5] R. Serth, Process Heat Transfer, Principles and Applications. Elsevier Science and Tech- nology Books.

[6] E. Ozden and I. Tari, "Shell side cfd analysis of a small shell-and-tube heat exchanger,"Energy Conversion and Management, vol. 51, no. 5, pp. 1004 – 1014, 2010.

[7] J. J. Gay B, Mackley NV, "Shell-side heat transfer in baffled cylindrical shell and tube exchangers- an electrochemical mass transfer modelling technique," Int J Heat Mass Trans- fer, vol. 19, pp. 995–1002, 1976.

[8] G. V. Gaddis ES, "Pressure drop on the shell side of shell-and-tube heatexchangers with segmental baffles," ChemEng Process, vol. 36, pp.149–59, 1997.

[9] F. E. Idelchik, I.E., Handbook of hydraulic resistance. Hemisphere Publishing,New York, NY, second ed., 1986.

[10] M. J. Van Der Vyver H, Dirker J, "Validation of a cfd model of a three dimensional tube- in-tube heat exchanger.," 2003.

[11] S. B., "Computational heat transfer in heat exchangers," Heat Transfer Eng, pp. 24:895–7,2007.

[12] A. M. Prithiviraj M, "Shell and tube heat exchangers. Part 1: foundation and fluid mechan- ics," Heat Transfer, p. 33:799–816., 1998.

[13] K. T. C. S. Sha WT, Yang CI, "Multidimensional numerical modeling of heat exchangers.,"Heat Transfer, vol. 104, pp. 417–25, 1982.

[14] N. B. M. Bhutta, M.A.A Hayat, "Cfd applications in various heat exchangers design: AReview," Applied Thermal Engineering, vol. 32, pp. 1–12, 2011.

[15] J. G. Jae-Young, K. Afshin, "Comparison of near-wall treatment methods for high reynolds number backward-facing step flow," Int. J Computational Fluid Dynamics, vol. 19, 2005.

[16] E. Cao, Heat Transfer in Process Engineering. Mcgraw Hill, 2009.

[17] B. Andersson, R. Andersson, L. Ha˚kansson, M. Mortensen, R. Sudiyo, and B. V. Wachem,Computational Fluid Dynamics for Chemical Engineers. Sixth ed., 2010.

[18] H. K. Versteeg and M. W., An Introduction to Computational Fluid Dynamics, The FiniteVolume Method. Pearson Education Limited, 2007.

[19] "Ansys fluent theory guide." Http://www.ansys.com, 2010.

[20] "CFD-online wiki." Http://http://www.cfd-online.com/Wiki/Main_Page,2011.