Fluid flow and heat transfer analysis of shell and tube heat exchanger of variant tube pattern

[1]Rakesh Kumar Yadu, [2]Abhishek Kumar Gupta

[1]M.tech Scholar, [2]Assosiate Professor, Shri sankracharya institute of technology, Bhilai.
[1]NMechanical Engineering Department,
[1] Shri sankracharya institute of technology, Bhilai, India

Abstract: In the present paper, heat exchanger is of shell and tube type is proposed three models are compared with each other. The model predicts the heat transfer and pressure drop. In this computational fluid dynamics (CFD) technique which is a computer based analysis is used to simulate the heat exchanger involving fluid flow, heat exchanger. CFD resolve the entire heat exchanger in discrete elements to find the temperature gradients, pressure distribution and velocity vectors. The turbulence model k-ε is used for accurate results from CFD and the model can be rectifying in order to acquire improved heat transfer in twice ways. Either, reduce the shell diameter to keep the outer fluid mass flux lower or tube spacing can increase to enhance the inner fluid mass flux. Just doing this might not be enough, because it is seen that the shell side fluid after 3 m 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 does not remain parallel to the tubes. It will allow the outer shell fluid to mix with the inner shell fluid and will automatically increases the heat transfer.

***IndexTerms* – Computational fluid dynamics (CFD), Heat exchanger, k-ε Model.**

I. INTRODUCTION

Heat exchangers are one of the mostly useful equipment in the process industries. Heat exchangers are used to transfer heat between two process streams. One can realize their usage that any process which involve cooling, heating, condensation, boiling or evaporation will require a heat exchanger for these purpose. Process fluids, usually are heated or cooled before the process or undergo a phase change. Different heat exchangers are named according to their application. For example, heat exchangers being used to condense are known as condensers, similarly heat exchanger for boiling purposes are called boilers. Performance and efficiency of heat exchangers are measured through the amount of heat transfer 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 insight 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.

Thermal design considerations:

Thermal design of a shell and tube heat exchanger typically includes the determination of heat transfer area, number of tubes, tube length and diameter, tube layout, number of shell and tube passes, type of heat exchanger (fixed tube sheet, removable tube bundle etc.) tube pitch, number of baffles, its type and size, shell and tube side pressure drop etc.

1.1. Shell:

Shell is the container for the shell fluid and the tube bundle is placed inside the shell. Shell diameter should be selected in such a way to give a close fit of the tube bundle. The clearance between the tube bundle and inner shell wall depends on the type of exchanger. Shells are usually fabricated from standard steel pipe with satisfactory corrosion allowance. The shell thickness of 3/8 inch for the shell ID of 12-24 inch can be satisfactorily used up to 300 psi of operating pressure.

1.2. Tube:

Tube OD of ¾ and 1inch are very common to design a compact heat exchanger. The most efficient condition for heat transfer is to have the maximum number of tubes in the shell to increase turbulence. The tube thickness should be enough to withstand the internal pressure along with the adequate corrosion allowance. The tube thickness is expressed in terms of BWG (Birmingham Wire Gauge) and true outside diameter (OD). The tube length of 6, 8, 12, 16, 20 and 24 ft are preferably used. Longer tube reduces shell diameter at the expense of higher shell pressure drop. Finned tubes are also used when fluid with low heat transfer coefficient flows in the shell side. Stainless steel, admiralty brass, copper, bronze and alloys of copper-nickel are the commonly used tube materials.

1.3. Tube sheet:

The tubes are fixed with tube sheet that form the barrier between the tube and shell fluids. The tubes can be fixed with the tube sheet using ferrule and a soft metal packing ring. The tubes are attached to tube sheet with two or more grooves in the tube sheet wall by „tube rolling". The tube metal is forced to move into the grooves forming an excellent tight seal. This is the most common type of fixing arrangement in large industrial exchangers. The tube sheet thickness should be greater than the tube outside diameter to make a good seal. The recommended standards (TEMA) should be followed to select the minimum tube sheet thickness.

1.6.3 Baffles:

Baffles are used to increase the fluid velocity by diverting the flow across the tube bundle to obtain higher transfer co-efficient. The distance between adjacent baffles is called baffle-spacing. The baffle spacing of 0.2 to 1 times of the inside shell diameter is commonly used. Baffles are held in positioned by means of baffle spacers. Closer baffle spacing gives greater transfer co-efficient by inducing higher turbulence. The pressure drop is more with closer baffle spacing. In case of cut-segmental baffle, a segment (called baffle cut) is removed to form the baffle expressed as a percentage of the baffle diameter. Baffle cuts from 15 to 45% are normally used. A baffle cut of 20 to 25% provide a good heat-transfer with the reasonable pressure drop. The % cut for segmental baffle refers to the cut away height from its diameter.

II. LETERATURE REVIEW

Shell and tube heat exchanger design is normally based on correlations, among these, the Kern method [1] and Bell-Delaware method [2] 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 [3]. These correlations are developed for baffled shell and tube heat exchangers generally. Our studies aims at studying simple un-baffled heat exchanger, which is more similar to the double pipe heat exchangers. Almost no studies 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 [4] and pressure drop[5] 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 [6]. Handbook of hydraulic resistance provides the correlations for the pressure losses in these three regions separately by introducing the pressure loss coefficients. Entrance and exit losses are calculated by the equations 1.1 and 1.2 respectively.

$$\Delta P_{en} = (1 - \sigma_e^2 + K_c)\frac{G^2}{2\rho} \quad (1.1)$$

$$\Delta P_{ex} = -(1 - \sigma_e^2 - K_e)\frac{G^2}{2\rho} \quad (1.2)$$

Where,
G = Mass velocity (kg/m2s)
σ = Minimum Flow area / Frontal area = A1/A2
Kc = Entrance pressure drop coefficient
Ke = Exit pressure drop coefficient

As the heat exchanger under our study is un-baffled, thus making it similar to the straight annular pipe. Pressure drop in the shell side except the inlet and outlet regions can be estimated. side pressure drop and heat transfer rate results showed good agreement with experimental data. 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 [7]. 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 [8]. 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 [8]. 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 [9] 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 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. 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. 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 re-laminarization zones. According to a recent review [10], commercial and noncommercial software are used to model different types of heat exchangers. Normally, for modeling the flow, two equation models are the most commonly used models. k − ε models are mostly used in industrial designs along with wall functions. Jae et al [11] compared the different near wall treatment methods for high Reynolds number flows. It was found that non-equilibrium wall 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.

III. PROBLEM IDENTIFICATION

➢ To increase the efficiency of the heat exchanger we have to consider the optimal value of the surface contact area of shell fluid and the u-tube wall containing another fluid.

➢ To increase the contact time it is necessary for the shell fluid to remain in the shell for optimal period of time, for this we need to provide some hindrance inside the shell so that increase the fluid travelling time

➢ To increase the surface contact area and contact time we have to design the profile of U-Tubes in a definite pattern to get the optimal value of both.

➢ To avoid leakage for increasing the efficiency we have to provide O-Rings at the matching surfaces.

➢ Fluid pressure drop is controlled by a variety of design variables during heat exchanger engineering. When the process fluid flow and conditions are decided for the tube side stream, the controlling variables are shell diameter, tube length, tube geometry, (straight or U-tubes), number of tube passes and number of shells in series or parallel.

➢ Identifying cracked heat exchangers quickly and efficiently, saving time and money

➢ With the process of walking through inspections with your customer in order to prove your findings.

➢ Locating wear patterns and areas prone to defects on all makes and models of furnaces.

IV. METHODOLOGY

In this paper, the governing equations solved by FLUENT and the turbulence models used for this simulation are explained. Two equation models are used for the simulations. Flow equations and energy equations are described in detail. The wall treatment methods are also discussed and how they are important for modelling the heat transfer is also described.

4.1 Flow Calculation

The flow is governed by the continuity equation, the energy equation and Navier-Stokes momentum equations. Transport of mass, energy and momentum occur through convective flow and diffusion of molecules and turbulent eddies. All equations are set up over a control volume where i,j,k = 1,2,3 correspond to the three dimensions[12]. 4.2 Turbulence Modelling.

4.2. Definition of Turbulence

Turbulent flows have some characteristic properties which distinct them from laminar flows [12].

1. The motions of the fluid in a turbulent flow are irregular and chaotic due to random movements by the fluid. The flow has a wide range of length, velocity and time scales.
2. Turbulence is a three dimensional diffusive transport of mass, momentum and energy through the turbulent eddies that result in faster mixing rates.
3. Energy has to be constantly supplied or the turbulent eddies will decay and the flow will become laminar, the kinetic energy becomes internal energy.

Turbulence arises due to the instability in the flow. This happens when the viscous dampening of the velocity fluctuations is slower than the convective transport, i.e. the fluid element can rotate before it comes in contact with wall that stops the rotation. For high Reynolds numbers the velocity fluctuations cannot be dampened by the viscous forces and the flow becomes turbulent.

4.3 Wall Treatment Methods

The near-wall modelling considerably effects the reliability of numerical solutions, because walls are the major cause of mean vorticity and turbulence. Near the wall, gradients of variable such as velocity and pressure are high and other scalar variables also undergo sudden increase or decrease. So, precise estimation of flow variables in these regions is of major concern, which will lead to good predictions of turbulence as well [13].

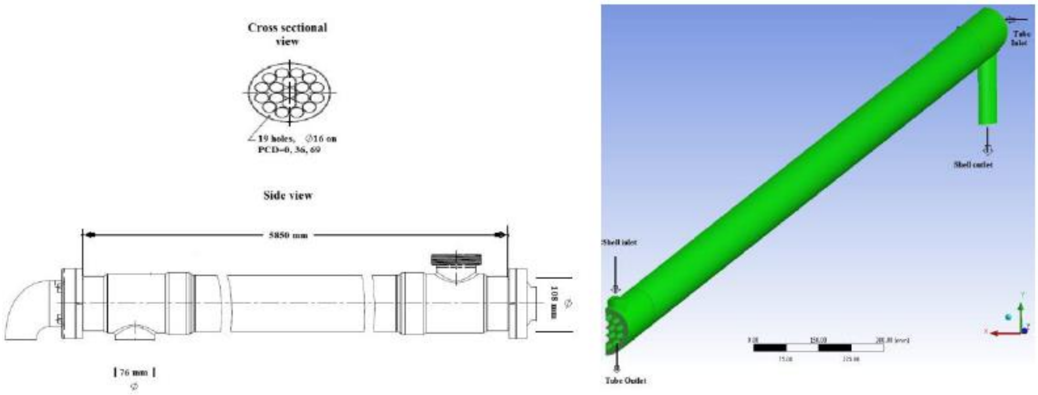
It is known that the region near wall can be divided into three sub sections. The section/layer next to the wall is named as viscous sub-layer. The flow in this layer is entirely laminar and molecular viscosity is major factor in calculating the heat and momentum transfer. In this region turbulent viscosity assumption is not valid at all. While, the section farthest from the wall inside the near wall region, is called the fully turbulent layer. Here the assumption of turbulent viscosity is valid and turbulence has a major effect over the heat and momentum transport. Then there is a transition region in between these two sections called buffer layer. In this layer, both molecular and turbulent viscosity is important [13].

4.4 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.

4.4.1 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 and 3 half tubes considering the symmetry. The shell outlet length is also increased to facilitate the modelling program to avoid the reverse flow condition. In the Fig. 1, the original geometry along with the simplified geometry can be seen.



(a) Original Geometry (b) Simplified Geometry

Fig. 1: systematic diagram of 3D model of shell and tube het exchanger

4.4.2 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.

4.5 Solution

4.5.1 Boundary Conditions

Boundary conditions are used according to the need of the model. The inlet velocities and temperature are used similar to the experimental conditions in order to have a comparison. 11 tubes have 11 similar inlet and outlet boundary conditions. General correlations 3.1 and 3.2 are used to estimate the turbulence boundary conditions which are specified by estimating the turbulence intensity and length scale.

$$I = 0.16Re^{-1/8} \quad (3.1)$$

$$l = 0.07L \quad (3.2)$$

Later it is seen that the turbulence boundary conditions have a very little affect over the results and solution. 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. The tube walls are set to 'coupled' for transferring of heat between shell and tube side fluids. The details about all boundary conditions can be seen in the Table 1.

Table 1: Boundary Conditions

	BC Type	Shell	Tube
Inlet	Mass flow rate	10kg/sec	5kg/sec
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	300K	500K

V. Results and Discussion

5.1 Model Comparison

Different models are evaluated to investigate their application for our case. Each model along with different wall treatment methods is used with fine mesh. A comparison obtained from these models can be seen in the Figures respectively. Knowing the temperatures from CFD results. Due to the available experimental data for comparison, three model is taken.

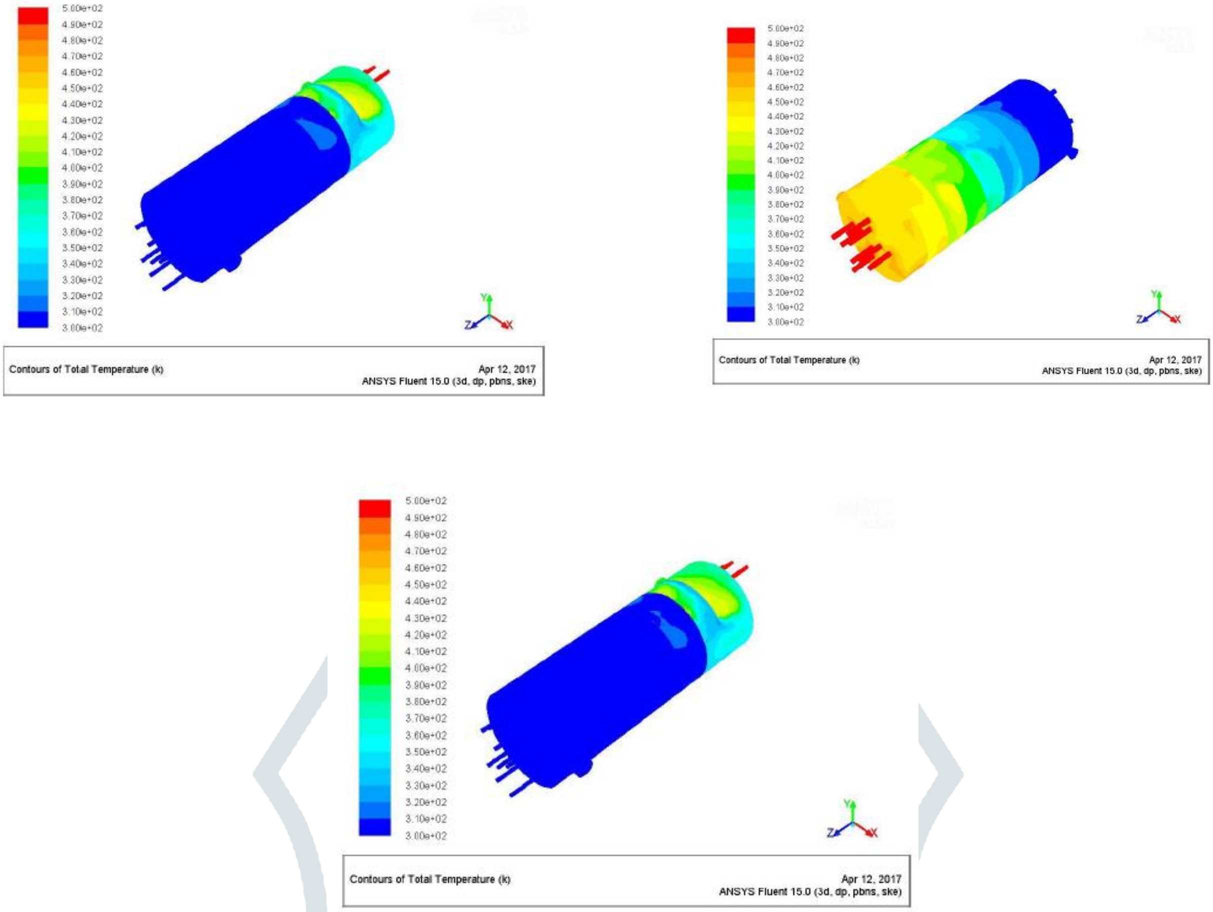


Fig. 2: Systematic 3D view of temperature contour of three different model

It is seen that the shell side inlet region of the heat exchanger involves boundary layer separation (adverse pressure gradient) and impinging flow on the tubes. Moreover, shell fluid's Reynolds number is found to be low in the core of the shell. Standard k − ε model is used at first to get a picture of the flow distribution but it is not good for predicting the boundary layer separation and impinging flows. Thus results are expected to be deviating from experimental results. Whereas, the Realizable k − ε model is known for predicting the flow separation better than Standard k − ε. For this reason, Realizable k − ε model is used with standard and then non-equilibrium wall functions. Non-equilibrium wall functions are better than standard wall functions because of their applicability in the regions of variable shear and departure from equilibrium. These wall functions also take into account the affect of high pressure gradient. The standard wall functions are over-predicting the pressure drop and heat transfer as well. Whereas, the non-equilibrium wall functions with Realizable k − ε model give better results than standard k–ε model. The pressure drop heat transfer still are being over-predicted by almost 25%, which is probably due to y+ values limitations at tube walls as mention.

Thus in order to avoid this and to include the low Reynold modification SST k − ω model can be also used. The reason being, it uses both k − ε and k − ω model in the region of high and low Reynolds number respectively.

The pressure drop in shell and tube side is respectively. The pressure drop in the shell is under-predicted by the SST k − ω model by almost 20-27%. This could be due to the several reasons including complicated geometry of the shell side and numerical diffusion.

5.2 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 Fig. 2 shows the whole length of heat exchanger. 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 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 different positions and the details of the temperature distribution in comparison to the velocity distribution can be observed.

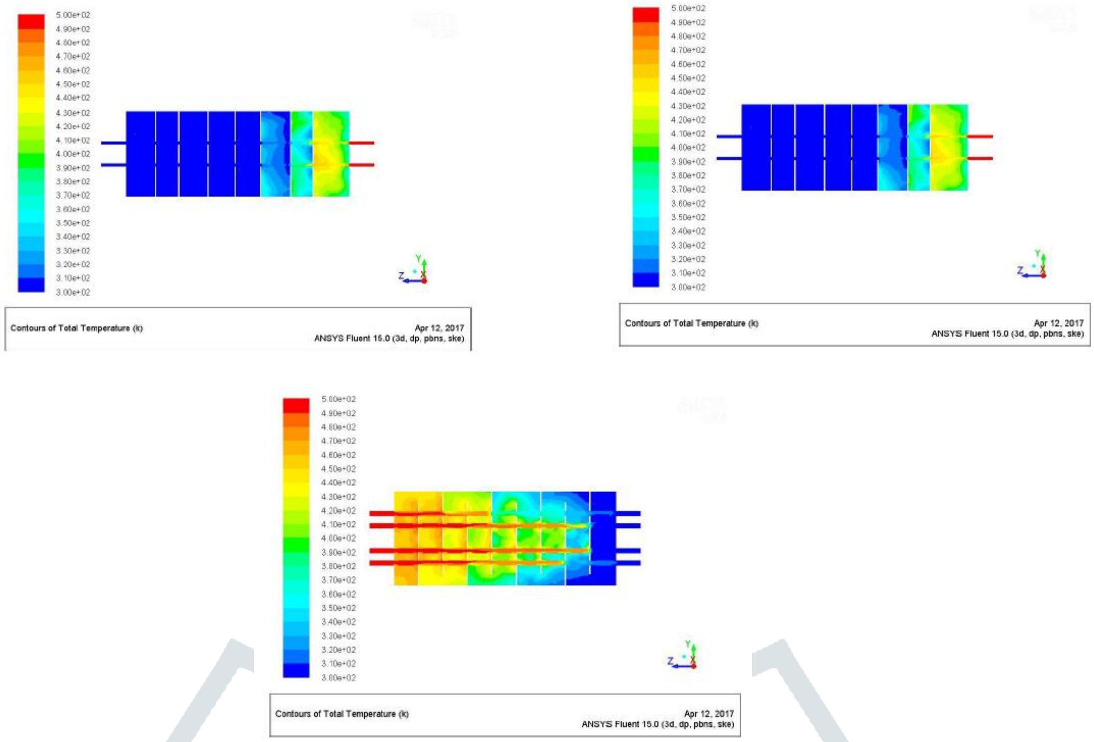


Fig.2: Section view of temperature conture

5.6 Pressure Profiles

Pressure profiles in the shell and tube side can be drawn in several ways. Similar to velocity profiles, pressure is also drawn across the cross section at different positions.

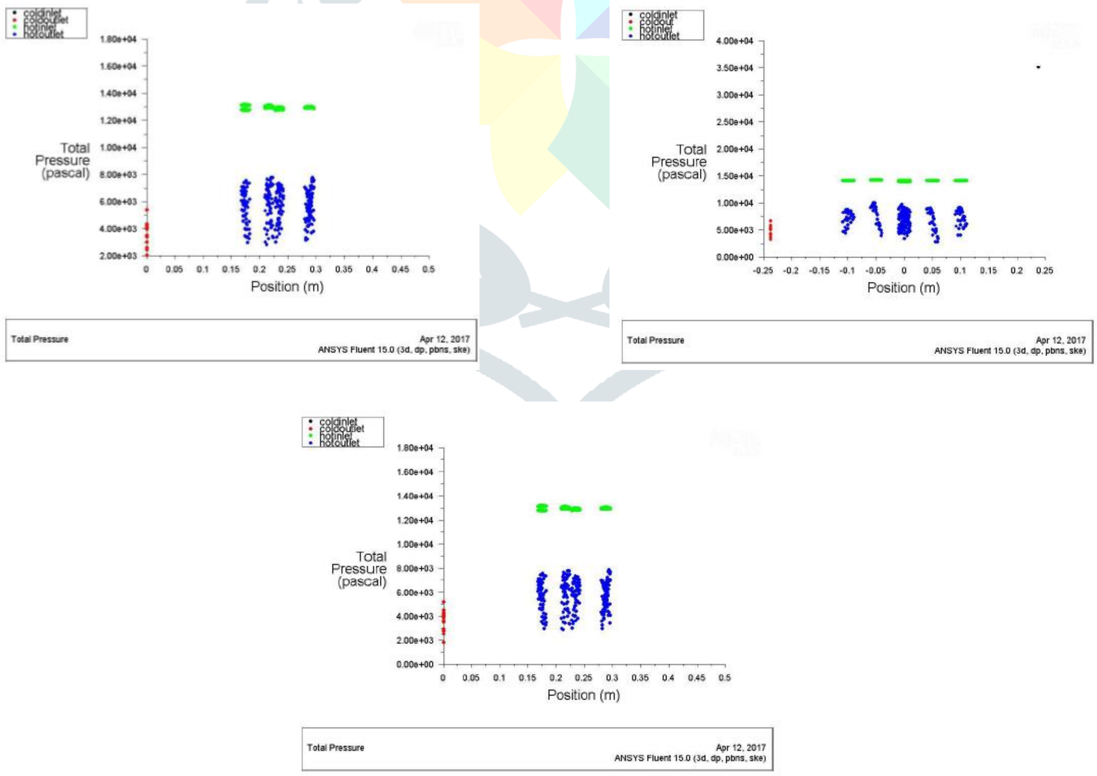


Fig. 3: Pressure variation graph of three different model.

Shell Side Temperature Profiles Graph in the Fig. 3 shows the pressure profile along the cross section of the shell according to Figure 4.11(b). The red line shows the temperature profile at the inlet, which is more or less constant. 1 meter away from inlet, temperature falls down due to heat transfer to the tubes. This fall in temperature is not same across the cross section of the heat exchanger. The peaks show the variation in shell side temperature across the cross section of shell. It is observed that the fluid near the center of the shell loses temperature much more than the fluid at the outer edge as obvious from the smallest peak near to the center and larger peak farthest from center. This trend is obeyed until outlet region of heat exchanger.

VI. CONCLUSIONS AND FUTURE WORK

The heat transfers and flow distribution is discussed in detail and proposed three models are compared with each other. The model predicts the heat transfer and pressure drop. 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, k 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 Reynold 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 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 tubes. It will allow the outer shell fluid to mix with the inner shell fluid and will automatically increase the heat transfer.

REFERENCES

[1] D. Kern, Process Heat Transfer. McGraw-Hill, 1950.
[2] R. Serth, Process Heat Transfer, Principles and Applications. Elsevier Science and Technology Books,.
[3] 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.
[4] 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 Transfer, vol. 19, pp. 995–1002, 1976.
[5] G. V. Gaddis ES, "Pressure drop on the shell side of shell-and-tube heat exchangers with segmental baffles," Chem Eng Process, vol. 36, pp. 149–59, 1997.
[6] F. E. Idelchik, I.E., Handbook of hydraulic resistance. Hemisphere Publishing,New York, NY, second ed., 1986.
[7] M. J. Van Der Vyver H, Dirker J, "Validation of a cfd model of a three dimensional tubein-tube heat exchanger.," 2003.
[8] S. B., "Computational heat transfer in heat exchangers," Heat Transfer Eng, pp. 24:895–7, 2007.
[9] A. M. Prithiviraj M, "Shell and tube heat exchangers. part 1: foundation and fluid mechanics," Heat Transfer, p. 33:799–816., 1998.
[10] N. B. M. Bhutta, M.A.A Hayat, "Cfd applications in various heat exchangers design: A review," Applied Thermal Engineering, vol. 32, pp. 1–12, 2011.
[11] 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, 2009.38
[12] B. Andersson, R. Andersson, L. Hakansson, M. Mortensen, R. Sudiyo, and B. V. Wachem,˚ Computational Fluid Dynamics for Chemical Engineers. sixth ed., 2010.
[13] "Ansys fluent theory guide." http://www.ansys.com, 2010.