SHELL SIDE FLOW BEHAVIOUR ANALYSIS WITH VARIOUS TUBE BUNDLE ALIGNMENT IN SHELL AND TUBE HEAT EXCHANGER USING CFD

GURUPRASATH.K[1], PUSHPARAJ.T[2]

[1]P.G SCHOLAR, DEPARTMENT OF MECHANICAL ENGINEERING, KINGS COLLEGE OF ENGINEERING, PUNALKULAM, TAMILNADU, INDIA

[2]ASSOCIATE PROFESSOR, DEPARTMENT OF MECHANICAL ENGINEERING, KINGS COLLEGE OF ENGINEERING, PUNALKULAM, TAMILNADU, INDIA

Abstract - *An un-baffled shell-and-tube heat exchanger design with respect to variable tube bundle stack arrangements are investigated by Computational Fluid Dynamics analysis. The heat exchanger contained 7 tubes inside a 0.600m long and 0.090m diameter shell. The flow and temperature fields inside the shell are resolved using a commercial CFD package ANSYS CFX. A set of CFD simulations is performed for a 60^0, 70^0, 80^0 and 90^0 stacked tube heat exchanger for various inlet flow conditions. The results are found to be sensitive to turbulence model due to the complex flow happens in the shell. 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 occurs 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. Higher heat flux variation occurs at shell's inlet and outlet due to two reasons. Firstly due to the cross-flow and secondly due to higher temperature difference between tubes and shell side fluid.*

Keywords-various flow profiles, SHTE, Velocity, Temperature, CFD, streamline, tube bundles

I.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[1]. Heat exchangers differ from mixing chambers in that they do not allow the two fluids involved to mix. Heat exchangers are manufactured in a variety of types, such as double pipe heat exchanger, compact heat exchanger, shell-and-tube heat exchanger, plate and frame (or just plate) heat exchanger, regenerative heat exchanger, etc. Heat exchangers are also given specific names such as condenser and boiler, to reflect the specific application for which they are used.

2. SHELL AND TUBE HEAT EXCHANGERS

The most common type of heat exchanger in industrial applications is the shell-and-tube heat exchanger. As its name implies, this type of heat
Exchanger consists of a shell (a large pressure vessel) and a large number of tubes (sometimes
Several hundred) packed in a shell with their axes parallel to that of the shell. Heat transfer takes place as one fluid flows inside the tubes while the other fluid flows outside the tubes through the shell. Shell-and-tube heat exchangers are further classified according to the number of shell and tube passes involved. Shell and tube heat exchangers are applicable for wide range of operating temperature and pressure [2, 3]. They have larger ratio of heat transfer surface to volume than double pipe heat exchanger and they are easy to manufacture in large variety of sizes and flow configuration. Shell and tube heat exchangers find widespread use in refrigeration, power generation, heating and air conditioning, chemical processes, petroleum, medical applications.

2.1 Design

Efforts are being made to enhance the performance of shell and tube heat exchangers. At a given iteration, if the performance of the considered design is calculated to be unsatisfactory, a better performing design can be obtained by changing the design parameters in the right direction. For example, baffles are placed in the shell to enhance heat transfer. Baffles are used for directing the flow inside the shell from the inlet to the outlet while maintaining effective circulation of the shell side fluid hence providing effective use of the heat transfer area. But even after installing baffles, the shell side flow still has a complicated structure due to the existence of baffles. Although it is relatively simple to adjust the tube side parameters, it is very hard to get the right combination for the shell side. Single segmental baffle that is used in the present study is the most common baffle type. Baffle is provided with a cut (%) which is expressed as the percentage of the segment height to shell inside diameter. In general, baffle cut can vary between 15% and 45% of the shell inside diameter. This cut allows the fluid to pass through in parallel or counter flow direction. A number of baffles are placed along the shell in alternating orientations (cut facing up, cut facing down, cut facing up again, etc.) in order to create flow paths across the tube bundle (forming cross flow windows). The spacing between baffles determines the structure of the stream[4].

2.2 Factors affecting the Performance

For a given shell geometry, the ideal configuration depends on the baffle cut, the baffle spacing and the baffle inclination angle. Even after fixing the right baffle cut and baffle space, the performance can be still improved by varying the baffle inclination angle. Having lower inclination angle, increases the heat transfer at the cost of increased shell side pressure drop. On the other hand, increasing the angle beyond a value might result in reduced pressure drop, but with lesser heat transfer.

2.3 Role of CFD in Shell and Tube Heat Exchanger Analysis

CFD techniques can be used both in the rating, and iteratively in achieving the optimum combination of baffle arrangement for the shell side. CFD is particularly useful during initial design steps, reducing number of testing of prototype and providing a good insight in the transport phenomenon occurring in the heat exchanger [5, 6].

3. TUBULAR HEAT EXCHANGER

A. Heat Transfer

Heat transfer is the process for all process industries, during the heat transfer, energy of the fluid at higher temperature transfers to the fluid at lower temperature. Fluid may transfer heat through different mechanisms. Three different mechanisms of the heat transfer are conduction, convection, radiation. Conduction and convection are the mostly used methods of heat transfer in process industries. Radiation is not the common mode in process industries but it plays a vital role in heat transfer like in combustion chamber. Actual heat transfer between high and low temperature fluids is not exactly equal due to losses and resistances in the form of wall fouling. But here assumption is made that the amount of heat transferred from the hot fluid is equal to the amount of heat transferred to the cold fluid .heat exchangers are isolated to minimize the environmental loss.

4. COMPUTATIONAL FLUID DYNAMICS (CFD)

CFD is an advanced computationally-based outline and investigation method. CFD programming provides the ability to reproduce streams of gasses and fluids, heat and mass exchange, moving bodies, multiphase material science, chemical reactions, liquid structure interaction and acoustics through workstation displaying. This product can likewise construct a virtual model of the framework or gadget before could be apply to true physical science and science to the model, and the product will give pictures and information, which anticipate the execution of that outline. Computational Fluid Dynamics (CFD) is valuable in a wide mixed bag of uses and use in industry. CFD is one of the limbs of liquid mechanics that uses numerical strategies and calculation might be utilized to tackle and investigate issues that include liquid streams furthermore recreate the stream over a channelling, vehicle or apparatus. Workstations are utilized to perform the great much estimation needed to reproduce the communication of liquids and gasses with the complex surfaces utilized within building. More precise codes that can precisely and rapidly mimic even intricate situations, for example, supersonic and turbulent streams are progressing exploration. Onwards the aeronautic trade has incorporated CFD procedures into the configuration, R&D and assembling of airplane and plane motors. All the more as of late the techniques have been connected to the configuration of inward burning motor, ignition assemblies of gas turbine and heaters additionally liquid streams and hotness move in high temperature exchanger. Progressively CFD is turning into a basic part in the configuration of modern items and procedures.

5. GOVERNING EQUATIONS

In shell and tube heat exchangers, three dimensional flow has been simulated by solving the governing equations (In equ (1) to equ (5)) viz. Energy, conservation of mass and momentum using ANSYS CFX code. Shear stress transport (SST) k-ωmodel of closure Takes care the turbulence which has a blending function that supports standard k-ε elsewhere. And k-ω nears the wall.

5.1 Flow Calculation

Flow is governing by the energy equation, navier-stokes momentum equation and continuity equations. Convective flow, diffusion of molecules and turbulent eddies are the reason for occurring of transport of mass, energy and momentum. In three dimensions, where i, j, k= 1, 2, 3 setup over a control volume corresponding with all equations.

5.2 Continuity Equation

Conservation of mass Is described by the continuity equation, And the equation is written as

$$\frac{\partial \rho}{\partial t}+\frac{\partial \rho U_1}{\partial x_1}+\frac{\partial \rho U_2}{\partial x_2}+\frac{\partial \rho U_3}{\partial x_3}=0 \qquad (1)$$

OR

$$\frac{\partial \rho}{\partial t}+\frac{\partial \rho U_i}{\partial x_i}=0, i=1,2,3 \qquad (2)$$

Equation (1 and 2) describes the amount through the fluid faces is equal to the rate of increase of mass in a control volume for constant density continuity equation is reduced to,

$$\frac{\partial \rho U_i}{\partial x_i} = 0, i = 1, 2, 3 \quad (3)$$

5.3 Momentum Equations (Navier-Stokes Equations)

The navier-stokes equation follows Newton's second law and it is also known as momentum balance equation. Newton's second law states, that The change in momentum in all directions equals the sum of forces acting in those directions. Surface forces and body forces are the two different kinds of forces acting on a finite element. Surface forces have the pressure force and the viscous force. And the body forces have the gravity, centrifugal and electro-magnetic forces.
For a Newtonian fluid the momentum equation in tensor notation can be written as in Equation 4,

$$\frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + \nu \frac{\partial}{\partial x_i}\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) + g_i \quad (4)$$

In addition to gravity, there can be external sources that may affect the acceleration of fluid e.g. magnetic and electrical fields. Strictly the momentum equations that form the Navier-Stokes equations but sometimes Navier-Stokes equation is the combination of continuity and momentum equations. And it is limited to macroscopic conditions.

5. 4 ENERGY EQUATION

Flow includes many forms of energy (i.e), as thermal energy, as chemically bounded energy, as kinetic energy due to the mass and velocity of the fluid. The sum of all the above energies is called the total energy. In kinetic energy the transport equation may be written as,

$$\frac{\partial (h_m)}{\partial t} = -U_j \frac{\partial (h_m)}{\partial x_j} + P\frac{\partial U_i}{\partial x_i} - \frac{\partial (PU_i)}{\partial x_i} - \frac{\partial}{\partial x_i}(\tau_{ij}U_i) - \tau_{ij}\frac{\partial U_i}{\partial x_j + \rho g U_i} \quad (5)$$

Where, h_m is the kinetic energy. The workdone by the gravity force is represented by the last term in equation 5. Similarly, by adding the source terms in the kinetic energy equation, a balance in heat can be formulated.

$$\frac{\partial (\rho C_p T)}{\partial t} = -U_j \frac{\partial (\rho C_p T)}{\partial x_j} + k_{eff}\frac{\partial^2 T}{\partial x_j x_j} - P\frac{\partial (U_j)}{\partial x_j} - \tau_{kj}\frac{\partial U_k}{\partial x_j} \quad (6)$$

The first on the right is convection term. The term on the left side of the equation is accumulation term, conduction is the second term on the right side, Expansion is the third term on right side, and the last term is the dissipation. These terms are used for the transformation between kinetic and thermal energy, (i.e.), dissipation and expansion occurs as source terms.

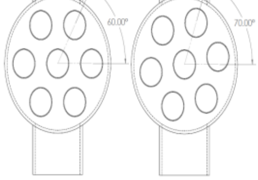
6. TURBULENCE MODELING

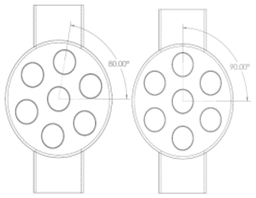
6.1 Definition of Turbulence

Turbulent flows have some characteristic properties which distinct them from laminar flows. 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. Turbulence is a three dimensional diffusive transport of mass, momentum and energy through the turbulent eddies that result in faster mixing rates. 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. 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 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.

6.2 STHE Configuration

In the analysis, the heat exchanger dimensions are taken from [1] and used to understand the fluid flow in the shell side.





The figures show various tube bundle configurations to be studied. Flow behaviour for various stack arrangements can be analyzed by computational models. The different arrangement of tube bundles exposes of different area of contact with fluid thereby heat exchange property changes. These different arrangements in the tube bundle disturb the fluid flow patterns within the shell. Disturbance causes variations in the heat exchange behaviour of the system. In all the cases the material for the heat exchanger is chosen as AISI 316 stainless steel as given in [2].

6.3 3D Model for CFD analysis

Pre-processing in the CFD analysis starts with creation of 3D models of heat exchanger. Since the complex fluid flow can't be visualized using 2D model, it is required to perform 3D fluid flow analysis. The required 3D models are given in an exploded view as follows. All configurations consist of one shell, two lids and seven tubes. These configurations of tubes are the variable parameter that is analyzed in this work.

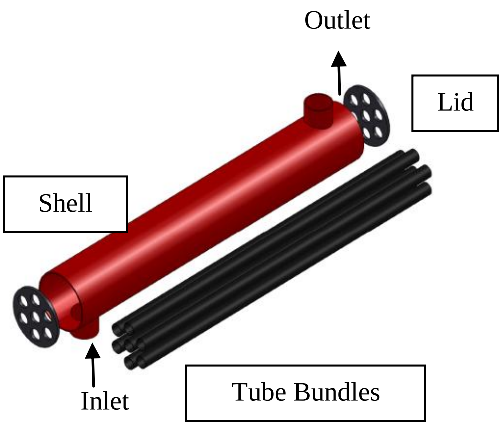


Fig.1. STHE 3D Model

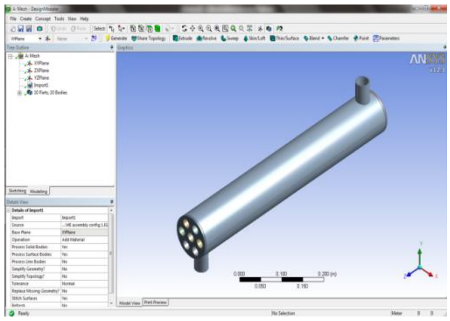
6.4 Fluid domain extraction steps

1) *Step 1:* The CAD model created using some third party software is saved as neutral file which can be accessed in ANSYS. In ANSYS Workbench environment the project was set up. In the project window, mesh module is taken. The geometry is loaded in the module for further processing.

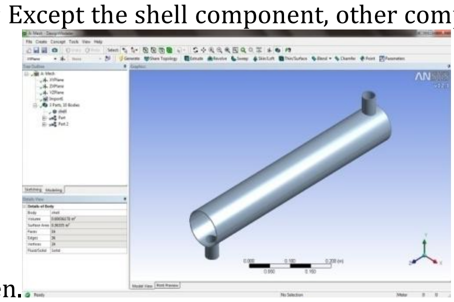
2)*Step 2:*After loading the CAD model in the geometry link, it is opened in Design Modular. Using design modular, the shell side fluid domain is extracted for further analysis.

3) *Step 3:*Appropriate unit is set once the design modular window is opened.

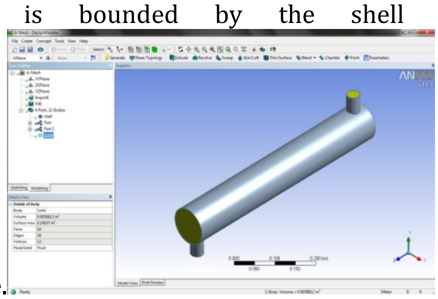
4) *Step 4:* Once the design modular is opened, using generate icon, the cad model is updated into this module.



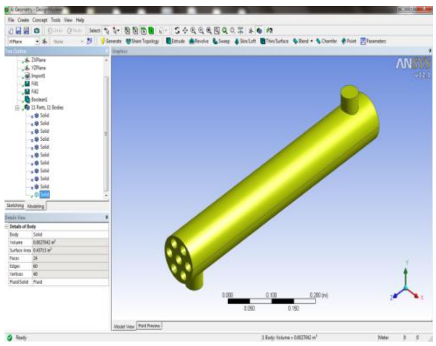
5) *Step 5:* Except the shell component, other components are hidden.



6)*Step 6:*Using fill option, the fluid domain is created which is bounded by the shell component surface.

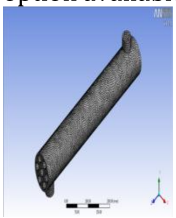


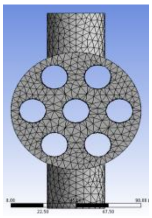
7) *Step 7:*After creating the shell side and tube side fluid domain using Boolean operation, shell side domain was subtracted. Then other shell component was hidden. The remaining fluid domain will be later meshed.



6.5 Problem Setup

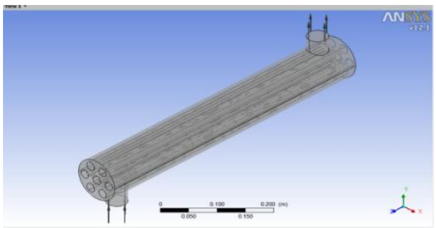
As mentioned in previous steps, the flow domain was extracted from the CAD model. It was meshed using tetrahedron elements available in the ANSYS mesh module with 5mm element size. The meshed model is shown in following figure. Element size was chosen based on the hardware capability to solve the problem. In the same way all the models were meshed and boundary zones were assigned using named selection option available in ANSYS.





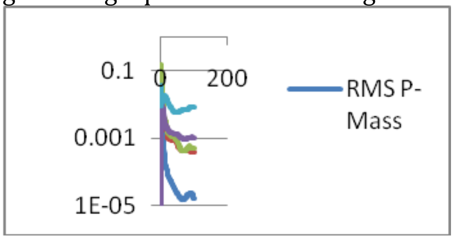
Boundary conditions were assigned on the meshed component using named selection option available in the module. Inlet condition and outlet condition were assigned on the appropriate faces. It was considered that the heat transfer between heat exchanger and atmosphere was negligible and the outer surface was assigned as insulation. But the heat transfer between shell side fluid and tube surface was significant so it was assigned with constant temperature. After meshing the mesh was transferred to ANSYS CFX module. In the CFX module the fluid properties were assigned as water. The problem was solved for four different velocities. So it was solved individually for various velocities. The velocities used in the problem were 0.002 m/s, 0.001 m/s, 0.0005 m/s, 0.00025 m/s. As shown in the following figure all the modules were coupled in the ANSYS workbench environment. The meshed model was transfer into CFX solver as follow.

In the CFX module, the required boundary conditions are assigned as given in the following figure.



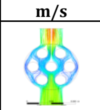
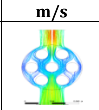
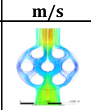
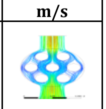
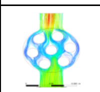
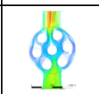
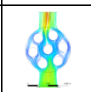
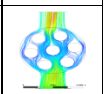
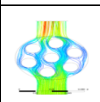
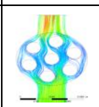
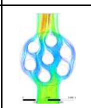
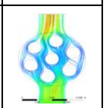
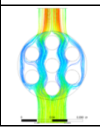
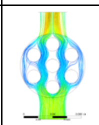
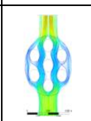
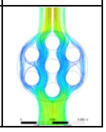
Inlet boundary was assigned with appropriate velocity and temperature of inlet fluid is assigned as 300K. In the outlet boundary the pressure value was assigned as 0Pa in order to create the pressure variation. As mentioned earlier the heat transfer between atmosphere and shell surface was negligible it was considered maintained at room temperature 300K. The heat transfer from hot fluid was assumed as constant temperature 500K. It was assigned on the inner tube surface of tube.

Once all the parameters were set, the problem was solved. Convergence of the results was monitored. The convergence plot is shown in the following graph. All the governing equations are converged.

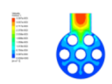
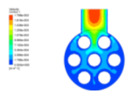
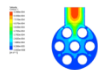
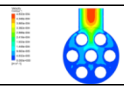
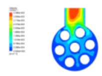
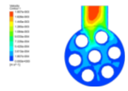
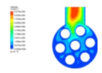
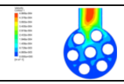
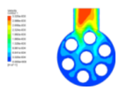
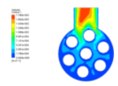
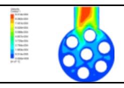
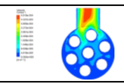
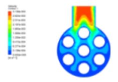
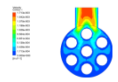
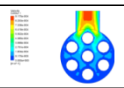
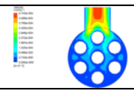


7. RESULTS AND DISCUSSIONS

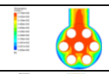
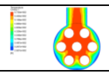
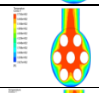
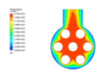
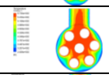
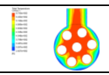
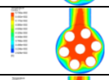
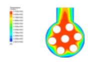
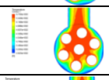
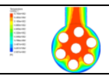
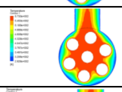
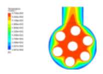
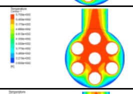
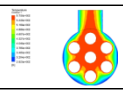
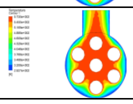
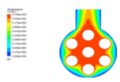
1) Fluid Streamline Profiles: The following table shows the results of streamline for all 16 different cases. Flow pattern for the various tube bundle arrangements show significant variation in the tube surface coverage. The flow is disturbed by the tubes arrangement. When the velocity was increased the fluid does not cover the tube surface properly thereby it will not good enough to observe the temperature from the hot fluid. Mean time the tube arrangement is prone to get faster failure due to direct impact of flow. If the speed is higher the impact of fluid on the tube is more and it leads to failure of hardware. It should be considered during the design process. It is clear from the following figure that the 90 degree tube bundle arrangement has direct impact of flow than other model. When the angle is reduced the flow path is diverted and the impact is reduced on the tube surface. Thereby the life of tube is increased. Also it is clear that the fluid flow with higher velocity does not cover the tube surface properly in all the cases compared to low speed flow. It is suggested that the speed should be maintained in such a way that the flow is sufficient to dissipate the temperature from the hot fluid. Based on the analysis the tube bundle with 60 degree arrangement show better tube surface coverage with less impact on the tube.

Flow patterns	0.002 m/s	0.001 m/s	0.0005 m/s	0.00025 m/s
60 deg				
70 deg				
80 deg				
90 deg				

2) Velocity Profiles: The velocity profile for various cases is given in the following table. The figures are arranged in clockwise direction start from left top. The results are taken at the outlet of heat exchanger. The contour plot shows variation in the velocity profile in all cases. It is clearly shown that the change of angle from 60 degree to 90 degree has significant disturbance in the fluid flow. The velocity profile is not symmetry when the angle is increased. It is the indication of the high impact fluid flow on the tube surface.From these results it is obvious that the flow coming out of the shell stopped by the tube closer to the outlet opening. Due to this change in angle flow is disturbed and the impact on the tube is increased. Mean time the tube surface coverage is significantly affected. The flow is symmetry in nature in the 60 degree and 90 degree arrangement but the 90 degree arrangement has high flow impact on the tube. it is suggested that the impact should be reduced. Compare to all the results, 60 degree arrangement shows better velocity profiles.

60 deg	Velocity		
			
70 deg	Velocity		
			
80 deg	Velocity		
			
90 deg	Velocity		
			

3) Temperature Profiles: The temperature profiles for various cases are given in the following table. The figures are arranged in clockwise direction start from left top. The results are taken at the outlet of heat exchanger. The contour plot shows variation in the temperature profile in all cases. Based on the result it is obvious that the temperature was observed from the tube surface and dissipated by the shell side fluid. It shows that the higher speed fluid flow could not cover entire surface of tube and temperature dissipation is not efficient. When the speed is reduced the temperature dissipation is enhanced. Meantime it is not good to have very high speed and very low speed. Because very slow fluid motion may have tube surface coverage but it will not have efficient dissipation. Temperature will be accumulated in very slow speed flows. From these figure it is obvious that the change in angle influence the tube surface coverage thereby the temperature distribution is disturbed. If the angle is at 60 degree the profile shows that the uniform coverage.The fluid flow speed is at 0.0005 m/s the temperature dissipation is more uniform and it has good tube surface coverage. By comparing all cases it can be suggested that the 60 degree angle with 0.0005 m/s speed shows better temperature dissipation.

60 degree	Temperature		
			
70 degree	Temperature		
			
80 degree	Temperature		
			
90 degree	Temperature		
			

8. APPLICATIONS OF THIS STUDY

CFD can be used to understand the flow behaviours of shell tube heat exchanger study by visualizing the flow patterns which cannot be seen in the experiment. Design and development of such heat exchanger model can be made efficiently using such computational approach. Various configuration can be studied without increasing the cost and time.

9. CONCLUSION

It is understood that the experimental observation of fluid flow for various tube bundle configuration cannot be made.CFD was used efficiently to understand the flow patterns in the shell and tube heat exchanger to visualize the flow patterns of various tube bundle configuration, velocity and temperature profiles. It is understand that increase in tube arrangements from 60 degree to 90 degree reducing its efficiency by increasing impact on tube surface. Mean time increasing fluid flow velocity reducing the tube surface coverage. But very low speed accumulates temperature in the shell side fluid. So the speed should be optimized in order to get proper surface coverage of tube and low impact on tube. By comparing all cases, it is suggested that 60 degree tube bundle arrangement shows symmetry flow pattern which reduces impact on tube surface. Also the speed at 0.0005 m/s it has very good tube surface coverage. Based on the parameters chosen in this study the 60degree tube arrangement with the fluid flow speed of 0.0005 m/s shows very good characteristics and suggested for the final design of this heat exchanger.

REFERENCES

[1] B.Ameel, J. Degroote, C. T'Joen, H. Huisseune, S.De Schampheleire, J. Vierendeels, *et al.*, "Accounting for the effect of the heat exchanger length in the performance evaluation of compact fin and tube heat exchangers," *Applied Thermal Engineering*, vol. 65, pp. 544-553, 2014/04// 2014.

[2] J. S. Jayakumar, S. M. Mahajani, J. C. Mandal, P. K. Vijayan, and R. Bhoi, "Experimental and CFD estimation of heat transfer in helically coiled heat exchangers," *Chemical Engineering Research and Design*, vol. 86, pp. 221-232, 2008/03// 2008.

[3] B. Selma, M. Désilets, and P. Proulx, "Optimization of an industrial heat exchanger using an open-source CFD code," *Applied Thermal Engineering*, vol. 69, pp. 241-250, 2014/08// 2014.

[4] W. Yongqing, G. Xin, W. Ke, and D. Qiwu, "Numerical Investigation of Shell-Side Characteristics of H-Shape Baffle Heat Exchanger," *Procedia Engineering*, vol. 18, pp. 53-58, 2011 2011.

[5] Q. Wang, Q. Chen, G. Chen, and M. Zeng, "Numerical investigation on combined multiple shell-pass shell-and-tube heat exchanger with continuous helical baffles," *International Journal of Heat and Mass Transfer*, vol. 52, pp. 1214-1222, 2009/02// 2009.

[6] J. Yang, L. Ma, J. Bock, A. M. Jacobi, and W. Liu, "A comparison of four numerical modeling approaches for enhanced shell-and-tube heat exchangers with experimental validation," *Applied Thermal Engineering*, vol. 65, pp. 369-383, 2014/04// 2014.

[7] Y. You, A. Fan, S. Huang, and W. Liu, "Numerical modeling and experimental validation of heat transfer and flow resistance on the shell side of a shell-and-tube heat exchanger with flower baffles," *International Journal of Heat and Mass Transfer*, vol. 55, pp. 7561-7569, 2012/12// 2012.

[8] Y. You, A. Fan, X. Lai, S. Huang, and W. Liu, "Experimental and numerical investigations of shell-side thermo-hydraulic performances for shell-and-tube heat exchanger with trefoil-hole baffles," *Applied Thermal Engineering*, vol. 50, pp. 950-956, 2013/01// 2013.

[9] F. Zaversky, M. Sánchez, and D. Astrain, "Object-oriented modeling for the transient response simulation of multi-pass shell-and-tube heat exchangers as applied in active indirect thermal energy storage systems for concentrated solar power," *Energy*, vol. 65, pp. 647-664, 2014/02// 2014.

[10] S. Zeyninejad Movassag, F. Nemati Taher, K. Razmi, and R. Tasouji Azar, "Tube bundle replacement for segmental and helical shell and tube heat exchangers: Performance comparison and fouling investigation on the shell side," *Applied Thermal Engineering*, vol. 51, pp. 1162-1169, 2013/03// 2013.