International Journal of Computational Research and Development (IJCRD)
Impact Factor: 4.775, ISSN (Online): 2456 - 3137
(www.dvpublication.com) Volume 1, Issue 2, 2016

PERFORMANCE AND OPTIMIZATION OF SHELL AND TUBE HEAT EXCHANGER USING CFD TOOLS BY CHANGING BAFFLE ARRANGEMENT

M. Bharanidharan* & S. Arul Angappan**
* PG student, SSM College of Engineering, Komarapalayam, Tamilnadu
** Assistant Professor, Department of Mechanical Engineering, SSM College of Engineering, Komarapalayam, Tamilnadu

Cite This Article: M. Bharanidharan & S. Arul Angappan, "Performance and Optimization of Shell and Tube Heat Exchanger Using CFD Tools by Changing Baffle Arrangement", International Journal of Computational Research and Development, Volume 1, Issue 2, Page Number 83-91, 2016.

Abstract:

The scope of this project is aimed at improving the amount of heat transfer in a shell and tube heat exchanger by varying the geometrical configuration of the baffles. The baffles through which the pipes go, increases the intensity of turbulence and effects better mixing of hot shell fluid over cold pipes. This increases the heat transfer rate one hand and on the other increases pressure drop as well. Thus we need more power to drive the fluid. This leads the design to find the optimum configuration of baffles. Though the number of empirical co-relations are available for the design of baffles, it is still complex because of different variables such as baffle-pitch, baffle- orientation, baffle- cut, number of baffles etc., Thus to optimize the baffle design one has to rely upon "protobased experimental technique". Because of cost, time involved in the experimental technique, modern industry effectively apply computational methodologies in modern days. In this work, ANSYS Fluent is used to analyze a shell and tube heat exchanger to arrive an optimum baffle configuration by conducting parametric study.

Key Words: Baffles, Orientation, Inclinations, Heat Transfer Co- Efficient

Introduction:

Heat exchangers are one of the mostly used 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 more 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. Heat exchangers are of two types:-

- ✓ Where both media between which heat is exchanged are in direct contact with each other is Direct contact heat exchanger,
- ✓ Where both media are separated by a wall through which heat is transferred so that they never mix, indirect contact heat exchanger.

A typical heat exchanger, usually for higher pressure applications up to 552 bars, is the shell and tube heat exchanger. Shell and tube type heat exchanger, indirect contact type heat exchanger. It consists of a series of tubes, through which one of the fluids runs. The shell is the container for the shell fluid. Generally, it is cylindrical in shape with a circular cross section, although shells of different shape are used in specific applications. For this particular study shell is considered, which a one pass shell is generally. A shell is the most commonly used due to its low cost and simplicity, and has the highest log-mean temperature-difference (LMTD) correction factor. Although the tubes may have single or multiple passes, there is one pass on the shell side, while the other fluid flows within the shell over the tubes to be heated or cooled. The tube side and shell side fluids are separated by a tube sheet. Baffles are used to support the tubes for structural rigidity, preventing tube vibration and sagging and to divert the flow across the bundle to obtain a higher heat transfer coefficient. Baffle spacing (B) is the centre line distance between two adjacent baffles, Baffle is provided with a cut (Bc) which is expressed as the percentage of the segment height to shell inside diameter. Baffle cut can vary between 15% and 45% of the shell inside diameter. In the present study 36% baffle cut (Bc) is considered. In general, conventional shell and tube heat exchangers result in high shell-side pressure drop and formation of recirculation zones near the baffles. Most of the researches now a day are carried on helical baffles, which give better performance than single segmental baffles but they involve high manufacturing cost, installation cost and maintenance cost. The effectiveness and cost are two important parameters in heat exchanger design. So, In order to improve the thermal performance at a reasonable cost of the Shell and tube heat exchanger, baffles in the present study are provided with some inclination in order to maintain a reasonable pressure drop across the exchanger. The complexity with experimental techniques involves quantitative description of flow phenomena using measurements dealing with one quantity at a time for a limited range of problem and operating conditions. Computational Fluid Dynamics is now an established industrial design tool, offering obvious advantages. In this study, a full 360° CFD model of shell and tube heat exchanger is considered. By modelling the geometry as accurately as possible, the flow structure and the temperature distribution inside the shell are obtained.

Related Works:

The purpose of this chapter is to provide a literature review of past research effort such as journals or articles related to shell and tube heat exchanger and computational fluid dynamics (CFD) analysis whether on two dimension and three dimension modelling. Moreover, review of other relevant research studies are made to provide more information in order to understand more on this research.

Purpose of Use of Helical Baffle:

A new type of baffle, called the helical baffle, provides further improvement. This type of baffle was first developed by Lutcha and Nemcansky. They investigated the flow field patterns produced by such helical baffle geometry with different helix angles. They found that these flow patterns were very close to the plug flow condition, which was expected to reduce shell-side pressure drop and to improve heat transfer performance. Stehlik et al. Compared heat transfer and pressure drop correction factors for a heat exchanger with an optimized segmental baffle based on the Bell–Delaware method, with those for a heat exchanger with helical baffles. Kral et al. discussed the performance of heat exchangers with helical baffles based on test results of various baffles geometries. One of the most important Geometric factors of the STHXHB is the helix angle. Recently a comprehensive comparison between the test data of shell-side heat transfer coefficient versus shell-side pressure drop was provided for five helical baffles and one segmental baffle measured for oil-water heat exchanger. It is found that based on the heat transfer per unit shell-side fluid pumping power or unit shell-side fluid pressured drop, the case of 400 helix angle behaves the best. The flow pattern in the shell side of the heat exchanger with continuous helical baffles was forced to be rotational and helical due to the geometry of the continuous helical baffles, which results in a significant increase in heat transfer coefficient per unit pressure drop in the heat exchanger. Properly designed continuous helical baffles can reduce fouling in the shell side and prevent the flow-induced vibration as well. The performance of the proposed STHXs was studied experimentally in this work. The use of continuous helical baffles results in nearly 10% increase in heat transfer coefficient compared with that of conventional segmental baffles for the same shell-side pressure drop. Based on the experimental data, the non- dimensional correlations for heat transfer coefficient and pressure drop were developed for the proposed continuous helical baffle heat exchangers with different shell configurations, which might be useful for industrial applications and further study of continuous helical baffle heat exchangers.

Problem Definition:

Based on the detailed literature survey, it is observed that by increasing the heat transfer co-efficient has to been investigated. So far we identified the baffle arrangement on by different types of helix angle or by in different configuration inside the heat exchanger May leads to hike the heat transfer coefficient and lowers the pressure drop more efficient. In this project going to optimizing and comparing about the different baffle arrangement and choose the optimal one to get more heat transfer co-efficient than other configure designs.

Computational Fluid Dynamics:

CFD is a sophisticated computationally-based design and analysis technique. CFD software gives you the power to simulate flows of gases and liquids, heat and mass transfer, moving bodies, multiphase physics, chemical reaction, fluid- structure interaction and acoustics through computer modelling. This software can also build a virtual prototype of the system or device before can be apply to real-world physics and chemistry to the model, and the software will provide with images and data, which predict the performance of that design. Computational fluid dynamics (CFD) is useful in a wide variety of applications and use in industry. CFD is one of the branches of fluid mechanics that uses numerical methods and algorithm can be used to solve and analyse problems that involve fluid flows and also simulate the flow over a piping, vehicle or machinery. Computers are used to perform the millions of calculations required to simulate the interaction of fluids and gases with the complex surfaces used in engineering. More accurate codes that can accurately and quickly simulate even complex scenarios such as supersonic and turbulent flows are ongoing research. Onwards the aerospace industry has integrated CFD techniques into the design, R & D and manufacture of aircraft and jet engines. More recently the methods have been applied to the design of internal combustion engine, combustion chambers of gas turbine and furnaces also fluid flows and heat transfer in heat exchanger (Figure 1). Furthermore, motor vehicle manufactures now routinely predict drag forces, under bonnet air flows and surrounding car environment with CFD. Increasingly CFD is becoming a vital component in the design of industrial products and processes.

Problem identification
↓
Computational design consideration
↓
Material specification, part diagram and assembly

↓
Optimization and investigation
↓
Result and discussion
↓
Conclusion

Computational Model for Heat Exchanger:

Problem Description:

Design of shell and tube heat exchanger with helical baffle using CFD. To study the temperature and pressure inside the tube with different mass flow rate.

Computational Model:

The computational model of an experimental tested Shell and Tube Heat Exchanger (STHX),the simulated STHX has six cycles of baffles in the shell side direction with total number of tube 7 .The whole computation domain is bounded by the inner side of the shell and everything in the shell contained in the domain. The inlet and outlet of the domain are connected with the corresponding tubes. To simplify numerical simulation, some basic characteristics of the process following assumption are made,

- ✓ The shell side fluid is constant thermal properties
- ✓ The fluid flow and heat transfer processes are turbulent and in steady state
- ✓ The leak flows between tube and baffle and that between baffles and shell are neglected
- ✓ The natural convection induced by the fluid density variation is neglected
- ✓ The tube wall temperature kept constant in the whole shell side
- ✓ The heat exchanger is well insulated hence the heat loss to the environment is totally Neglected.

Navier-Stokes Equation:

It is named after Claude-Louis Navier and Gabriel Stokes, He described the motion of fluid substances. It's also a fundamental equation being used by ANSYS and even in the present project work. These equation arise from applying second law of newton to fluid motion, together with the assumption that the fluid stress is sum of a diffusing viscous term, plus a pressure term. The derivation of the Navier Stokes equation begins with an application of second law of newton i.e. conservation of momentum. In an inertial frame of reference, the general form of the equations of fluid motion is

$$\partial_x u + \partial_y v = 0, \tag{1}$$

$$\partial_t u + u\partial_x u + v\partial_y u = -\partial_x p + \frac{1}{\mathrm{Re}}\left[\partial_x(\mu\partial_x u) + \partial_y(\mu\partial_y u) + \partial_x\mu\partial_x u + \partial_y\mu\partial_x v\right], \tag{2}$$

$$\partial_t v + u\partial_x v + v\partial_y v = -\partial_y p + \frac{1}{\mathrm{Re}}\left[\partial_x(\mu\partial_x v) + \partial_y(\mu\partial_y v) + \partial_y\mu\partial_y v + \partial_x\mu\partial_y u\right], \tag{3}$$

$$\partial_t T + u\partial_x T + v\partial_y T = -\frac{1}{\mathrm{Re}\ \mathrm{Pr}}\left[\partial_x(\kappa\partial_x T) + \partial_y(\kappa\partial_y T)\right], \tag{4}$$

Geometry and Mesh:

The model is designed according to TEMA (Tubular Exchanger Manufacturers Association) Standards Gaddis (2007).

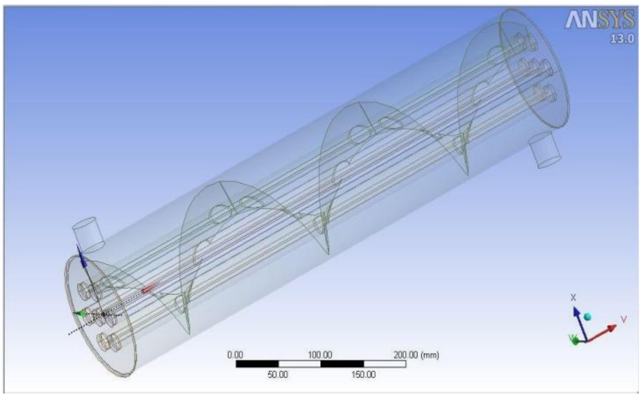

Figure 1: Isometric view of arrangement of baffles and tubes of shell and tube heat exchanger with baffle inclination.

Table 1: Geometric dimensions of shell and tube heat exchanger

Heat exchanger length, L	600mm
Shell inner diameter, Di	90mm
Tube outer diameter, do	20mm

Tube bundle geometry and pitch Triangular	30mm
Number of tubes, Nt	7
Number of baffles. Nb	6
Central baffle spacing, B	86mm

Grid Generation:

The three-dimensional model is then discretized in ICEM CFD. In order to capture both the thermal and velocity boundary layers the entire model is discretized using hexahedral mesh elements which are accurate and involve less computation effort. Fine control on the hexahedral mesh near the wall surface allows capturing the boundary layer gradient accurately. The entire geometry is divided into three fluid domains Fluid Inlet, Fluid Shell and Fluid Outlet and six solid domains namely Solid_Baffle1 to Solid_Baffle6 for six baffles respectively. The heat exchanger is discretized into solid and fluid domains in order to have better control over the number of nodes. The fluid mesh is made finer then solid mesh for simulating conjugate heat transfer phenomenon. The three fluid domains are as shown. The first cell height in the fluid domain from the tube surface is maintained at 100 microns to capture the velocity and thermal boundary layers. The discretized model is checked for quality and is found to have a minimum angle of 18° and min determinant of4.12. Once the meshes are checked for free of errors and minimum required quality it is exported to ANSYS CFX pre-processor.

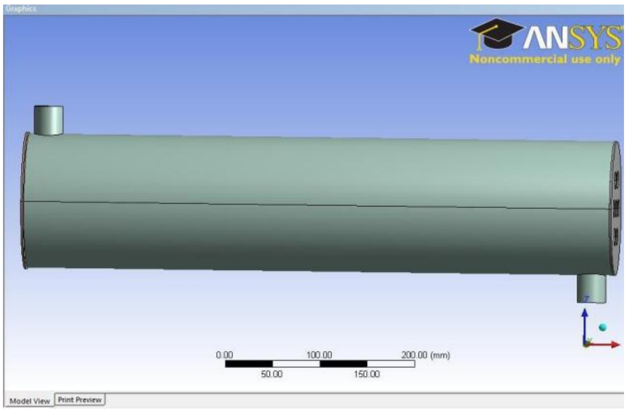

Figure 2: Complete Model of shell and tube heat exchanger

Meshing:

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.

Problem Setup:

Simulation was carried out in ANSYS® FLUENT® v13. In the Fluent solver Pressure Based type was selected, absolute velocity formation and steady time was selected for the simulation. In the model option energy calculation was on and the viscous was set as standard k-e, standard wall function (k-epsilon 2 eqn).

In cell zone fluid water-liquid was selected. Water-liquid and copper, aluminum was selected as materials for simulation. Boundary condition was selected for inlet, outlet. In inlet and outlet 1kg/s velocity and temperature was set at 353k.Across each tube 0.05kg/s velocity and 300k temperature was set. Mass flow was selected in each inlet. In reference Value Area set as 1m2, Density 998 kg/m3, enthalpy 229485 j/kg, length 1m, temperature 353k, Velocity 1.44085 m/s, Ration of specific heat 1.4 was considered.

Solution Initialization:

Pressure Velocity coupling selected as SIMPLEC. Skewness correction was set at zero. In Spatial Discretization zone Gradient was set as least square cell based, Pressure was standard, Momentum was First order Upwind, Turbulent Kinetic energy was set as First order Upwind, and Energy was also set as First order upwind. In Solution control, Pressure was 0.7, Density 1, Body force 1, Momentum 0.2, turbulent kinetic and turbulent dissipation rate was set at 1, energy and turbulent Viscosity was 1. Solution initialization was standard method and solution was initialize from inlet with 300k temperature.

Results:

Under the Above boundary condition and solution initialize condition simulation was set for 1000 iteration.

Convergence of Simulation:

The convergence of Simulation is required to get the parameters of the shell and tube heat exchanger in outlet. It also gives accurate value of parameters for the requirement of heat transfer rate. Continuity, X-velocity, Y-velocity, Z- velocity, energy, k, epsilion are the part of scaled residual which have to converge in a specific region. For the continuity,X-velocity ,Y-velocity, Z-velocity , k, epsilion should be less than 10^{-4} and the energy should be less than 10^{-7}. If these all values in same manner then solution will be converged.

0^0 Baffle Inclination:

For Zero degree baffle inclination solution was converged at 170th iteration. The following figure shows the residual plot for the above iterations

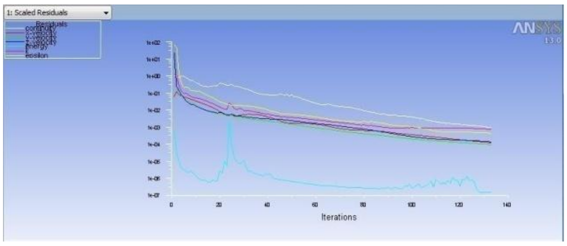

Figure 3: Conversion 0^0 Baffle inclination after 170th iteration

Variation of Temperature:

The temperature Contours plots across the cross section at different inclination of baffle along the length of heat exchanger will give an idea of the flow in detail. Three different plots of temperature profile are taken in comparison with the baffle inclination at 0^0, 10^0, 20^0

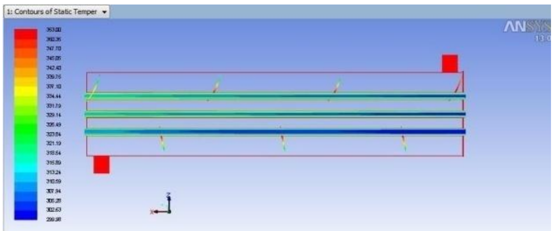

Figure 4: Temperature Distribution across the tube and shell

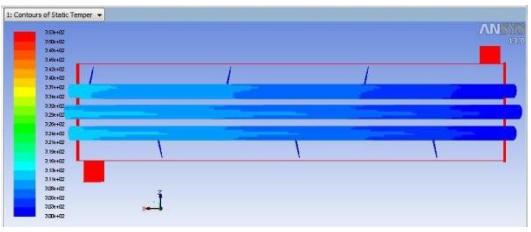

Figure 5: Temperature Distribution for 10^0 baffle inclination

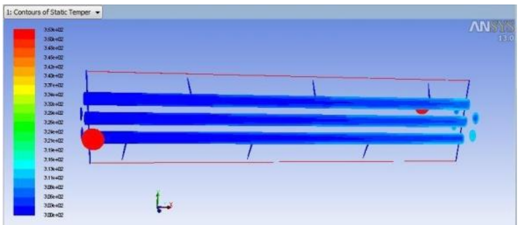

Figure 6: Temperature Distribution of 20^0 baffle inclination

Temperature of the hot water in shell and tube heat exchanger at inlet was 353k and in outlet it became 347k. In case of cold water inlet temperature was 300k and the outlet became 313k. Tube outlet Temperature Distribution was given below:

Exchanger:

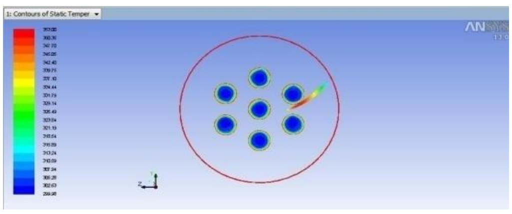

Figure 7: Temperature Distribution across Tube outlet in 0^0 baffle inclination

Variation of Velocity:

Velocity profile is examined to understand the flow distribution across the cross section at different positions in heat exchanger. Below in Figure shows the velocity profile of Shell and Tube Heat exchanger at different Baffle inclination. It should be kept in mind that the heat exchanger is modelled considering the plane symmetry. The velocity profile at inlet is same for all three inclination of baffle angle i.e 1.44086 m/s. Outlet velocity vary tube to helical baffle and turbulence occur in the shell region.

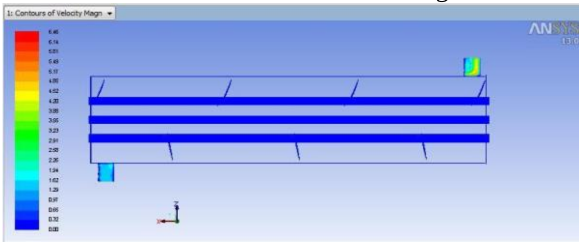

Figure 8: Velocity profile across the shell at 0^{0} baffle inclination.

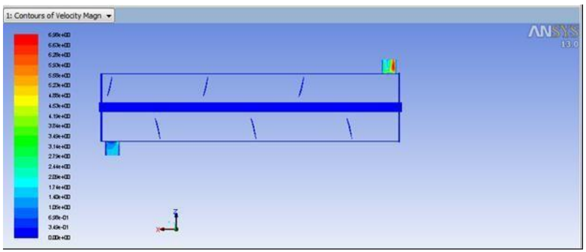

Figure 9: Velocity profile across the shell at 10^{0} baffle inclination.

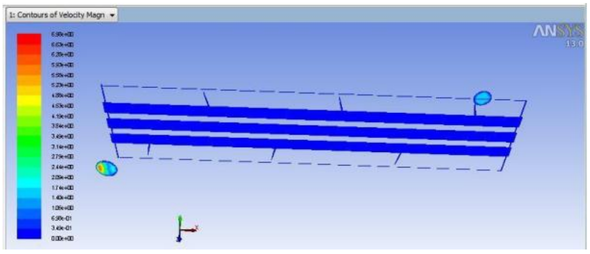

Figure 10: Velocity profile across the shell at 20^{0} baffle inclination.

Variation of Pressure:

Pressure Distribution across the shell and tube heat exchanger is given below in Fig. (14) (15) (16) .With the increase in Baffle inclination angle pressure drop inside the shell is decrease. Pressure varies largely from inlet to outlet. The contours of static pressure is shown in all the figure to give a detail idea.

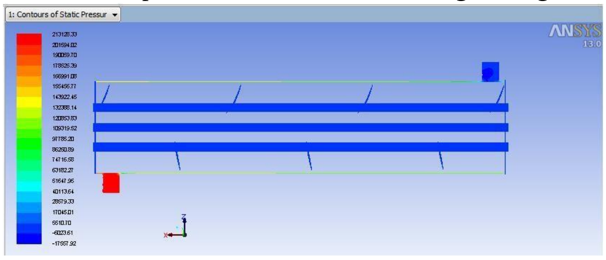

Figure 11: Pressure Distribution across the shell at 0^{0} baffle inclination.

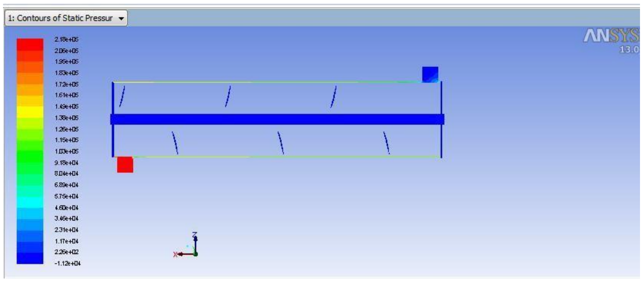

Figure 12: Pressure Distribution across the shell at 10^{0} baffle inclination

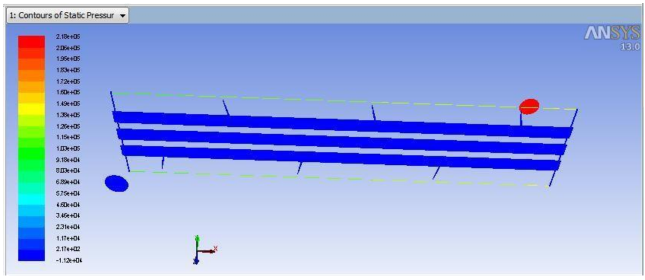

Figure 13: Pressure Distribution across the shell at 20^0 baffle inclination.

Table 2: The Outlet Temperature of the Shell side and Tube Side

Baffle Inclination Angle (Degree)	Outlet Temperature of Shell side	Outlet Temperature of Tube side
0	346	317
10	347.5	319
20	349	320

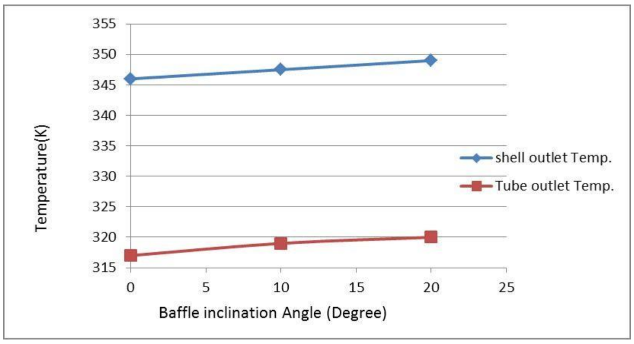

Figure 14: Plot of Baffle inclination angle vs Outlet Temperature of shell and tube side

It has been found that there is much effect of outlet temperature of shell side with increasing the baffle inclination angle from 0^0 to 20^0.

Table 4.2: The Pressure Drop inside Shell

Baffle Inclination Angle (Degree)	Pressure Drop Inside Shell (kPA)
0	230.992
10	229.015
20	228.943

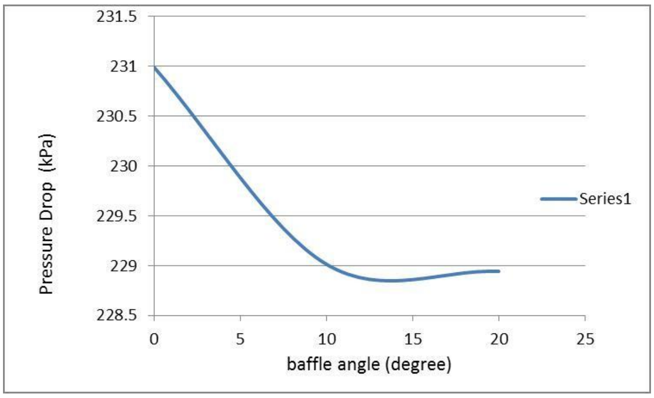

Figure 15: Plot of Baffle angle vs Pressure Drop

The shell-side pressure drop is decreased with increase in baffle inclination angle i. e., as the inclination angle is increased from 0° to 20°. The pressure drop is decreased by 4 %, for heat exchanger with 10° baffle inclination angle and by 16 % for heat exchanger with 20° baffle inclination compared to 0° baffle inclination heat exchanger as shown in fig. 18. Hence it can be observed with increasing baffle inclination pressure drop decreases, so that it affect in heat transfer rate which is increased.

Table 4.3: Velocity inside Shell

Baffle Inclination Angle (Degree)	Velocity inside shell (m/sec)
0	4.2
10	5.8
20	6.2

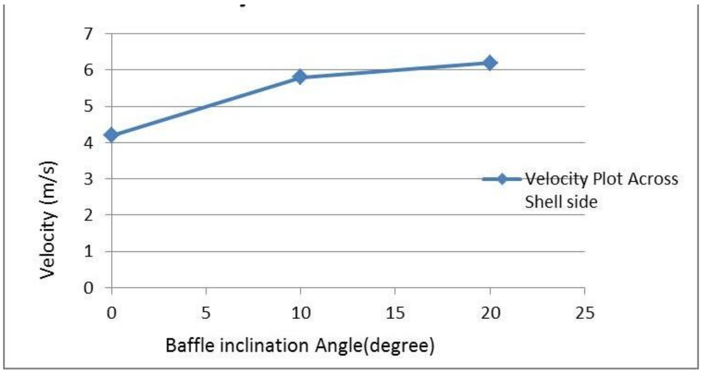

Figure 16: Plot of Velocity profile inside shell

The outlet velocity is increasing with increase in baffle inclination. So that more will be heat transfer rate with increasing velocity.

Heat Transfer Rate,

$$Q = m * Cp * \Delta T$$

m=mass flow rate

Cp = Speific Heat of Water

ΔT = Temperature Difference Between Tube Side

Table 4.4: Heat Transfer Rate across Tube side

Baffle Inclination Angle (Degree)	Heat Transfer Rate Across Tube side (w/m2)
0	3557.7
10	3972.9
20	4182

The heat transfer rate is calculated from above formulae from which heat transfer rate is calculated across shell side.

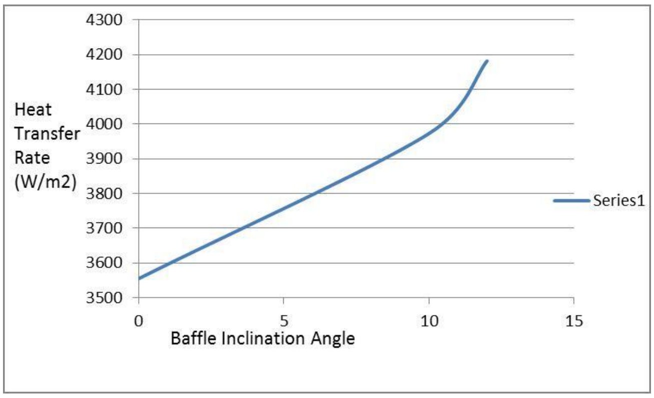

Figure 17: Heat Transfer Rate Along Tube side

The Plot showing the with increasing baffle inclination heat transfer rate increase. For better heat transfer rate helical baffle is used and the resulting is shown.

Table 4.5: Overall Calculated value in Shell and Tube heat exchanger in this simulation.

Baffle inclination (in Degree)	Shell outlet Temperature	Tube Outlet Temperature	Pressure Drop	Heat Transfer Rate(Q) (in W/m^2)	Outlet Velocity (m/s)
0^0	346	317	230.992	3554.7	4.2
10^0	347.5	319	229.015	3972.9	5.8
20^0	349	320	228.943	4182	6.2

- ✓ The shell side of a small shell-and-tube heat exchanger is modeled with sufficient detail to resolve the flow and temperature fields.
- ✓ The pressure drop decreases with increase in baffle inclination.

- ✓ The heat transfer rate is very slow in this model so that it affect the outlet temperature of the shell and tube side.

Conclusion:

The heat transfer and flow distribution is discussed in detail and proposed model is compared With increasing baffle inclination angle. The model predicts the heat transfer and pressure drop with an average error of 20%. Thus the model can be improved. The assumption worked well in this geometry and meshing expect the outlet and inlet region where rapid mixing and change in flow direction takes place. Thus improvement is expected if the helical baffle used in the model should have complete contact with the surface of the shell, it will help in more turbulence across shell side and the heat transfer rate will increase. If different flow rate is taken, it might be help to get better heat transfer and to get better temperature difference between inlet and outlet. Moreover the model has provided the reliable results by considering the standard k-e and standard wall function model, but this model over predicts the turbulence in regions with large normal strain. Thus this model can also be improved by using Nusselt number and Reynolds stress model, but with higher computational theory. Furthermore the enhance wall function are not use in this project, but they can be very useful. The heat transfer rate is poor because most of the fluid passes without the interaction with baffles. Thus the design can be modified for better heat transfer in two ways either the decreasing the shell diameter, so that it will be a proper contact with the helical baffle or by increasing the baffle so that baffles will be proper contact with the shell. 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 have contact with the inner shell fluid, thus heat transfer rate will increase.

References:

1. Emerson, W.H., "Shell-side pressure drop and heat transfer with turbulent flow in segmentally baffled shell-tube heat exchangers", Int. J. Heat Mass Transfer 6 (1963), pp. 649–66.
2. Haseler, L.E., Wadeker, V.V., Clarke, R.H., (1992), "Flow Distribution Effect in a Plate and Frame Heat Exchanger", IChemE Symposium Series, No. 129, pp. 361-367.
3. Diaper, A.D. and Hesler, L.E., (1990), "Crossflow Pressure Drop and Flow Distributions within a Tube Bundle Using Computational Fluid Dynamic", Proc. 9th Proc. 9th Heat Transfer Conf., Israel, pp. 235-240.
4. ian-Fei Zhang, Ya-Ling He, Wen-Quan Tao , " 3d numerical simulation of shell and tube heat exchanger with middle-overlapped helical baffle", a journal ,School of energy and power engineering,china.
5. Li, H., Kottke, "Effect of baffle spacing on pressure drop and local heat transfer in shell and tube heat exchangers for staggered tube arrangement", source book on Int. Heat Mass Transfer 41 (1998), 10, pp. 1303–1311.
6. Thirumarimurugan, M., Kannadasan, T., Ramasamy, E., Performance Analysis of Shell and Tube Heat Exchanger Using Miscible System, American Journal of Applied Sciences 5 (2008), pp. 548-552.
7. Usman Ur Ehman , G¨oteborg, Sweden 2011, Master's Thesis 2011:09 on "Heat Transfer Optimization of Shell-and-Tube & Heat Exchanger through CFD".
8. Professor Sunilkumar Shinde, Mustansir Hatim Pancha / International Journal of Engineering Research and Applications (IJERA) ,"Comparative Thermal performance of shell and tube heat Exchanger with continuous helical baffle using ", Vol. 2, Issue4, July-August 2012.
9. Khairun Hasmadi Othman, " CFD simulation of heat transfer in shell and tube heat exchnager", A thesis submitted in fulfillment for the award of the Degree of Bachelor in chemical Engineering (Gas Technology),April 2009.