North American Academic Research

Journal homepage: http://twasp.info/journal/home

Research

Characteristics of shell and tube heat exchanger by design and computational dynamics

Hafiz Muhammad Azeem Akhtar[1*], Muhizi Musa[1], Yang Yunru[1]

[1]*College of Mechanical and Electronics Engineering, Shandong University of Science and Technology. Qingdao, Shandong Province, P. R. China, Post Code: 266590*

[*]***Corresponding author***

***Accepted**: 01 March, 2020; **Online**: 11 March, 2020*

***DOI** : https://doi.org/10.5281/zenodo.3706378*

Abstract: *A heat exchanger is a gadget that is utilized to move heat vitality (enthalpy) between two or more liquids, between a strong surface and a liquid, or between strong particulates and a liquid, at various temperatures and in heat contact. From various sorts of heat exchangers the shell and cylinder heat exchangers with straight cylinders and single pass is to be under examination. Here the update happens in view of temperature vacillation at the ninth zone of the pasteurizer. Heat and mechanical structure is run so as to upgrade the yield temperature of the cool liquid at the last heat exchanger in which it is splashed on the lager prepared for client use. In heat structure part geometry enhancement is finished through experimentation. Furthermore, for Mechanical structure part the regular frequency& vortex shedding of various parts of heat exchangers are researched through overseeing conditions of vibrations under unique liquid with in tubes. Utilizing computational liquid elements (CFD) the heat move of the two liquid is examined utilizing FEM recreation programming's Gambit1.3 and Fluent 6.1and the presentation of the STHEx decided in wording of factors, for example, pressure, temperature, stream rate, vitality input/yield, mass stream rate and mass exchange rate that are specifically compelling in STHEx examination.*

***Keywords**: Shell and tube heat exchanger, design, modelling*

Introduction

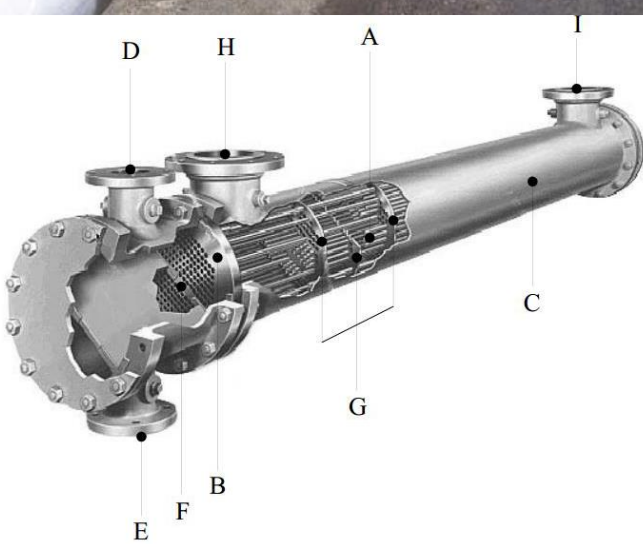
Heat exchangers are one of the mostly used equipment in the process industries. Heat exchangers are used to transfer heat between two process streams. One can realize their usage that any process which involve cooling, heating, condensation, boiling or evaporation will require a heat exchanger for these 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 is indirect contact type heat exchanger. The details are provided in Fig. 1.2.1, 1.2.2 and 1.2. 3. 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 is generally a one pass shell. 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 center 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 per cent and 45 per cent of the shell inside diameter. In the present study 36 per cent 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



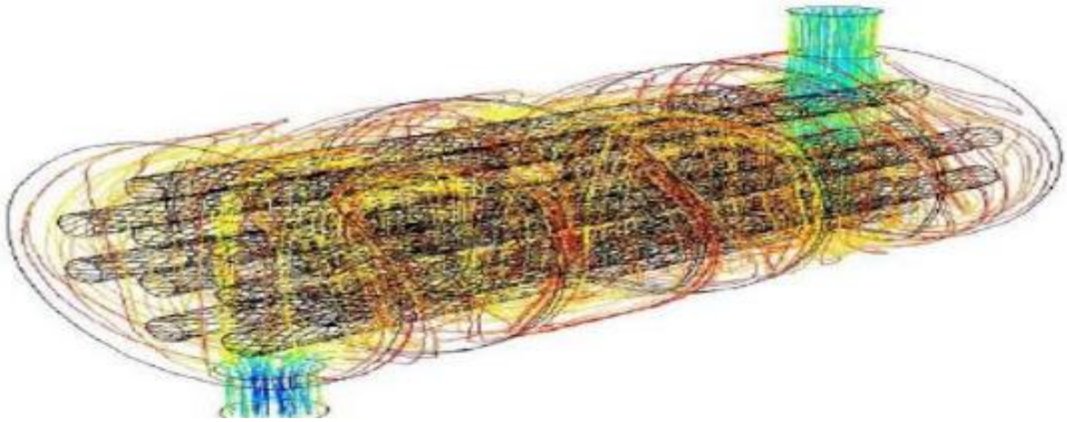


Fig. 1 : Tube Bundle projected outside the Shell. Fig. 1.2.3. Parts of typical shell and tube heat exchanger. A: Tubes, B: Tube Sheets, C: Shell, D: Tube-Side Inlet (Outlet) Nozzle, E: Tube-Side Outlet (Inlet) Nozzle, F: Pass Divider, G: Baffles, H: Shell-Side Inlet (Outlet) Nozzle, I: Shell-Side Outlet (Inlet) Nozzle.

The primary goal of this task is the structure and recreation of shell and cylinder heat exchanger with helical astound utilizing devices to diminish shell-side weight drop and to improve heat move execution by differing helix edge.

MATERIALS AND METHODS

ANSYS

Ansys is the finite element analysis code widely used in computer aided engineering (CAE) field. ANSYS software help us to construct computer models of structure, machine, components or system, apply operating loads and other design criteria, study physical response such as stress level temperature distribution, pressure etc. In Ansys, following Basic step is followed:

- During preprocessing the geometry of the problem is defined. Volume occupied by fluid is divided into discrete cells (the mesh). The mesh may be uniform or non-uniform. The physical modelling is defined. Boundary condition is defined. This involves specifying the fluid behavior of the problem. For transient problem boundary condition are also defined.

- The simulation is started and the equation are solved iteratively as steady state or transient.

- Finally a post procedure is used for the analysis and visualisation of the resulting problem.

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) with 10 helix angle is shown in Fig. 2.2.4.6., and the geometry parameters are listed in Table 2.2.4.1. As can be seen from Fig. 2.2.4.6., 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 out let of the domain are connected with the corresponding tubes.

To simplify numerical simulation, some basic characteristics of the process following assumption are made:

1. The shell side fluid is constant thermal properties

2. The fluid flow and heat transfer processes are turbulent and in steady state

3. The leak flows between tube and baffle and that between baffles and shell are neglected

4. The natural convection induced by the fluid density variation is neglected

5. The tube wall temperature kept constant in the whole shell side

6. 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 (Temam, 1977), 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:

$$\partial_x u + \partial_y v = 0,$$

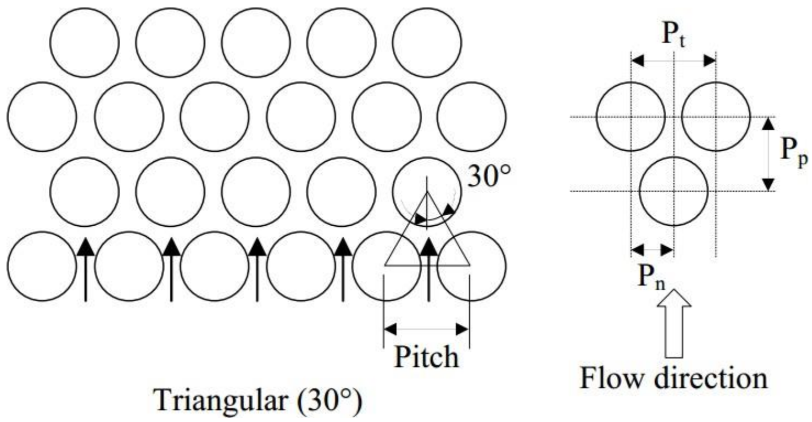
$$\partial_t u + u\partial_x u + v\partial_y u = -\partial_x p + \frac{1}{Re}\left[\partial_x(\mu\partial_x u) + \partial_y(\mu\partial_y u) + \partial_x\mu\partial_x u + \partial_y\mu\partial_x v\right]$$

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

$$\partial_t T + u\partial_x T + v\partial_y T = -\frac{1}{RePr}\left[\partial_x(K\partial_x T) + \partial_y(K\partial_y T)\right]$$

Geometry and Mesh:

The model is designed according to TEMA (Tubular Exchanger Manufacturers Association) Standards, Gaddis (2007) and the tube layout is provided in Fig.



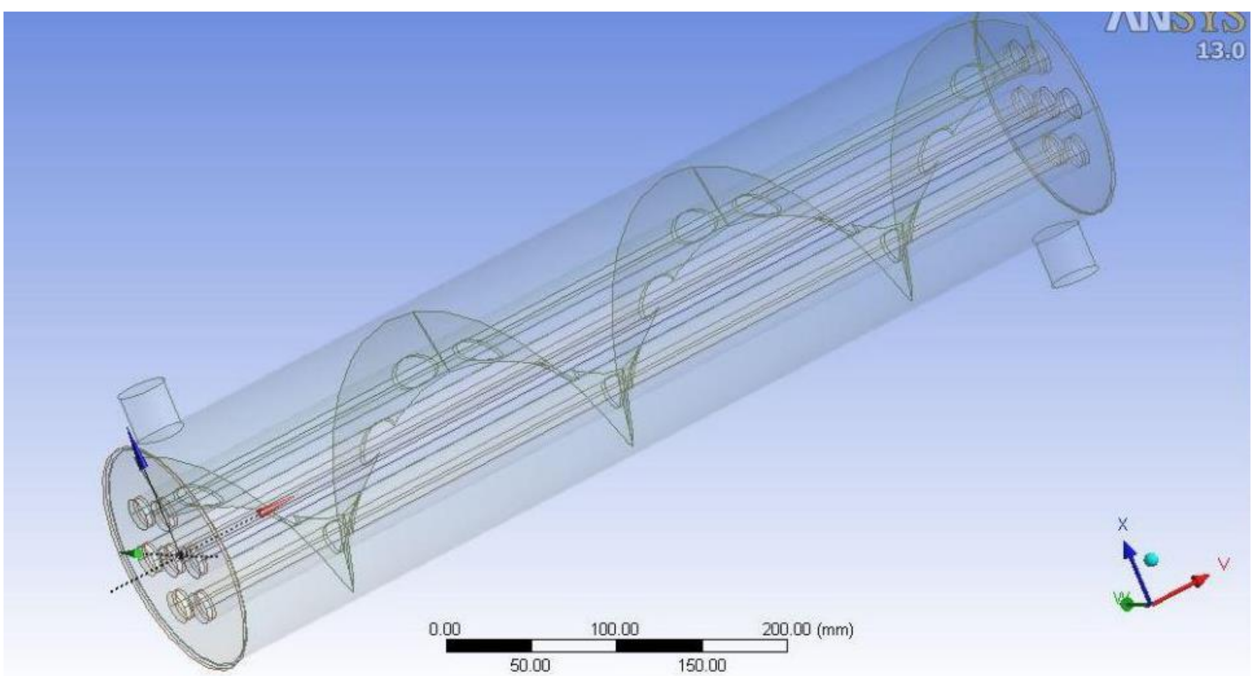


Fig. Isometric view of arrangement of baffles and tubes of shell and tube heat exchanger with baffle inclination.

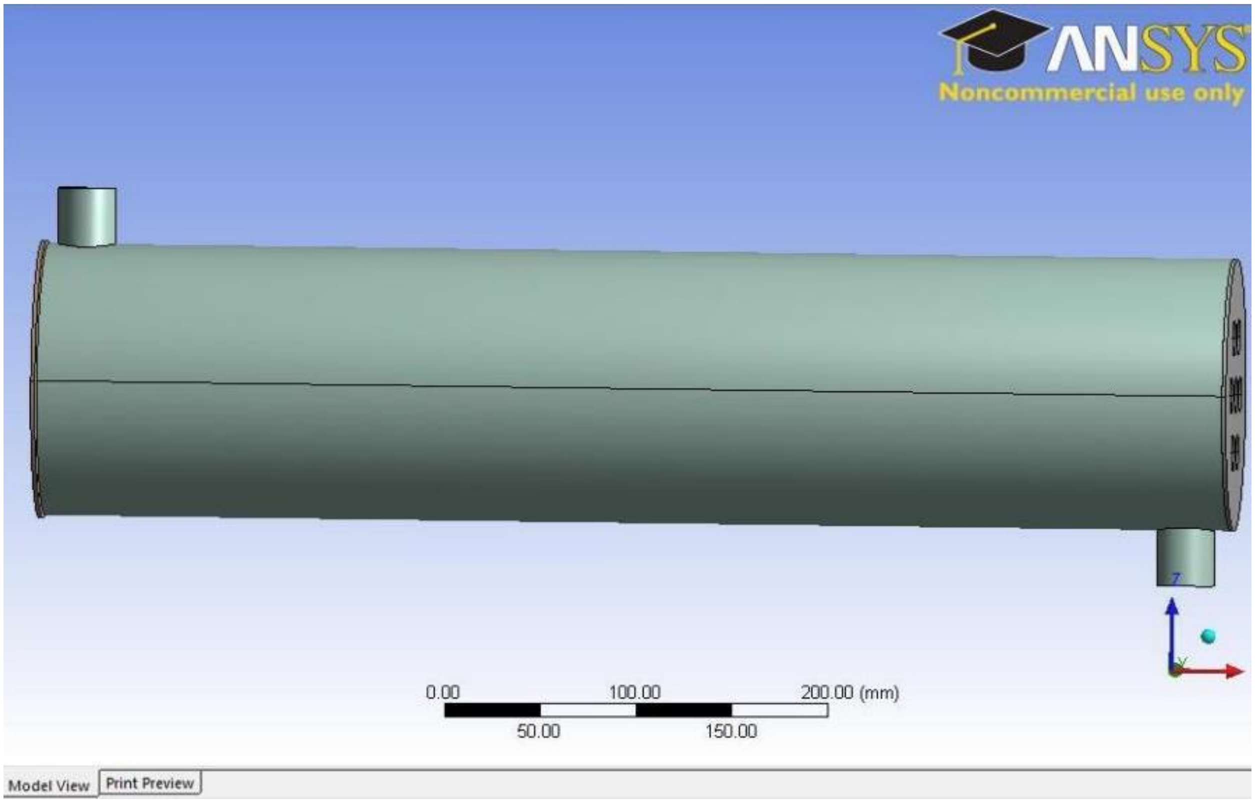
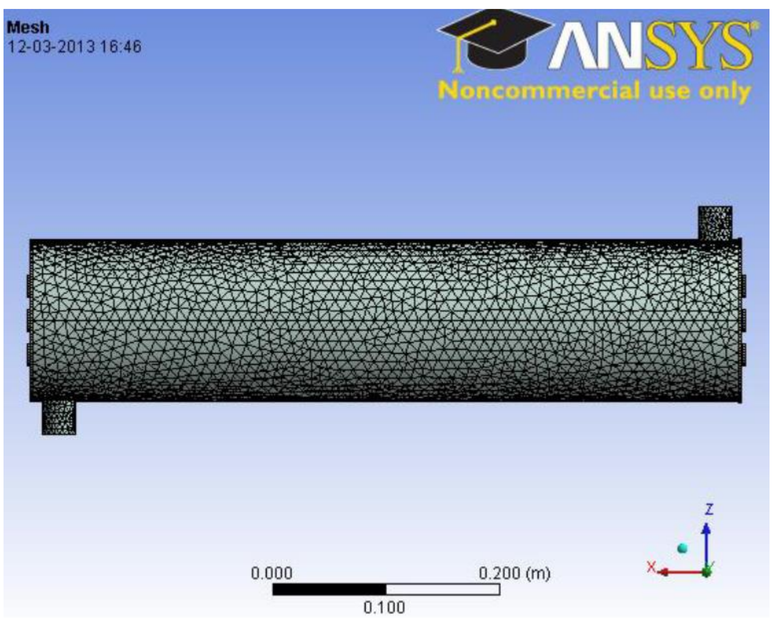


Fig. Complete model of shell and tube heat exchanger



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 cupper, 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 , 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.

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 ,Yvelocity, 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.00 Baffle inclination For Zero degree baffle inclination solution was converged at 170th iteration. The following figure shows the residual plot for the above iterations:

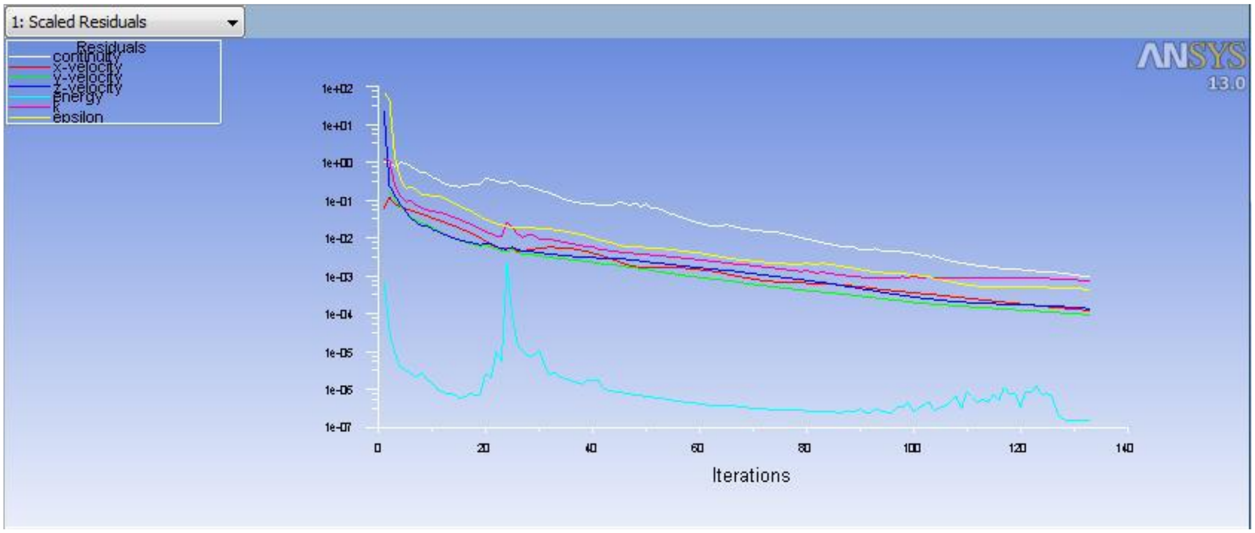


Fig. For Conversion 00 Baffle inclination after 170th iteration 100 Baffle inclination: Simulation of 100 Baffle inclination is converged at 133th iteration. The following figure shows the residual plot:

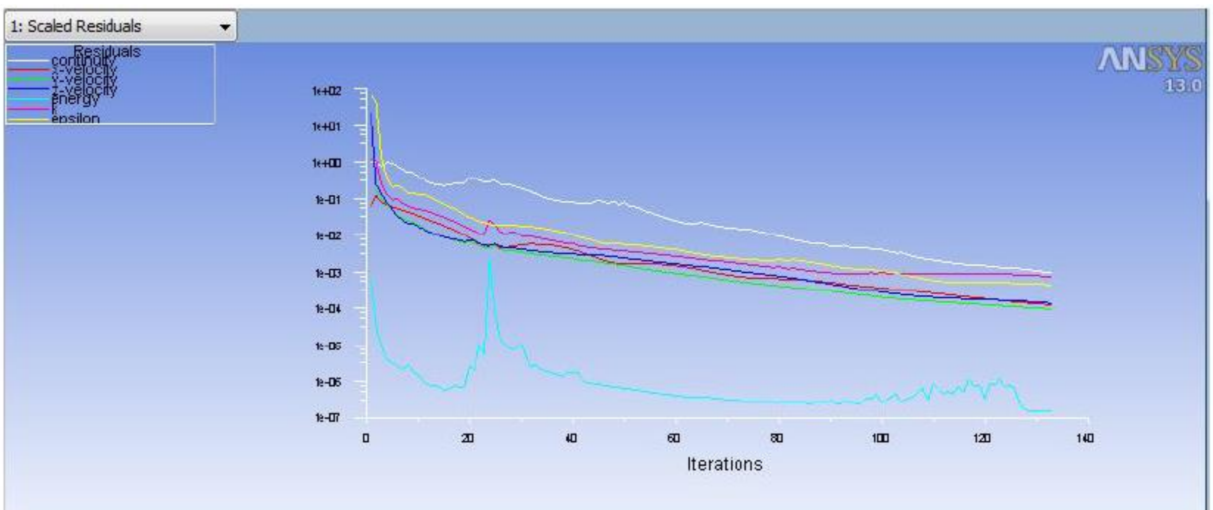
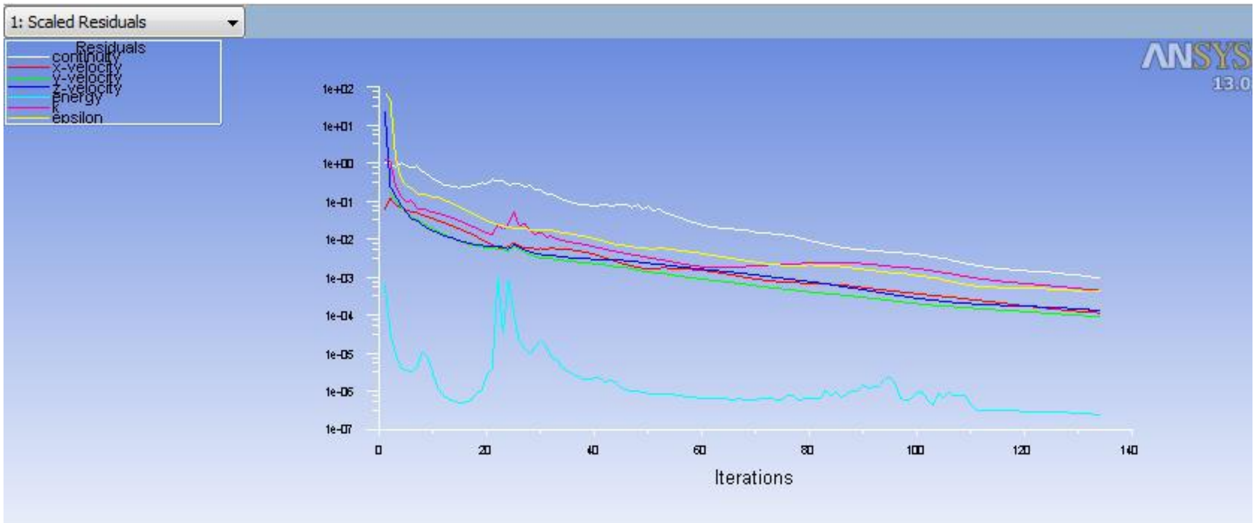


Figure. Converge simulation of 100 baffle inclination at 133th iteration.200 Baffle inclination: Simulation of 200 baffle inclination is converged at 138th iteration. The following figure shows the residual plot:



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 00, 100 , 200 .

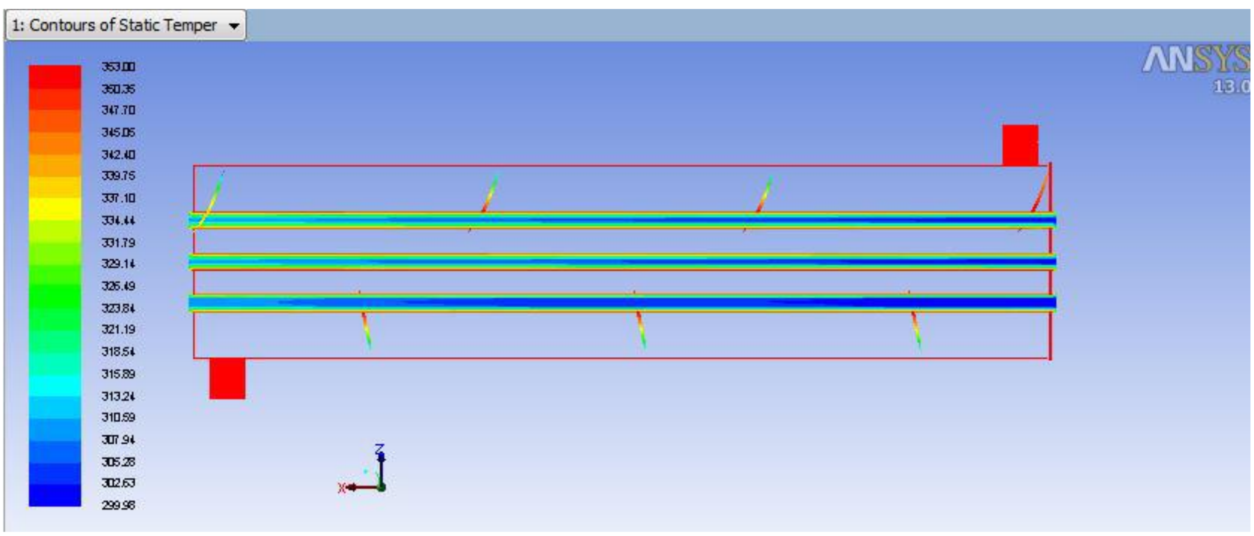


Figure. Temperature Distribution across the tube and shell .

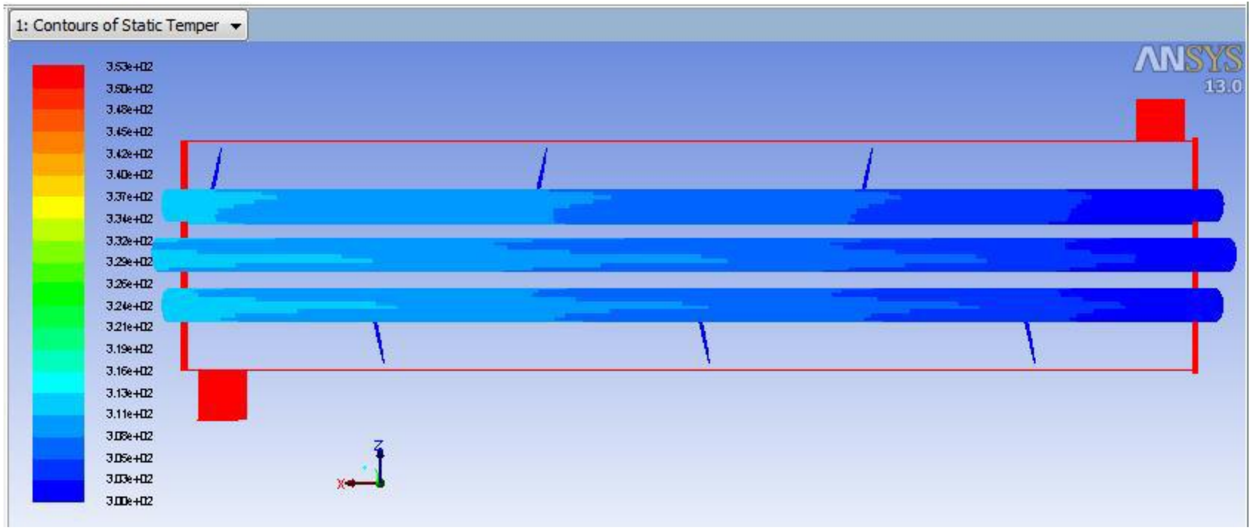


Figure Temperature Distribution for 100 baffle inclination

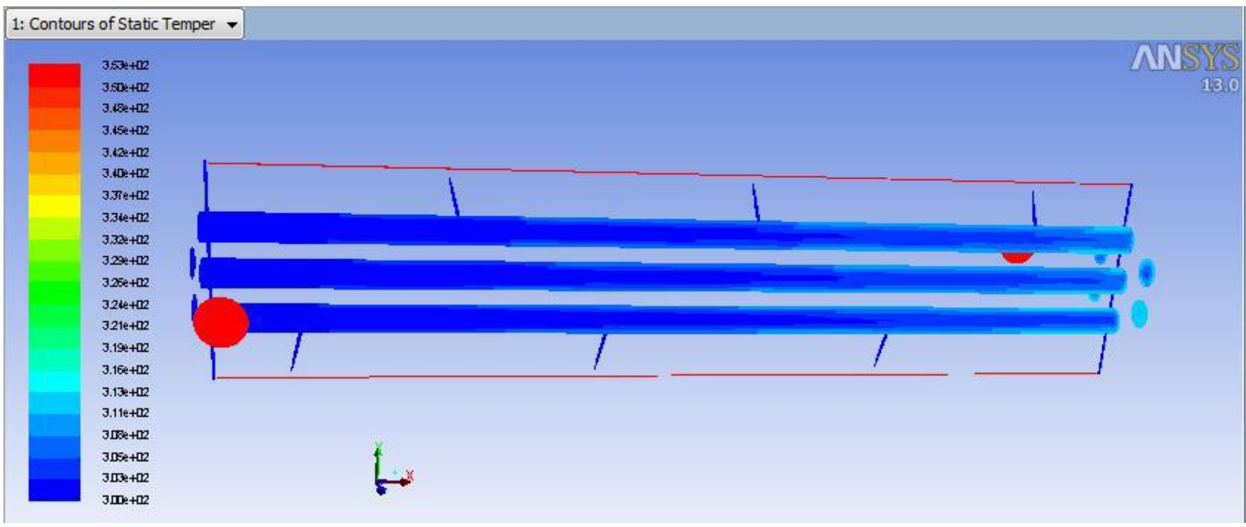


Figure Temperature Distribution of 200 baffle inclination

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

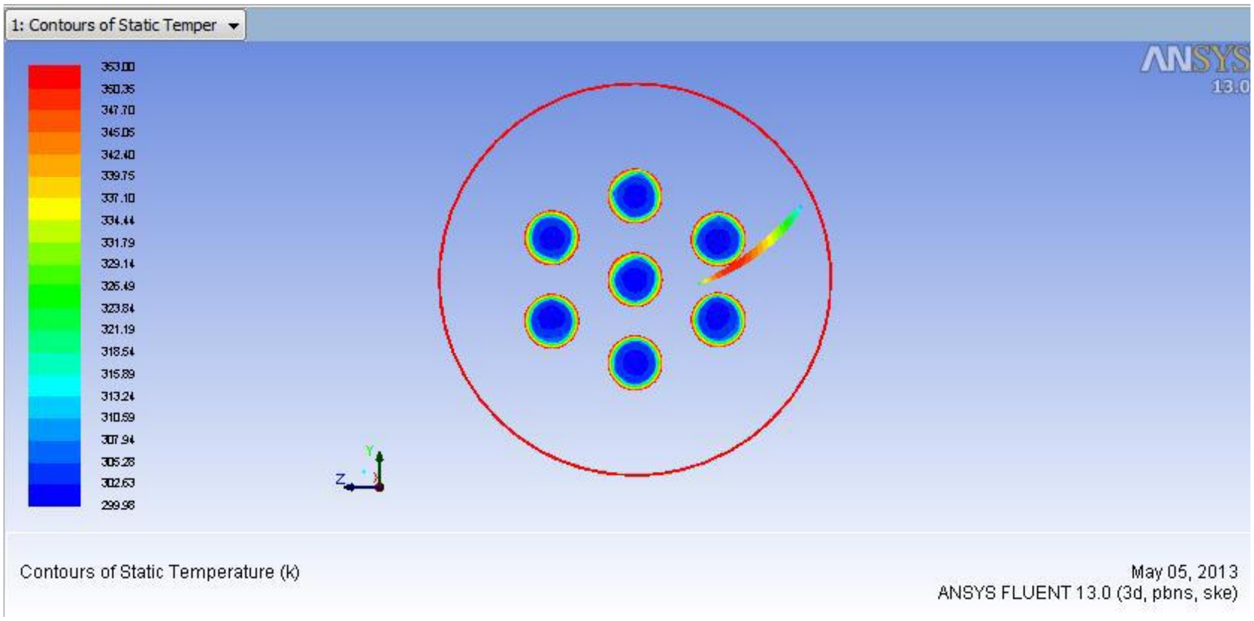


Figure Temperature Distribution across Tube outlet in 00 baffle inclination

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 (12) (13) (14) is the velocity profile of Shell and Tube Heat exchanger at different Baffle inclination. It should be kept in mind that the heat exchanger is modeled 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.

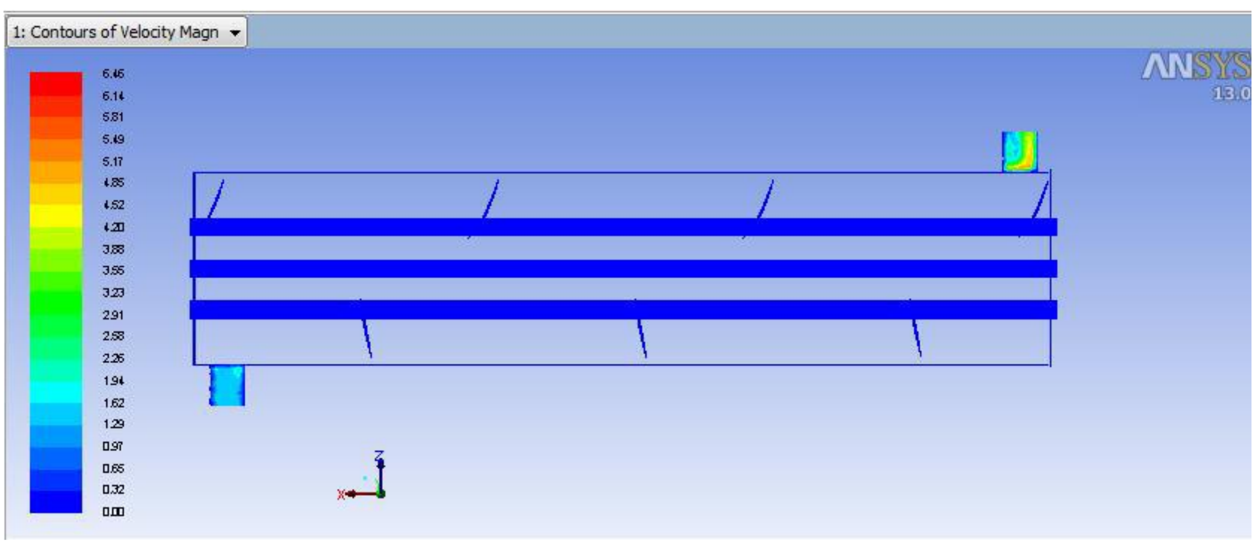


Figure Velocity profile across the shell at 0 0 baffle inclination.

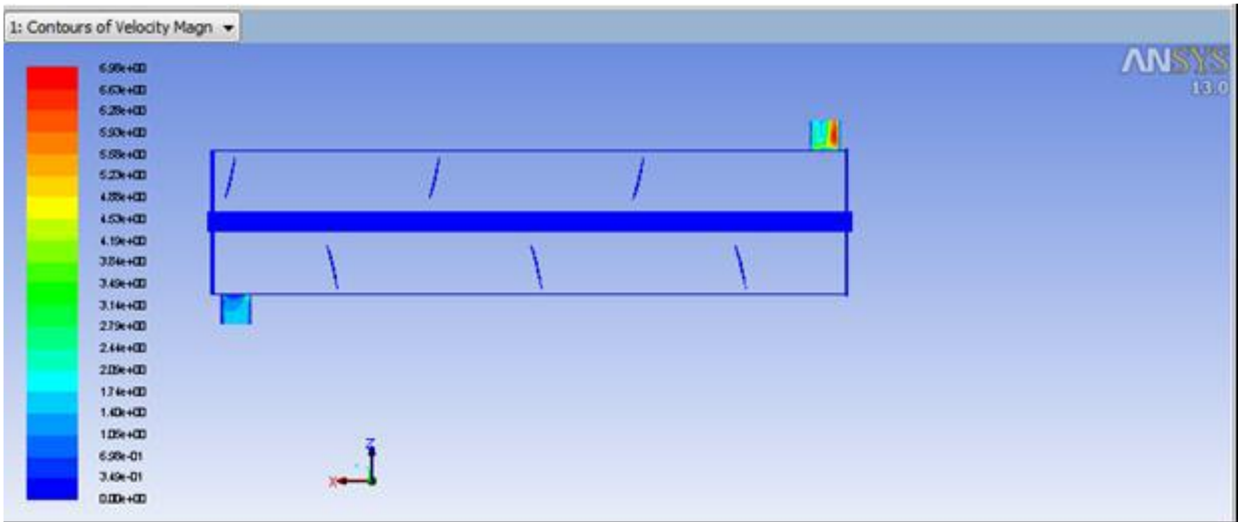


Figure Velocity profile across the shell at 100 baffle inclination.

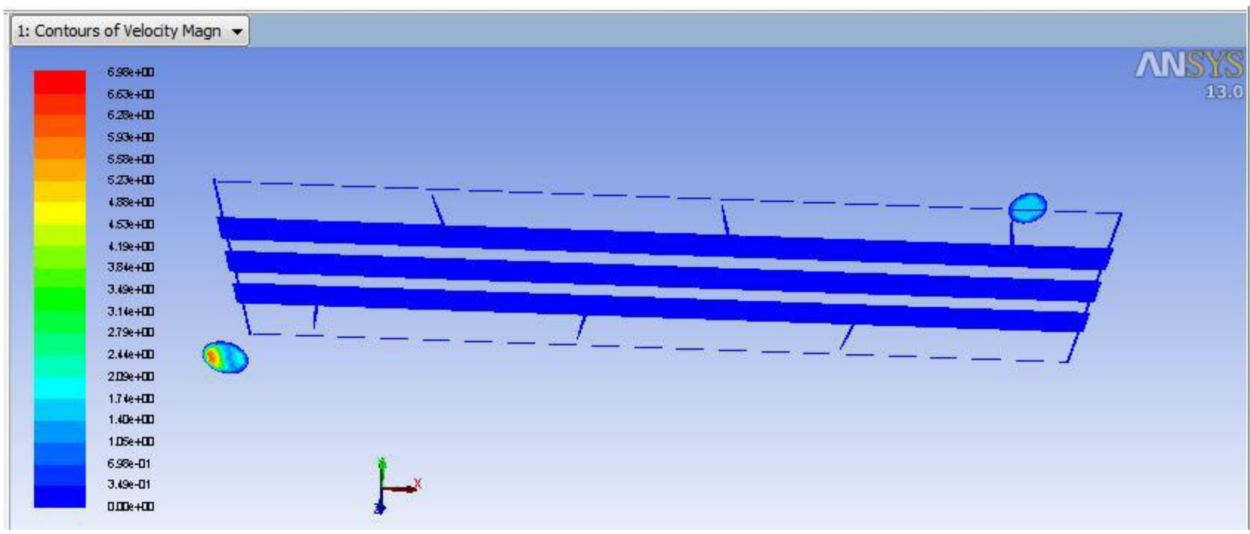
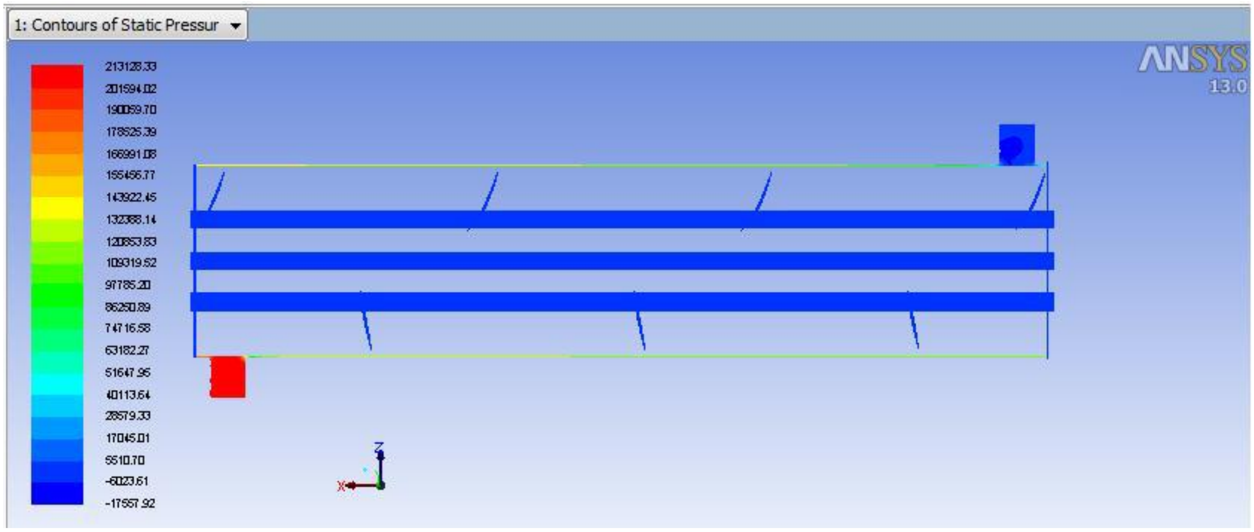


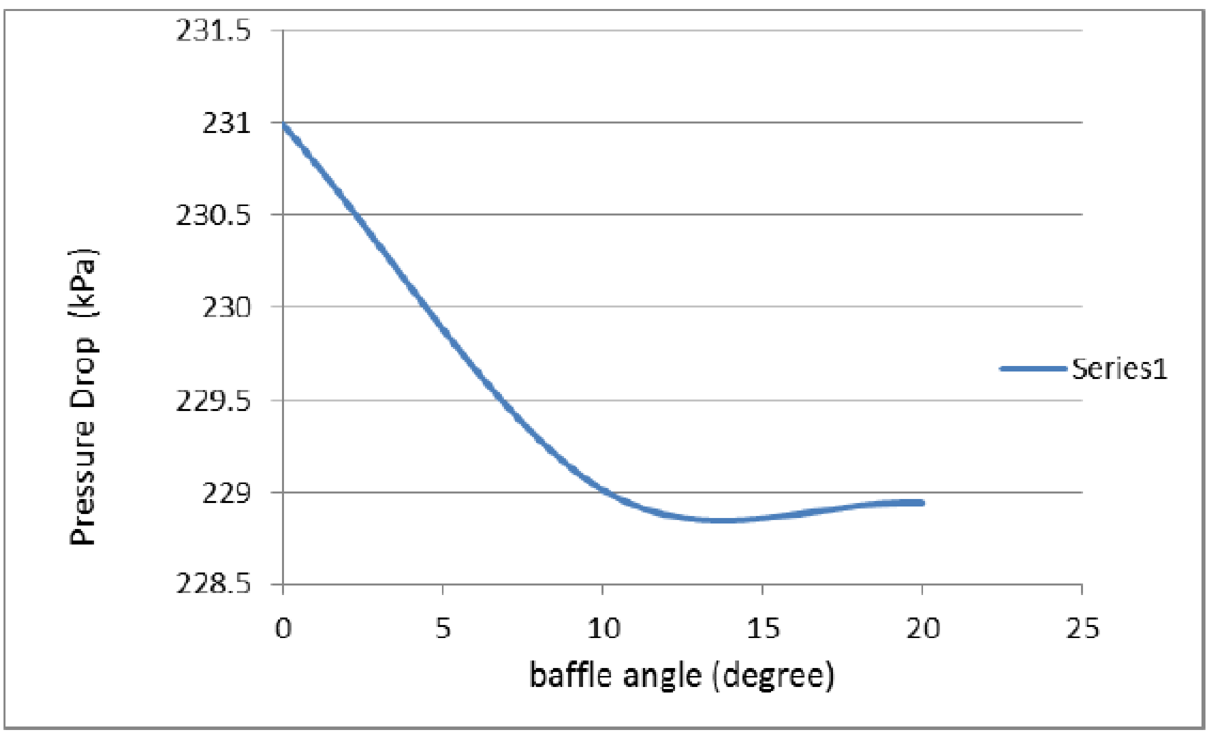
Figure Velocity profile across the shell at 200 baffle inclination.

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 vary largely from inlet to outlet. The contours of static pressure is shown in all the figure to give a detail idea.



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



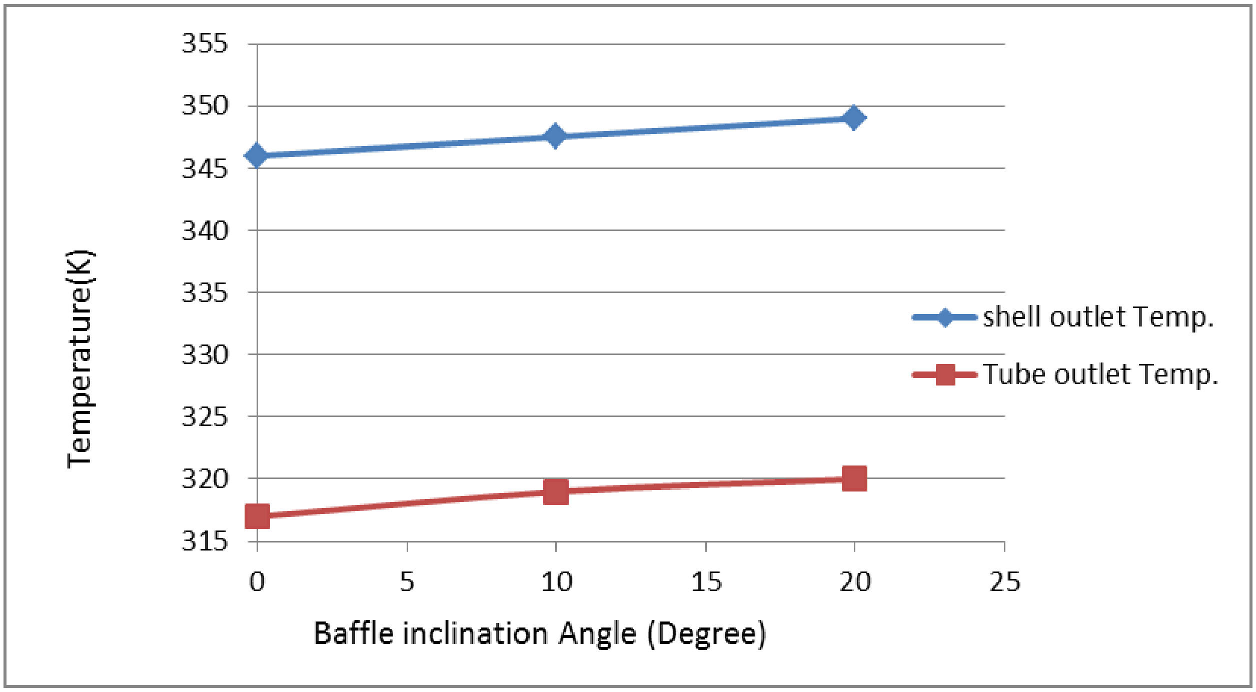


Figure Plot of Baffle angle vs Pressure Drop

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.

Conclusion

The heat move and stream dissemination is talked about in detail and proposed model is looked at With expanding perplex tendency point. The model predicts the heat move and weight drop with a normal blunder of 20%. In this manner the model can be improved. The presumption functioned admirably right now coinciding expect the outlet and bay district where fast blending and change in stream course happens. In this way improvement is normal if the helical puzzle utilized in the model ought to have total contact with the outside of the shell, it will help in additional choppiness across shell side and the heat move rate will increment. On the off chance that distinctive stream rate is taken, it may be help to improve heat move and to show signs of improvement temperature contrast among channel and outlet. In addition the model has given the dependable outcomes by considering the standard k-e and standard divider work model, yet this model over predicts the choppiness in districts with huge ordinary strain. Along these lines this model can likewise be improved by utilizing Nusselt number and Reynolds stress model, however with higher computational hypothesis. Moreover the upgrade divider work are not use right now, they can be exceptionally valuable. The heat move rate is poor on the grounds that the majority of the liquid goes without the collaboration with confuses. In this manner the configuration can be changed for better heat move in two different ways either the diminishing the shell distance across, so it will be an appropriate contact with the helical puzzle or by expanding the perplex so that perplexes will be appropriate contact with the shell. It is on the grounds that the heat move region isn't used productively. In this way the plan can additionally be improved by making cross-stream areas in such a way, that stream doesn't stay corresponding to the cylinders. It will permit the external shell liquid to have contact with the inward shell liquid, consequently heat move rate will increment.

References

Diaper, A.D. and Hesler, L.E. Cross flow Pressure Drop and Flow Distributions within a Tube Bundle Using Computational Fluid Dynamic, Proc. Heat Transfer Conf., Israel, (1990), pp. 235-240.

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.

Gaddis, D. Standards of the Tubular Exchanger Manufacturers Association, TEMA Inc, 9th ed., Tarrytown, N. Y., USA, (2007).

Haseler, L.E., Wadeker, V.V. and Clarke, R.H. Flow Distribution Effect in a Plate and Frame Heat Exchanger, IChemE Symposium Series, No. 129, (1992), pp. 361-367.

Khairun, H. O. CFD simulation of heat transfer in shell and tube heat exchanger, Bachelor in chemical Engineering (Gas Technology) Thesis, (2009).

Kral, D., Stehlik, P., Ploeg V.D.H.J. and Master, B.I. Helical baffles shell-and tube heat exchangers, 1: Experimental verification, Heat Transfer Engg. 17(1), (1996) pp. 93–101.

Li, H. and Kottek .V. Effect of baffle spacing on pressure drop and local heat transfer in shell and tube heat exchangers for staggered tube arrangement, Int. J. Heat Mass Transfer, 41:10, (1998), pp. 1303–1311.

Lutcha, J. and Nemcansky, J. Performance improvement of tubular heat exchangers by helical baffles, Chemical Engineering Research and Design, 68, (1990) pp.263-270.

Stehlík, P., Nemcansky, J., Kral, D. and Swanson, L.W. Comparison of correction factors for shell-and-tube heat exchangers with segmental or helical baffles, Heat Transfer Engineering, 15(1), (1994) pp. 55-65.

Sunil K.S. and Pancha, M.H. Comparative thermal performance of shell and tube heat Exchanger with continuous helical baffle using different angles, International Journal of Engineering Research and Applications, 2:4, (2012).

Temam, R. (Ed.) Claude-Louis Navier and Gabriel Stokes Navier-Stokes Equations: Theory and Numerical Analysis, Institut for Scientific Computing and Applied Mathematics, Indiana University, USA (1977)

Thirumarimurugan, M., Kannadasan, T. and Ramasamy, E. Performance Analysis of Shell and Tube Heat Exchanger Using Miscible System, American Journal of Applied Sciences, 5, (2008), pp. 548-552. 34

Dedication

Not mentioned.

Conflicts of Interest

There are no conflicts to declare.