https://doi.org/10.56452/2022-12-022

ISSN: 0974-5823 Vol. 7 No. 07 December, 2022

International Journal of Mechanical Engineering

Design and Analysis of Shell and Tube Type of Heat Exchanger

Miss. Manasi B. Gangurde
Dept. of Mechanical Engineering, SRES's Sanjivani College of Engineering, Kopargoan, Maharashtra, India
Email: manasigangurde123@gmail.com.

Prof. Santosh. V. Bhaskar
Professor, Dept. of Mechanical Engineering, SRES's Sanjivani College of Engineering, Kopargoan, Maharashtra, India
Email:bhaskarsantoshmech@sanjivani.org.in.

Prof. Bhaskar Dhiraj Popatrao
Associate Professor, Dept. of Mechanical Engineering, SRES's Sanjivani College of Engineering, Kopargoan, Maharashtra, India
Email: bhaskardhirajmech@sanjivani.org.in

ABSTRACT

In present day, shell and tube heat exchanger is the most common type heat exchanger widely use in oil refinery and other large chemical process, because it suits high pressure application. The process of solving simulation consists of modeling and meshing the basic geometry of shell and tube heat exchanger using CFD package ANSYS 13.0. The objective of the project is design of shell and tube heat exchanger with helical baffle and study the flow and temperature field inside the shell using ANSYS software tools. The heat exchanger contains 7 tubes and 600 mm length shell diameter 90 mm. The helix angle of helical baffle was varied from 0^0 to 20^0. Simulation predicts how the pressure varies in shell due to different helix angle and flow rate. 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.

Keywords: Heat exchanger, Modeling, Analysis, Maximum shear stress, Total Deformation, Design.

I. 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 [1, 2]. 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 morebetter 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. 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 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.

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.

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(1). 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 then 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.

II. LITERATUREREVIEW

In 2018 they presented evaluation of shell and tube heat exchanger, thermal performance and pressure drop are considered as major factors. Both, thermal performance and pressure drop are dependent on the path of fluid flow and types of baffles in different orientations respectively. Increasing the complexity of baffles enhances heat transfer which also results in higher pressure drop which means higher pumping power is required. This reduces the system efficiency. This paper presents the numerical simulations carried out on different baffles i.e. single segmental, double segmental and helical baffles. This shows the effect of baffles on pressure drop in shell and tube heat exchanger. Single segmental baffles show the formation of dead zones where heat transfer cannot take place effectively. Double segmental baffles reduce the vibrational damage as compared to single segmental baffles. The use of helical baffles shows a decrease in pressure drop due to the elimination of dead zones. The less dead zones result in better heat transfer. The lower pressure drop results in lower pumping power, which in turn increases the overall system efficiency. The comparative results show that helical baffles are more advantageous than other two baffles.

In 2018 there are presented work related to The specification design of this heat exchanger has been taken from a real project executed by Clough. The process data given matches there application exactly and was used to design the heat exchanger. The Regeneration Gas Heater was manufactured in Malaysia, and was shipped to a fabrication yard in Thailand for installation with in apre-assemble d module.

This paper illustrates fully the thermal and mechanical design for this heat exchanger. Here, there design takes place to improve the design by reselect different parameters which can enhance the heat transfer through the exchanger. As well as, new correlation is developed to predict Nusselt number for tube side which reduces the error percent with Kern's method from 15.25% to 12.64% based on simulation data. Where, the proposed and Kern's correlations compared against experimental data to show that the proposed correlation is quite accurate. After many iterations, the new design suggested that four tube spasses need to be used with 1.38 m tube length. In addition, the tube arrangement chosento be square type with 0.9 m shell diameter. With the separameters, the exchanger achieves high enough heat transfer coefficient and the pressure drop with in specification

In 2019 they performed the the Segmental baffles are used in the conventional shell and tube heat exchanger (STHX). Helical baffles can be used in place of segmental baffles, which increase the heat transfer efficiency and lessen the pressure losses. To avoid flow induced vibrations and eliminate stagnant recirculation zones, a new type of clamping anti-vibration baffle is proposed. Three dimensional CFD simulations, have been performed on ANSYS FLUENT to study and compare the pressure drop and heat transfer coefficient between the newly designed clamping anti-vibration baffles with square twisted tubes, helical baffles with cylindrical tubes and conventional segmental baffle with cylindrical tubes. For the numerical comparison whole heat exchangers are modeled, the numerical model shows the thermo- hydraulic performance with good accuracy by comparing it with Kern and Esso methods for single segmental baffles. It is then used to compare the performance of the same heat exchanger with clamping anti-vibration baffles with square twisted tubes (CBSTT-STHX) and helical baffles with cylindrical tubes (HBCT-STHX). CBSTT-STHX has a higher heat transfer coefficient than segmental baffles with cylindrical tubes (SGCT-STHX) and less pressure drop than both helical baffle with cylindrical tube (HBCT-STHX) and segmental baffle with cylindrical tubes (SGCT-STHX), while its comprehensive performance is higher than SGCT-STHX and slightly less than HBCT- STHX.structural analysis of steering knuckle for weight reduction. The topology optimization is used for 11% weight reduction. HYPERMESH software is used for finite element modeling. The areas for redesign are located by obstructsoftware.

III. METHODOLOGY

The methodology to be worked out to achieve the above mentioned objectives is as follows:

Modeling

To prepare Models of heat exchanger using modeling software that is solid edge .

FEA Analysis

Modeling will be continued with structural analysis software's. In this analysis pre-processing is done using the applying material property, load and boundary condition to heat exchanger. Using ANSYS software, we will get stress and displacement results.and simulation done with variation temperature ,pressure & velocity.

Result Evaluation

We are validating the results to comparing values we get form software with the Analytical or mathematical results also by Using differant temperature ,pressure & velocity for heat exchanger helical baffle and Comparing results.

Designing

Design heat exchanger as per ASME code and calculation run in PV ELIET SOFTWARE.

IV. MODELING ANDANALYSIS

Solid edge is the world's engineering and design leading software for product 3D CAD design. It is used to design, simulate, analyze, and manufacture products in a variety of industries including aerospace, automotive, consumer goods, and industrial machinery etc. The heat exchanger are modeled by using modeling software solid edge and it is the most powerful and widely used CAD software.

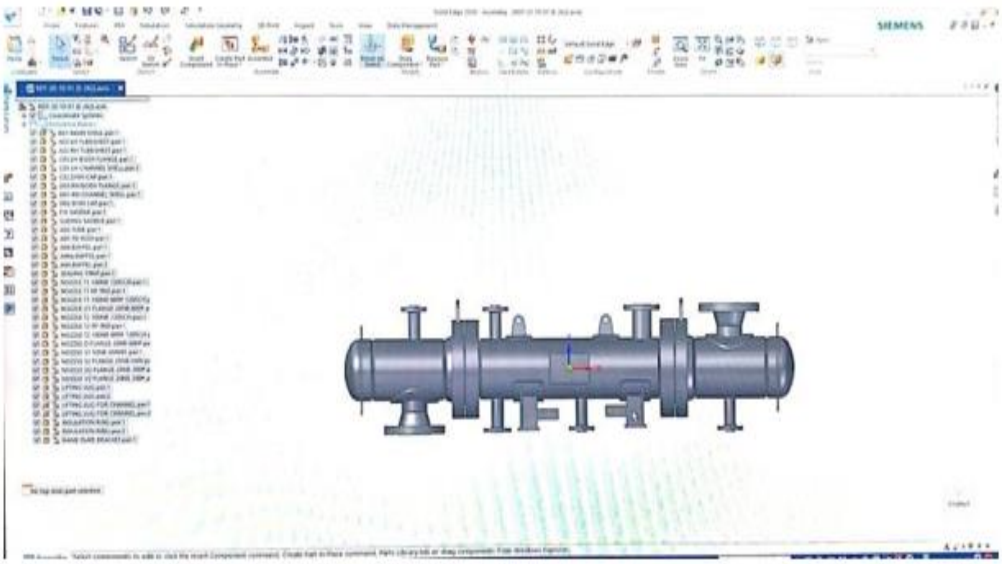


Figure 1: heat exchanger with helical baffle

V. COMPUTATIONAL MODEL FOR HEATEXCHANGER

Computational Model: The computational model of an experimental tested Shell and Tube Heat Exchanger (STHX) with 10 helix angle, and the geometry parameters are listed in Table 1.As can be seen from Fig 2 ,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(3).

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 steadystate
3. The leak flows between tube and baffle and that between baffles and shell areneglected
4. The natural convection induced by the fluid density variation isneglected
5. The tube wall temperature kept constant in the whole shellside
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 , He described the motion of fluid substances. Its 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, thegeneral form of the equations of fluid motion is :-

$$\partial_x u + \partial_y v = 0, \quad (1)$$

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

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

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

This Navier Stokes Equation slove in every mess shell and the simulation shows the result.

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

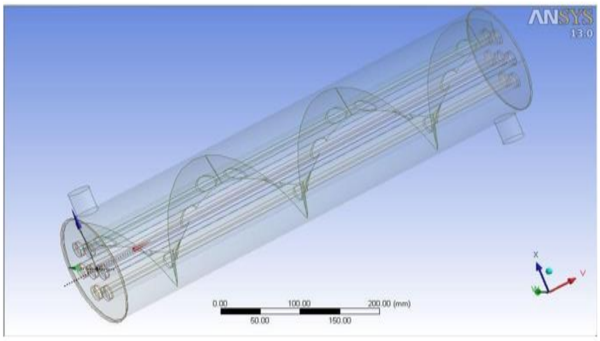


Fig 2: Isometric view of arrangement of baffles and tubes of shell and tube heat exchanger with baffle inclination.

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
Baffle inclination angle, θ	0 to 40^0

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(3). 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 bafflesrespectively.

https://doi.org/10.56452/2022-12-022

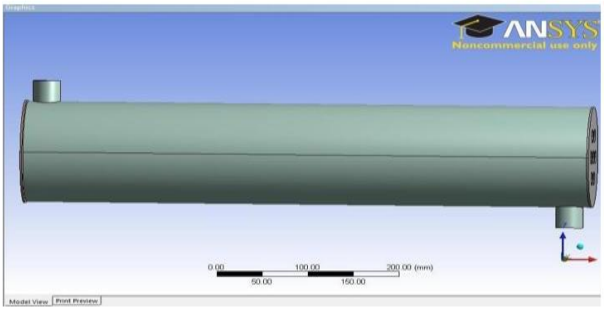


Fig 3. complete model of shell and tube heat exchanger

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 finelymeshed.

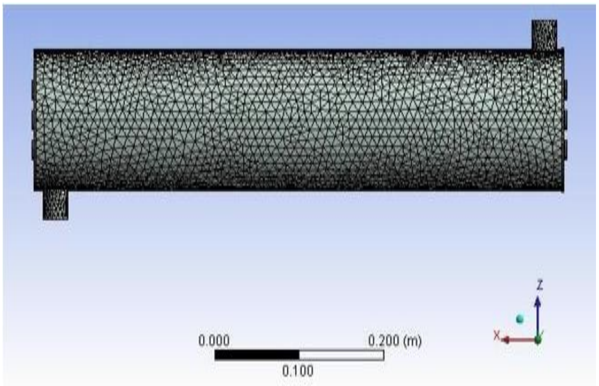


Fig 4: Meshing diagram of shell and tube heat exchanger

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 $1m^2$,Density 998 kg/m^3,enthalpy 229485 j/kg , length 1m , temperature 353k, Velocity 1.44085 m/s, Ration of specific heat 1.4 wasconsidered.

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 300ktemperature.

VI. RESULT

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(4). 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.

0^0Baffle inclination

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

https://doi.org/10.56452/2022-12-022

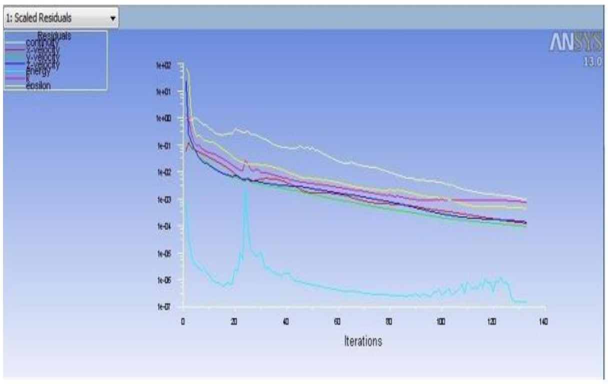


Figure 5: For Conversion 0^0Baffle inclination after 170^{th}iteration

10^0Baffle inclination:

Simulation of 10^0Baffle inclination is converged at 133th iteration. The following figure shows the residual plot:

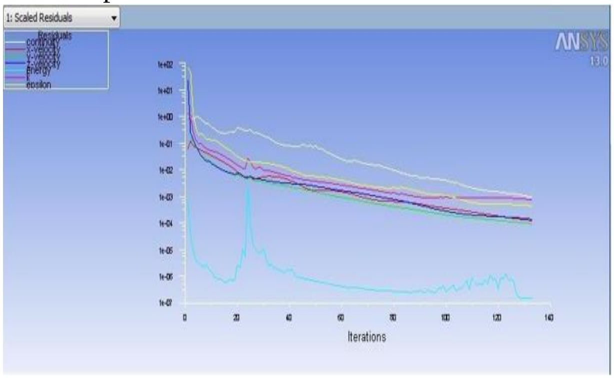


Figure 6: Converge simulation of 10^0baffle inclination at 133th iteration. 20^0Baffle inclination

20^0Baffle inclination:

Simulation of 20^0baffle inclination is converged at 138^{th}iteration. The following figure shows the residual plot:

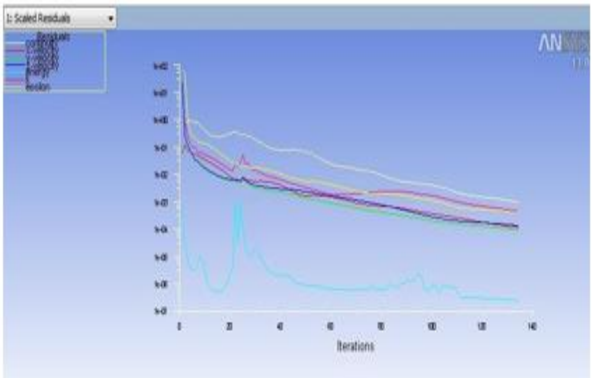


Figure 7: Convergence of 20^0baffle inclination at 138^{th}iteration

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.

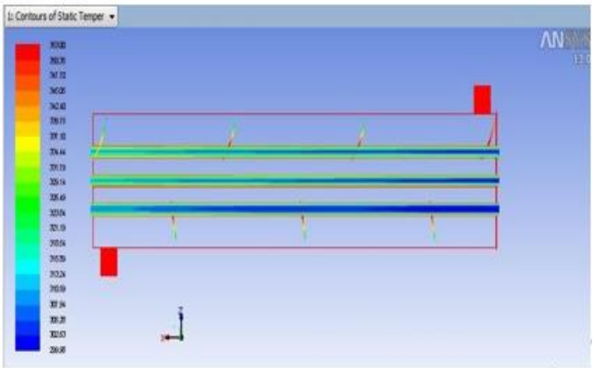


Figure 8 : Temperature Distribution across the tube and shell .

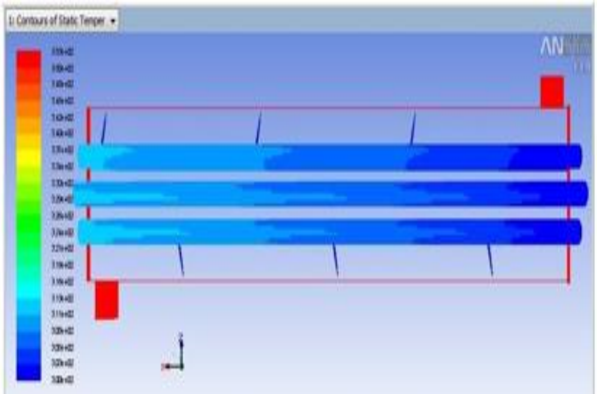


Figure 9 : Temperature Distribution for 10^0baffle inclination

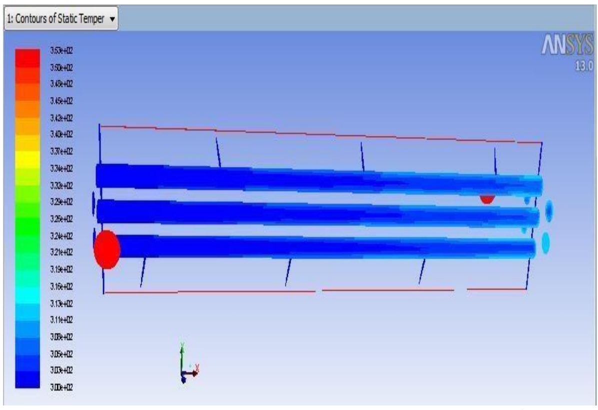


Figure 10 : Temperature Distribution of 20^0baffle inclination

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

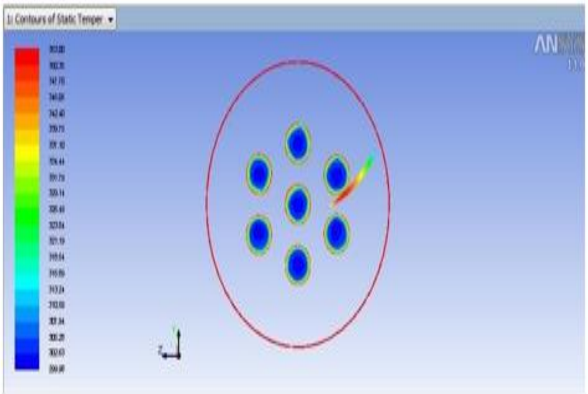


Figure 11 : Temperature Distribution across Tube outlet in 0^0baffle inclination

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(4). 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 shellregion.

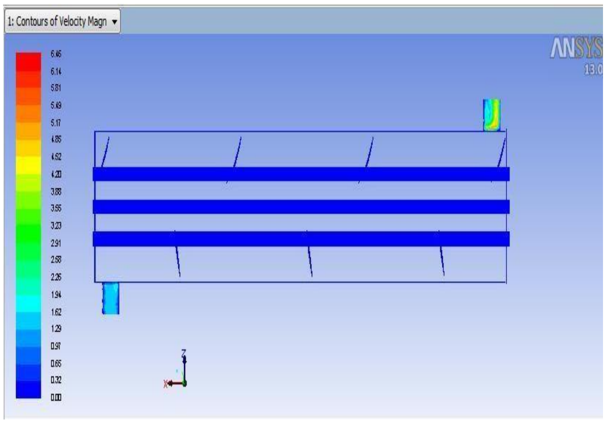


Figure 12 : Velocity profile across the shell at 0^0baffle inclination.

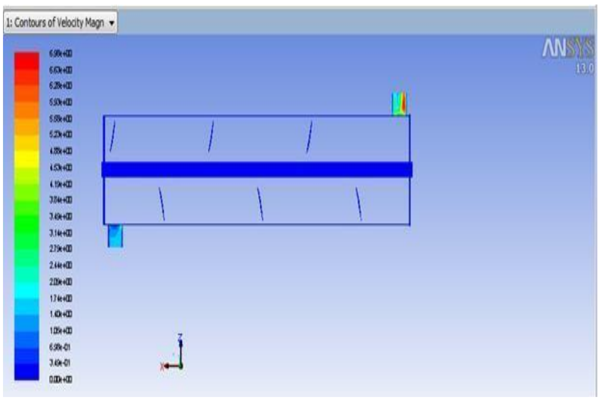


Figure 13: Velocity profile across the shell at 10^0baffleinclination.

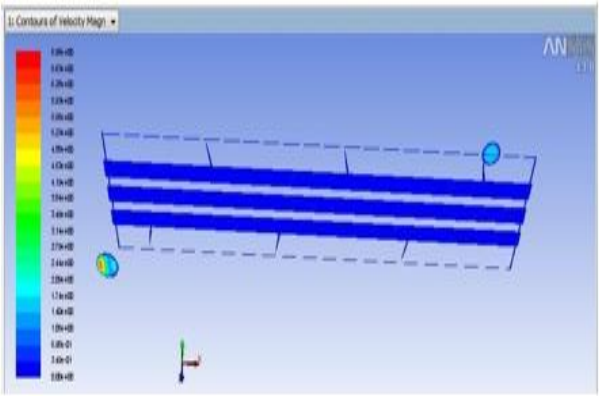


Figure 14: Velocity profile across the shell at 20^0baffleinclination.

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.

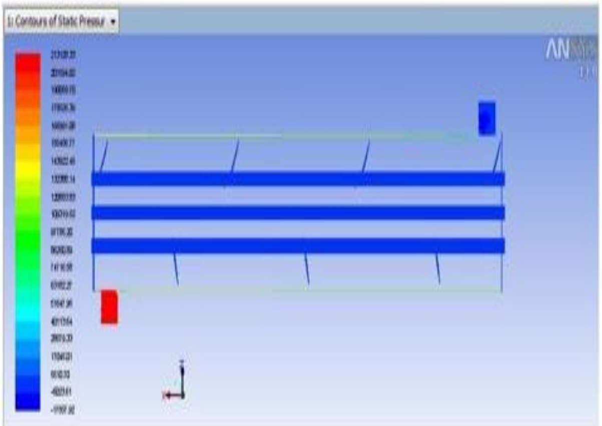
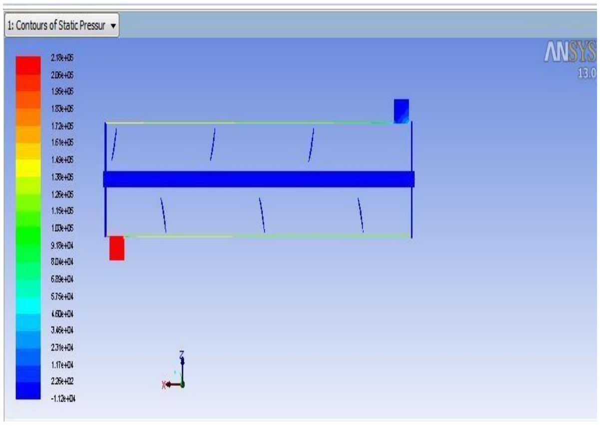


Figure 15 Pressure Distribution across the shell at 0^0baffle inclination



https://doi.org/10.56452/2022-12-022

Figure 16: Pressure Distribution across the shell at 10^0baffleinclination

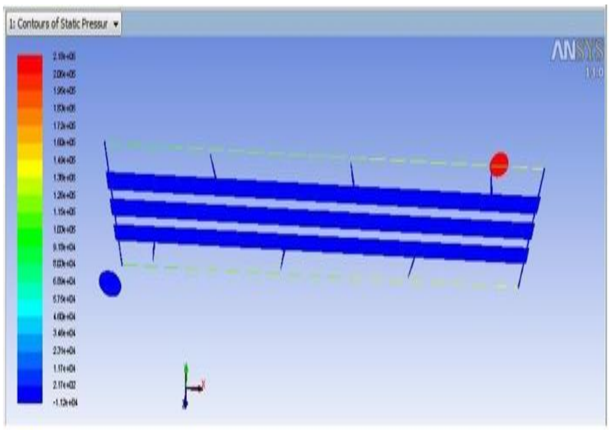


Figure 17: Pressure Distribution across the shell at 20^0baffleinclination.

Table 2: for 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

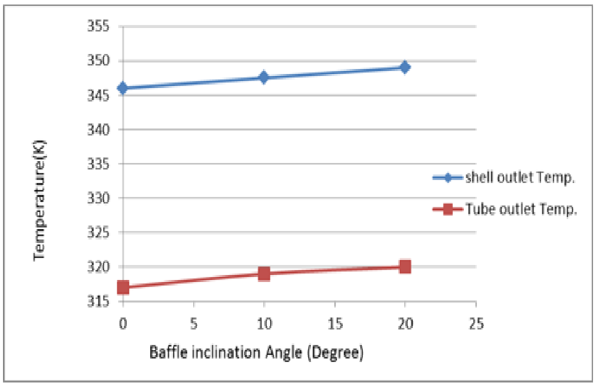


Figure 18: Plot of Baffle inclination angle vs Outlet Temperature of shell and tube side

It has been found that there is much effect of outlet temperature of shell side with increasing the baffle inclination angle from 0^0to20^0.

Table 3: for the Pressure Drop inside Shell

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

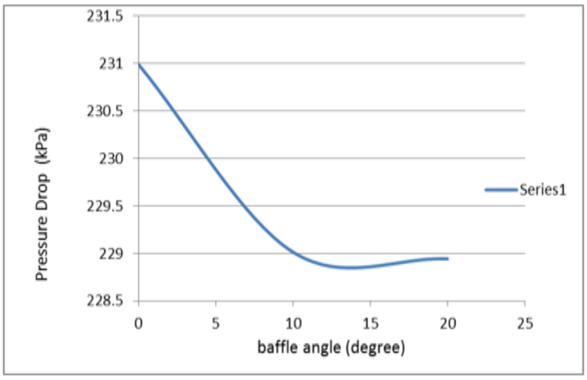


Figure 19: Plot of Baffle angle vs Pressure Drop

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: for Velocity inside Shell

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

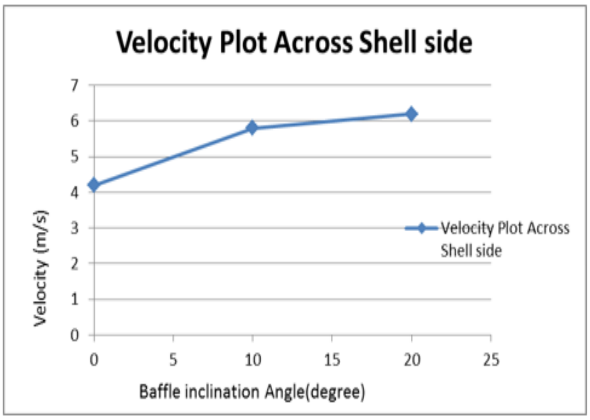


Figure 20 : Plot of Velocity profile inside shell

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

Heat Transfer Table 5 : for Rate Across Tube side

Baffle Inclination Angle (Degree)	Heat Transfer Rate Across Tube side (w/m^2)
0	3557.7
10	3972.9
20	4182

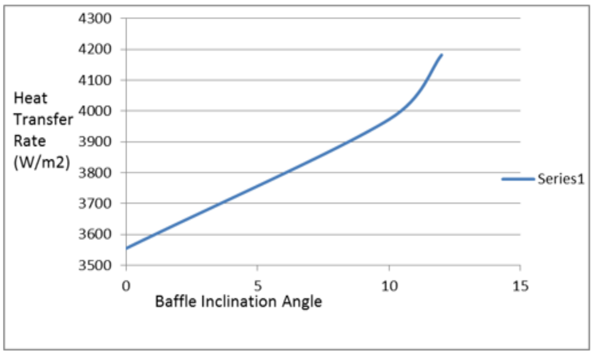


Figure 21: Heat Transfer Rate Along Tube side

The heat transfer rate is calculated from above formulae from which heat transfer rate is calculated across shell 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 in figure 20.

Table 6: for the 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)
00	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

(i) The shell side of a small shell-and-tube heat exchanger is modeled with sufficient detail to resolve the flow and temperaturefields.

(ii) The pressure drop decreases with increase in baffleinclination.

(iii) The heat transfer rate is very slow in this model so that it affect the outlet temperature of the shell and tubeside.

VII. 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.

VIII. REFERENCES

1. Pranita Bichkar, Ojas Dandgaval, Pranita Dalvi, Rhushabh Godase and Tapobrata Dey "Study of Shell and Tube Heat Exchanger with the Effect of Types of Bafflel" Procedia Manufacturing 20 (2018)195–200
2. AmmarAliAbd,SamahZakiNaji,"Analysisstud yofshellandtubeheatexchangerforcloughcompa ny with reselect different parameters to improve the design" case studies in therm alengg 10 (2017)455-467
3. Ammar AliAbd, Mohammed Qasim Kareem, Samah ZakiNaji, "Perform anceanaly sisof shell and tube heat exchanger Parametric study "case studies in therm alengg12(2018)563-568
4. S.M.A. Naqvi, K.E. Elfeky, Yiping Cao, Qiuwang Wang, "Numerical analysis on performances of shell side in segmental baffles, helical baffles and novel clamping anti-vibration baffles with square twisted tubes shell and tube heat exchangers" Energy Procedia 158 (2019)5770–5775

AUTHOR'S PROFILE

Mtech Design, BE Mechanical, Department of Mechanical Engineering, Sanjivani College of Engineering an Autonomous Institution Kopargaon Affiliated to Savitribai Phule Pune University Maharashtra, India,
manasigangurde123@gmail.com