A PROJECT REPORT

ON

“CFD ANALYSIS ON A HELICAL BAFFLES OF SHELL AND TUBE TYPE HEAT EXCHANGER”

Submitted to

UNIVERSITY OF MUMBAI

In Partial Fulfilment of the Requirement for the Award of

BACHELOR’S DEGREE IN
MECHANICAL ENGINEERING

BY

RUMANE ABDULLAH	**15ME36**
SHETTY SANJOTH	**15ME55**
SINGH VISHAL	**15ME58**
GUPTA VIPIN	**16DME138**

UNDER THE GUIDANCE OF
PROF. RAHUL THAVAI

DEPARTMENT OF MECHANICAL ENGINEERING
Anjuman-I-Islam’s Kalsekar Technical Campus
School Of Engineering & Technology
Plot No. 2 3, Sector - 16, Near Thana Naka,
Khandagaon, New Panvel - 410206
2018-2019

AFFILIATED TO

UNIVERSITY OF MUMBAI

A PROJECT II REPORT
ON

"CFD ANALYSIS ON HELICAL BAFFLES ON SHELL AND TUBE TYPE HEAT EXCHANGER"

Submitted to
UNIVERSITY OF MUMBAI

In Partial Fulfilment of the Requirement for the Award of

**BACHELOR'S DEGREE IN
MECHANICAL ENGINEERING**

BY

RUMANE ABDULLAH	15ME36
SHETTY SANJOTH	15ME55
SINGH VISHAL	15ME58
GUPTA VIPIN	16DME138

UNDER THE GUIDANCE OF
PROF. RAHUL THAVAI

DEPARTMENT OF MECHANICAL ENGINEERING
Anjuman-I-Islam's Kalsekar Technical Campus
School Of Engineering & Technology
Plot No. 2 3, Sector - 16, Near Thana Naka,
Khandagaon, New Panvel - 410206

2018-2019
AFFILIATED TO

UNIVERSITY OF MUMBAI

Anjuman-I-Islam's Kalsekar Technical Campus

Department of Mechanical Engineering

SCHOOL OF ENGINEERING & TECHNOLOGY

Plot No. 2 3, Sector - 16, Near Thana Naka,

Khandagaon, New Panvel - 410206

CERTIFICATE

This is certify that the project entitled

"CFD ANALYSIS ON HELICAL BAFFLES OF SHELL AND TUBE TYPE HEAT EXCHANGER"

submitted by

RUMANE ABDULLAH	15ME36
SHETTY SANJOTH	15ME55
SINGH VISHAL	15ME58
GUPTA VIPIN	16DME138

is a record of bonafide work carried out by them, in the partial fulfilment of the requirement for the award of Degree of Bachelor of Engineering (Mechanical Engineering) at ***Anjuman-I-Islam's Kalsekar Technical Campus, Navi Mumbai*** under the University of MUMBAI. This work is done during year 2018-2019, under our guidance.

Date:30 /04 /2019

(Prof. RAHUL THAVAI)
Project Supervisor

(Prof. RIZWAN SHAIKH)
Project Coordinator

(Prof. ZAKIR ANSARI)
HOD, Mechanical Department

DR. ABDUL RAZAK HONNUTAGI
Director

External Examiner

ACKNOWLEDGEMENT

I would like to take the opportunity to express my sincere thanks to my guide **RAHUL THAVAI**, Assistant Professor, Department of Computer Engineering, AIKTC, School of Engineering, Panvel for his invaluable support and guidance throughout my project research work. Without his kind guidance & support this was not possible.

I am grateful to him for his timely feedback which helped me track and schedule the process effectively. His time, ideas and encouragement that he gave is help me to complete my project efficiently.

We would like to express deepest appreciation towards **DR. ABDUL RAZAK HONNUTAGI**, Director, AIKTC, Navi Mumbai, **Prof. ZAKIR ANSARI**, Head of Department of Mechanical Engineering and **Prof. RIZWAN SHAIKH**, Project Coordinator whose invaluable guidance supported us in completing this project.

At last we must express our sincere heartfelt gratitude to all the staff members of Mechanical Engineering Department who helped me directly or indirectly during this course of work.

RUMANE ABDULLAH
SHETTY SANJOTH
SINGH VISHAL
GUPTA VIPIN

Project Approval for Bachelor of Engineering

This project entitled ***ĊFD ANALYSIS OF HELICAL BAFFLES ON SHELL AND TUBE TYPE HEAT EXCHANGER¨*** by ***Rumane Abdullah, Shetty Sanjoth, Singh Vishal, Gupta Vipin*** is approved for the degree of ***Bachelor of Engineering in Department of Mechanical Engineering.***

Examiners

1. ....
2. ....

Supervisors

1. ....
2. ....

Chairman

....

DECLARATION

I declare that this written submission represents my ideas in my own words and where others ideas or words have been included, I have adequately cited and referenced the original sources. I also declare that I have adhered to all principles of academic honesty and integrity and have not misrepresented or fabricated or falsified any idea/data/fact/source in my submission. I understand that any violation of the above will be cause for disciplinary action by the Institute and can also evoke penal action from the sources which have thus not been properly cited or from whom proper permission has not been taken when needed.

Student Name
Rumane Abdullah
Shetty Sanjoth
Singh Vishal
Gupta Vipin

Roll Number:
15ME36
15ME55
15ME58
16DME138

ABSTRACT

In present day, shell and tube heat exchanger is the most common type heat exchanger widely used in oil refinery and other large chemical process, because it suits high pressure application.

The aim of this work is to design shell and tube type heat exchanger with a continuous helical baffle and comparing with segmental baffle with CFD analysis using ANSYS FLUENT software tools.

The model contains 7 copper tubes each having 20mm external diameter and 17mm internal diameter, length 600mm and inner diameter of steel shell is 90 mm and outer diameter 110mm. 7 tubes are held by a continuous helical aluminium baffle. All the models are design by using AUTODESK INVENTOR software tools.

In this research how the pressure drop and overall heat transfer coefficient varies due to change in the inlet velocity of the shell side has been studied. The flow pattern in the shell side of the shell and tube type heat exchanger with continuous helical baffles are 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: *Continuous helical baffles, Shell and tube heat exchanger, Overall heat transfer Coefficient, Pressure drop, CFD.*

Contents

List of Figures

List of Tables

Chapter 1

INTRODUCTION

A Heat Exchanger may be defined as an equipment which transfers energy from a hot fluid to a cold fluid, either with a maximum or minimum rate within minimum investment and running cost. In this process never two fluids mixed with each other. This device provides a flow of thermal energy between two or more fluids at different temperatures. Shell and tube heat exchangers are most versatile type of heat exchanger, they are used in a wide variety of engineering applications like power generation, waste heat recovery, manufacturing industry, air conditioning, refrigeration, space applications, petrochemical industries etc. This design trend requires heat exchangers which use as little of the process stream momentum as possible in flow through equipment. Precisely to vary the velocity and turbulence on the shell side and therefore the heat transfer coefficient, baffles can be used, in order to induce the fluid on the shell side to travel a tortuous path. The baffles have, in addition, an important task from the mechanical point of view; they support, in fact, the tubes of the bundle, keeping them equally spaced, and preventing the vibrations of the tubes themselves. On the shell side, the conventional segmental baffle exhibits significant pressure differences to produce a sufficiently high heat transfer rate, low efficiency zones, mixed flow, bypass or recirculated currents. Therefore, new types of baffles, such as helical baffles, have been proposed.

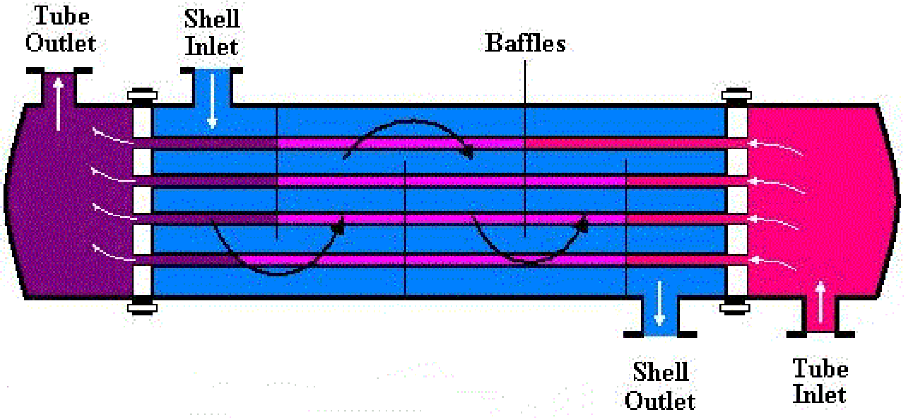

Figure 1.1: Shell and Tube Heat Exchanger

A shell-and-tube heat exchanger with continuous helical baffles, commonly called “helical baffles” heat exchanger, is a viable alternative to the traditional heat exchanger with segmental baffles. It succeeds, in fact, in reconciling an increase in equipment performance with a relatively complex production and simple installation technology. This type of baffle is represented by a continuous helical plates forming a helix in order to impart a helical path to the fluid,on the shell side. “Helical baffles” heat exchangers are generally used in oil and gas plants and refineries in the petrochemical and chemical industries. In the work,a 3D numerical simulation of a real heat exchanger was performed, considering first the type with segmental baffles which was taken from the base reference paper and, later ,the one with continuous helical baffles,through the use of commercial codes, in order to compare the performance of the two types taken into consideration. Moreover, the performance of“helical baffles ”heat exchanger with different shell side inlet velocity were studied. CFD simulations results, finally, were compared with those obtained by the application of some correlations available in the literature.

Heat exchangers are of two types:-

a. Where both media between which heat is exchanged are in direct contact with each other is known as ”Direct contact heat exchanger”,

b. Where both media are separated by a wall through which heat is transferred so that they never mix are known as ”Indirect contact heat exchanger”.

A typical heat exchanger, usually for higher pressure applications upto 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. 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 (CFD) 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.

1.1 OBJECTIVES

A. To study the process in solving simulation consists of modelling and meshing the basic geometry of shell and tube heat exchanger.

B. To design a shell and tube heat exchanger with continuous helical baffle and study the flow and temperature field inside the shell using ANSYS software tools.

Chapter 2

LITERATURE SURVEY

2.1 INTRODUCTION

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, a review of other relevant research studies are made to provide more information in order to understand more on this research.

2.1.1 PURPOSE OF USE OF HELICAL BAFFLE

The subject of baffle in shell and tube heat exchanger (STHE) has a wide variety of processes. A large number of works has been published regarding STHE which depicts various factors affecting the thermal efficiency of the STHE. On the basis of that a brief summary is reviewed as follows:-

Rajiv Mukherjee, [1] explains the basics of exchanger thermal design, covering such topics as: STHE components; classification of STHEs according to construction and according to service; data needed for thermal design; tube side design; shell side design, including tube layout, baffling, and shell side pressure drop; and mean temperature difference. The basic equations for tube side and shell side heat transfer and pressure drop. Correlations for optimal condition are also focused and explained with some tabulated data. This paper gives overall idea to design optimal shell and tube heat exchanger. The optimized thermal design can be done by sophisticated computer software however a good understanding of the underlying principles of exchanger designs needed to use this software effectively.

M. Serna and A. Jimenez, [2] they have presented a compact formulation to relate the shell-side pressure drop with the exchanger area and the film coefficient based on the full Bell–Delaware method. In addition to the derivation of the shell side compact expression, they have developed a compact pressure drop equation for the tube-side stream, which accounts for both straight pressure drops and return losses. They have shown how the compact formulations can be used within an efficient design algorithm. They have found a satisfactory performance of the proposed algorithms over the entire geometry range of single phase, shell and tube heat exchangers.

Lei et al., [3] have showed the effects of baffle inclination angle on flow and heat transfer of a heat exchanger with helical baffles, where the helical baffles are separated into inner and outer parts along the radial direction of the shell. While both the inner and outer helical baffles baffle the flow consistently, smoothly and gently, and direct flow in a helical fashion so as to increase heat transfer rate and decrease pressure drop and impact vibrations, the outer helical baffle becomes easier to manufacture due to its relatively large diameter of inner edge.

Lutcha and Nemcansky, [4] have done experiments to the improvement of tubular heat exchangers with helical baffles for investigation of the flow field patterns generated by various helix angles which is expected to decline pressure at shell side and increase heat transfer process significantly.

Pardeep Kumar et al., [5] experimental investigation has been carried-out to know the thermal performance of Helix exchanger with plain copper tubes or with grooved copper tubes of same size and specification by using co-current flow. During this experimental investigation attempts were made for both exchangers at same operating conditions and it was found that grooved copper tubes helix changer have a better thermal performance as compared to plain copper tubes helix changer at a particular angle, 25.

Sunilkumar Shinde et al., [6] were done analyses the conventional segmental baffle heat exchanger by using the Kern method with varied shell side flow rates. They evaluated form their results high heat transfer Co-efficient and lower pressure drop are more effectively obtained in a helix changer. The flow pattern in the shell side of the continuous helical baffle heat exchanger is rotational helical due to the geometry of continuous helical baffles results in significant increase in heat transfer coefficient.

Wang , [7] the cases are run for mass flow rates ranging from 1 to 5 kg/s. Also the paper is taken as the base paper.

2.1.2 BAFFLE

Baffle is a device used to put down the flow of a fluid, gas etc. Baffles serve two important functions. They support the tubes during assembly and operation and help prevent vibration from flow induced eddies and direct the shell side fluid back and forth across the tube bundle to provide effective velocity and Heat Transfer rates. The diameter of the baffle must be slightly less than the shell inside diameter to allow assembly, but must be close enough to avoid the significant performance penalty caused by fluid bypass around the baffles. Shell roundness is important to achieve effective sealing against excessive bypass. Baffles can be made from a variety of materials compatible with the shell side fluid. They can be punched or machined. Some baffles are made by a punch which provides a lip around the tube hole to provide more surfaces against the tube and eliminate tube wall cutting from the baffle edge. Baffles may be classified as transverse and longitudinal types. The purpose of longitudinal baffles is to control the overall flow direction of the shell fluid such that a desired overall flow arrangement of the two fluid streams is achieved.

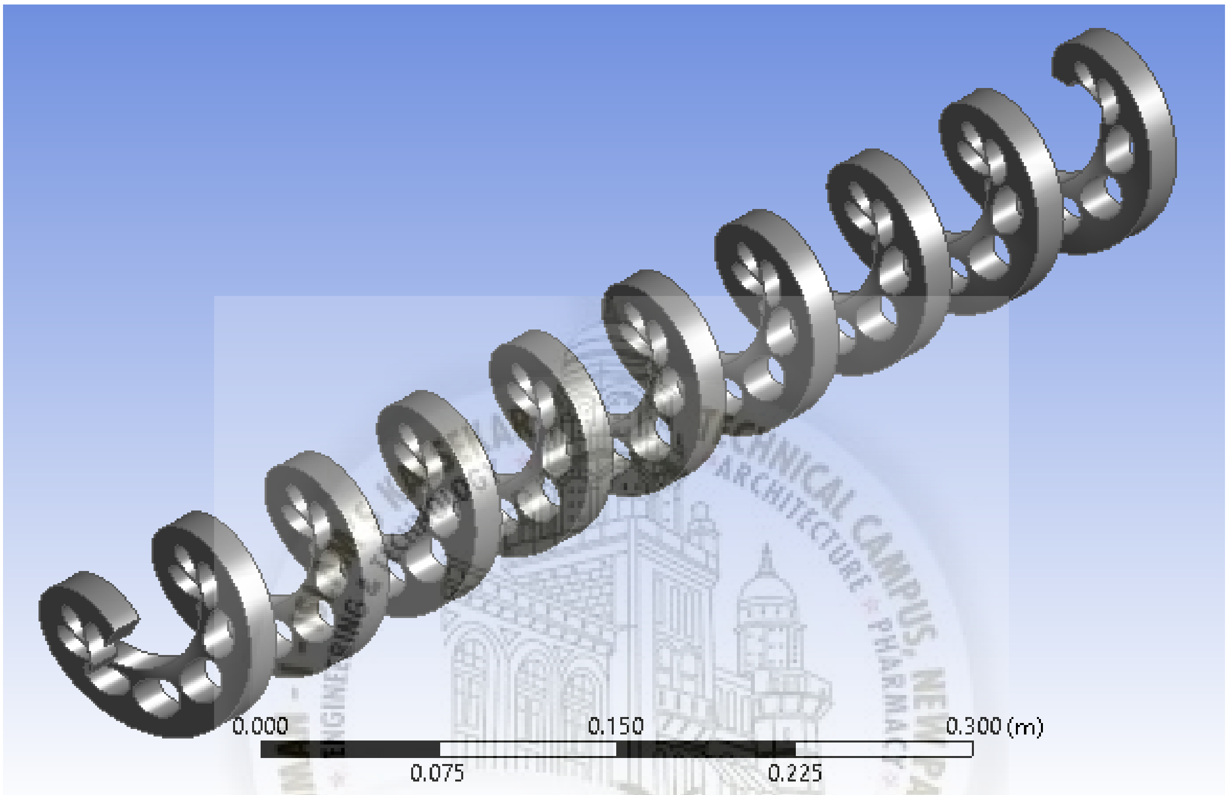

Figure 2.1: Continuous Helical Baffle.

2.1.3 HELICAL BAFFLE HEAT EXCHANGER

The Helical Baffle Heat Exchanger is also known as a Helix changer is a solution that removes many of the deficiencies of Segmental Baffle Heat Exchanger. It is very effective where heat exchanger is predicted to be faced with vibration condition. A continuous type helical baffle is arranged right angle to the tube axis in a sequential pattern that guide the shell side flow in a helical path over the tube bundle. The Helical flow provides the necessary characteristics to reduce flow dispersion and generate near plug flow conditions. The shell side flow configuration offers a very high conversion of pressure drop to heat transfer. Advantages over segmental STHX are increased :- heat transfer rate, reduced bypass effects, reduced Shell Fouling Factor, Prevention of flow induced vibration Reduces Pumping cost. Shell and tube type heat exchanger with helical baffle diagram is shown in Figure 2.2.

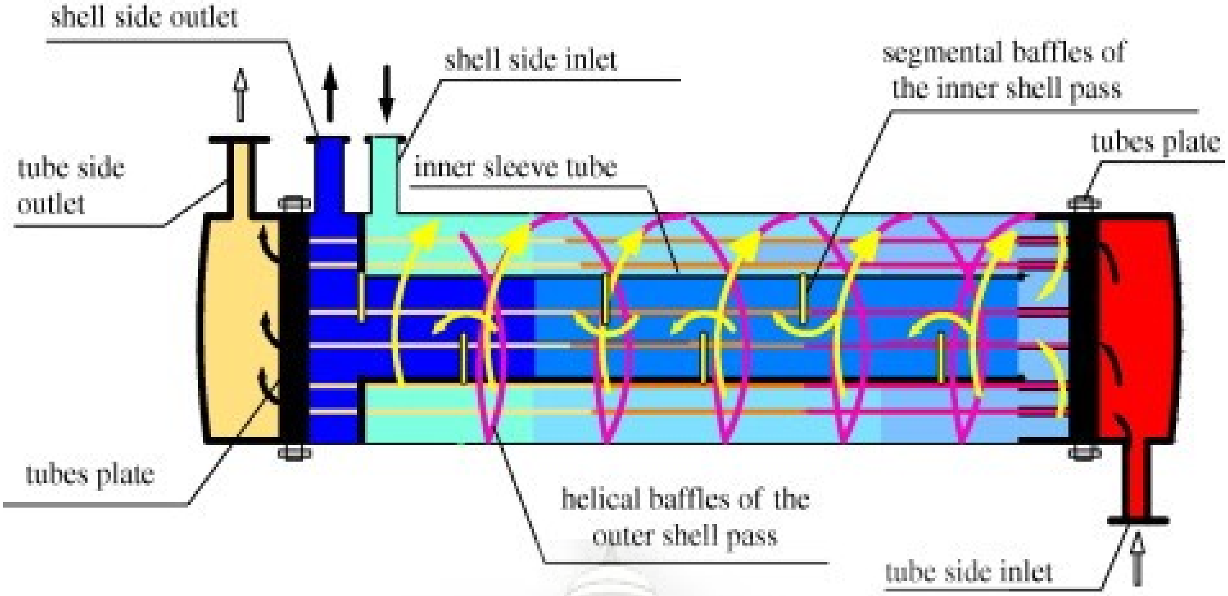

Figure 2.2: Helixchanger.

2.1.4 COMPUTATIONAL FLUID DYNAMICS (CFD)

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 on going research. Onwards the aerospace industry has integrated CFD techniques into the design, R and 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. Furthermore,motor vehicle manufactures now routinely predict drag forces,underbonnet air flows and surrounding car environment with CFD. Increasingly CFD is becoming a vital component in the design of industrial products and processes.

2.1.5 APPLICATIONS OF CFD

CFD not just spans on chemical industry, but a wide range of industrial and non-industrial application areas which are :-

a. Aerodynamics of aircraft and vehicle.
b. Combustion in IC engines and gas turbine in power plant.
c. Loads on offshore structure in marine engineering.
d. Blood flows through arteries and vein in biomedical engineering.
e. Weather prediction in meteorology.
f. Flow inside rotating passages and diffusers in turbo-machinery.
g. External and internal environment of buildings like wind loading and heating or ventilation system.
h. Mixing and separation or polymer moldings in chemical process engineering.
i. Distribution of pollutants and effluent in environmental engineering

2.2 ANSYS

Ansys is the finite element analysis code widely use 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 steps are followed:-

1. During pre-processing 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 behaviour of the problem. For transient problem boundary condition are also defined.

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

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

Chapter 3

COMPUTATIONAL MODEL FOR HEAT EXCHANGER

3.1 PROBLEM DESCRIPTION

Design and analysis of shell and tube heat exchanger with continuous helical baffle using CFD. To study the pressure drop and heat transfer coefficient with different shell side inlet velocity.

3.2 COMPUTATIONAL MODEL

The computational model of an experimentally tested Shell and Tube Heat Exchanger (STHX) with continuous helical baffle is shown in Fig.2.1, and the geometry parameters are listed in Table 3.1. As it can be seen from Fig 3.1 ,the simulated Shell and Tube Heat Exchanger (STHX) has a continuous helical baffle in the shell side direction with total number of 7 tubes. 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 :

1. The shell side fluid has 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 is kept constant in the whole shell side.
6. The heat exchanger is well insulated hence the heat loss to the environment is totally neglected .

3.3 GOVERNING EQUATIONS

The turbulence model used is the realizable k- model with enhanced wall function. The k- model is well able to capture the physics of the flow under curvature, vortices or rotation. Here only the shell side analysis is done with the assumption that the flow and heat transfer are steady and turbulent, and the working fluid is incompressible.

The governing equations are as follow:-

$$\rho\frac{Du}{Dt} = -\frac{\partial p}{\partial x} + div(\mu grad u) + S_{Mx} \tag{1}$$

$$\rho\frac{Dv}{Dt} = -\frac{\partial p}{\partial y} + div(\mu grad v) + S_{My} \tag{2}$$

$$\rho\frac{Dw}{Dt} = -\frac{\partial p}{\partial z} + div(\mu grad w) + S_{Mz} \tag{3}$$

Figure 3.1: Navier Stokes Equation.

Governing equations of the flow of a compressible Newtonian fluid are as follows :-

Continuity

$$\frac{\partial \rho}{\partial x} + div(\rho u) = 0$$

x-momentum

$$\frac{\partial(\rho u)}{\partial x} + div(\rho u u) = -\frac{\partial p}{\partial x} + div(\mu grad u) + S_{Mx} \tag{4}$$

y-momentum

$$\frac{\partial(\rho v)}{\partial y} + div(\rho v u) = -\frac{\partial p}{\partial y} + div(\mu grad v) + S_{My} \tag{5}$$

z-momentum

$$\frac{\partial(\rho w)}{\partial z} + div(\rho w u) = -\frac{\partial p}{\partial z} + div(\mu grad w) + S_{Mz} \tag{6}$$

Energy

$$\frac{\partial(\rho i)}{\partial t} + div(\rho i u) = -p div u + div(k grad T) + \Phi + S_i$$

(7)

Figure 3.2: Governing Equations.

3.4 MODEL GEOMETRY

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

Table 3.1: Geometric dimensions of shell and tube heat exchanger.

Sr. No	Geometrical Parameters	Value(mm)
1.	Heat Exchanger length, L	600
2.	Shell Outer Diameter, Do	110
3.	Shell Internal Diameter, Di	90
4.	Number of Tubes, Nt	7nos
5.	Tube External Diameter, To	20
6.	Tube Internal Diameter, Ti	17

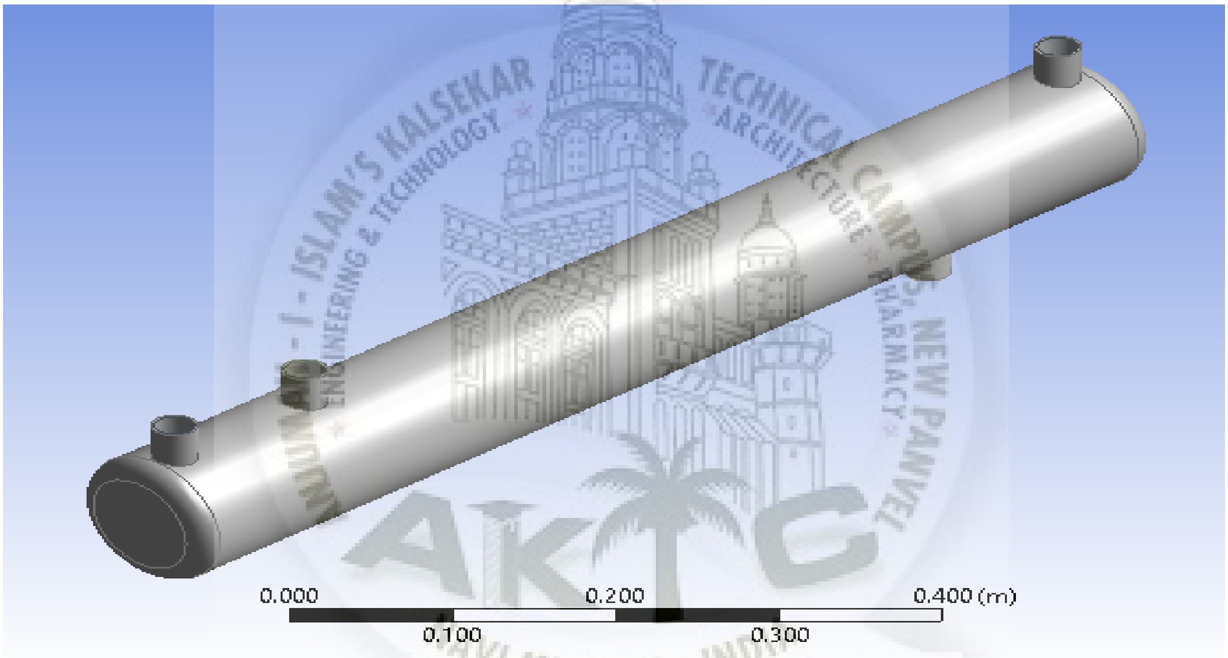

Figure 3.3: STHX.

The geometry of the model was created in AUTODESK INVENTOR 2017 and imported to ANSYS FLUENT as shown in the Fig 3.3. The fluidic domain inside the shell side and tube side flow was created in the ANSYS DESIGN MODELER. The Fig 3.4, shows the fluidic domain in both the side i.e. shell and tube side.

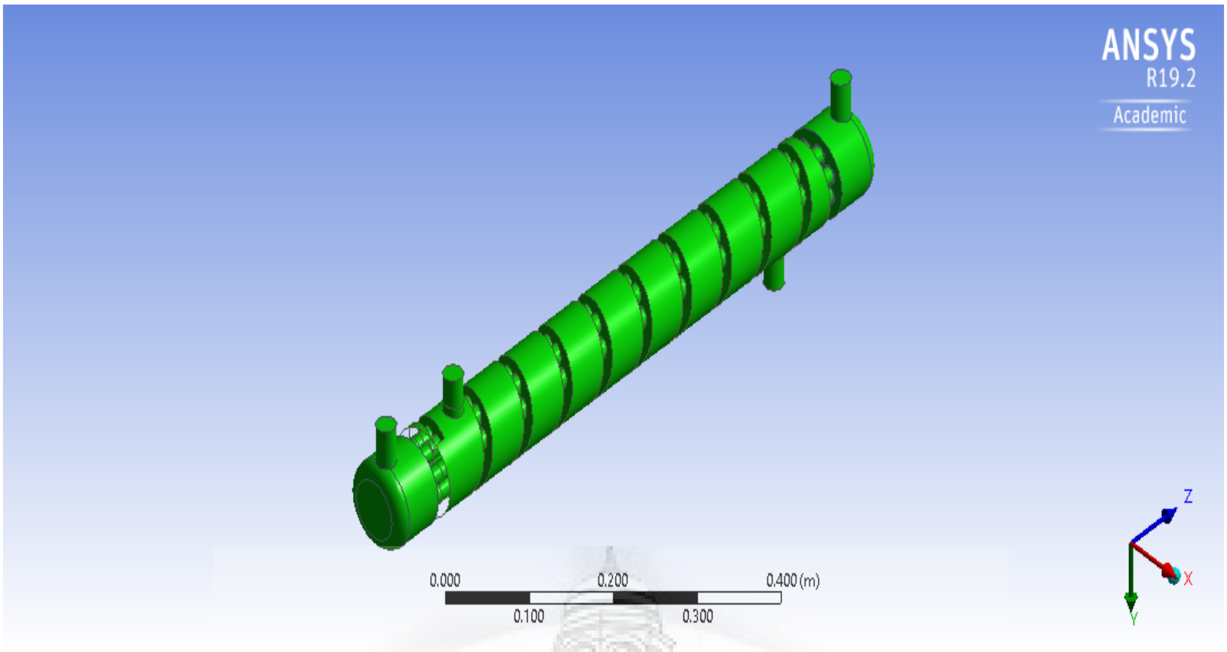

Figure 3.4: Fluidic domains of shell and tube side.

The material of the baffle is selected as Aluminium and that of shell and tubes is selected as copper. The working fluid in both shell side and tube side of heat exchanger is water.

Water is considered as a Newtonian and incompressible fluid with constant thermo-physical properties. Furthermore, the fluid flow and heat transfer processes are turbulent and in steady-state. The viscous heating and compression work are both trivial and thus are neglected in the energy equation. The heat exchanger is assumed to be newly built and thus has a negligible fouling resistance. The leakage between the tube and baffle and between baffle and shell is negligible and thus ignored.

In this study, a hydrodynamic model based on the unstructured-grid finite volume method was developed using ANSYS FLUENT software. This model was based on the numerical solution of continuity, momentum and energy equations.

3.5 BOUNDARY CONDITIONS

The boundary conditions which are used are as follows :-

1. Inlet Fluid Temperature : 300K
2. Shell Side Inlet Velocity : 0.25, 0.35, 0.45 m/s.
3. Outer Wall : Adiabatic, non-slip boundary.

3.6 MESHING

After the model was fully developed using AUTODESK INVENTOR and ANSYS DESIGN MODELER, it was then imported to ANSYS MESHING. The named selections were created and then the automatic contacts were checked. The defective contacts were corrected by using manual contact tool.

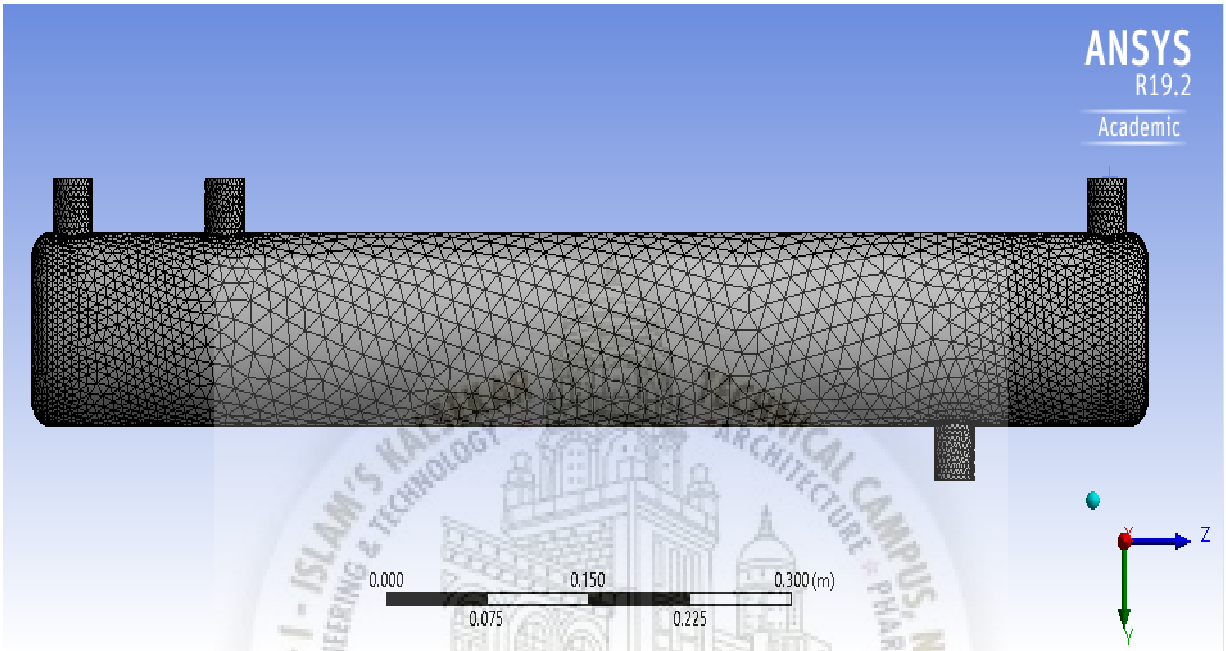

Figure 3.5: Front view of Automatic Meshing.

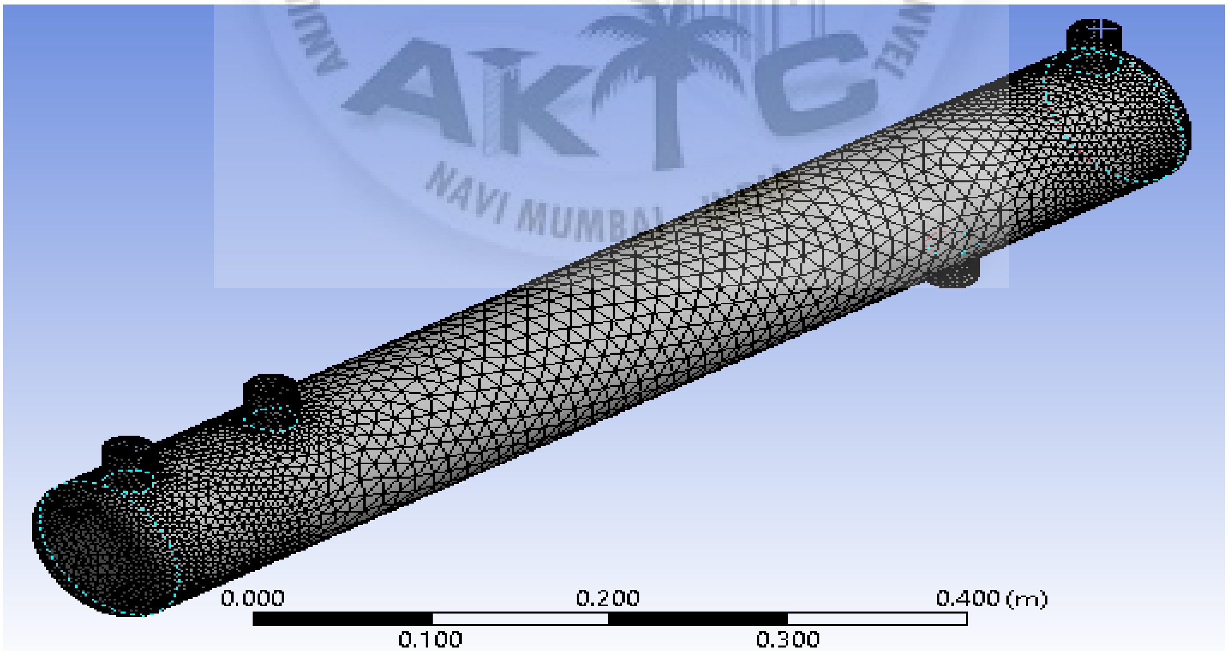

Figure 3.6: Isometric view of Automatic Meshing.

The mesh was generated using the Automatic mesh generation in the GUI. The result of it is as shown in the Fig3.5 and Fig 3.6, wherein the Tetrahedral mesh was dominant in the major parts of the geometry thereby giving 789512 cells. The ANSYS Pre-Processor STUDENT ACADEMIA software can handle only upto 512000 cells and hence a method of refinement and using the different meshing methods in order to reduce the cells and also to capture the proper flow simulation details was used.

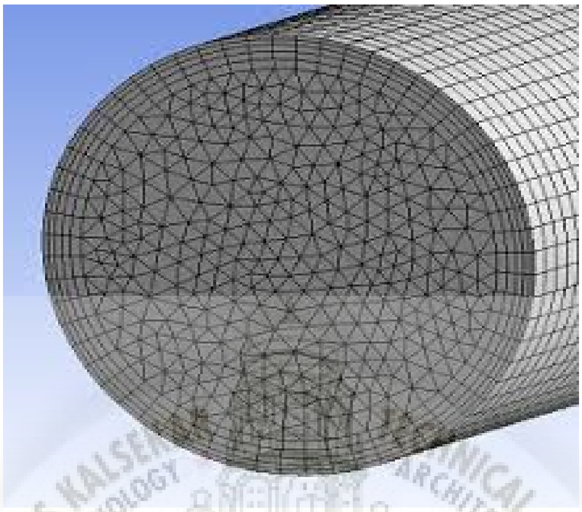

Figure 3.7: Inflation Layer.

Hex Dominant, Sweep and Tetrahedron-Patch Conforming methods were used in different parts of the geometry in order to obtain a clean mesh and also to reduce the number of cells. Inflation layer is grown on the tube walls and the outer wall with predetermined first cell height as shown in the Fig 3.7.

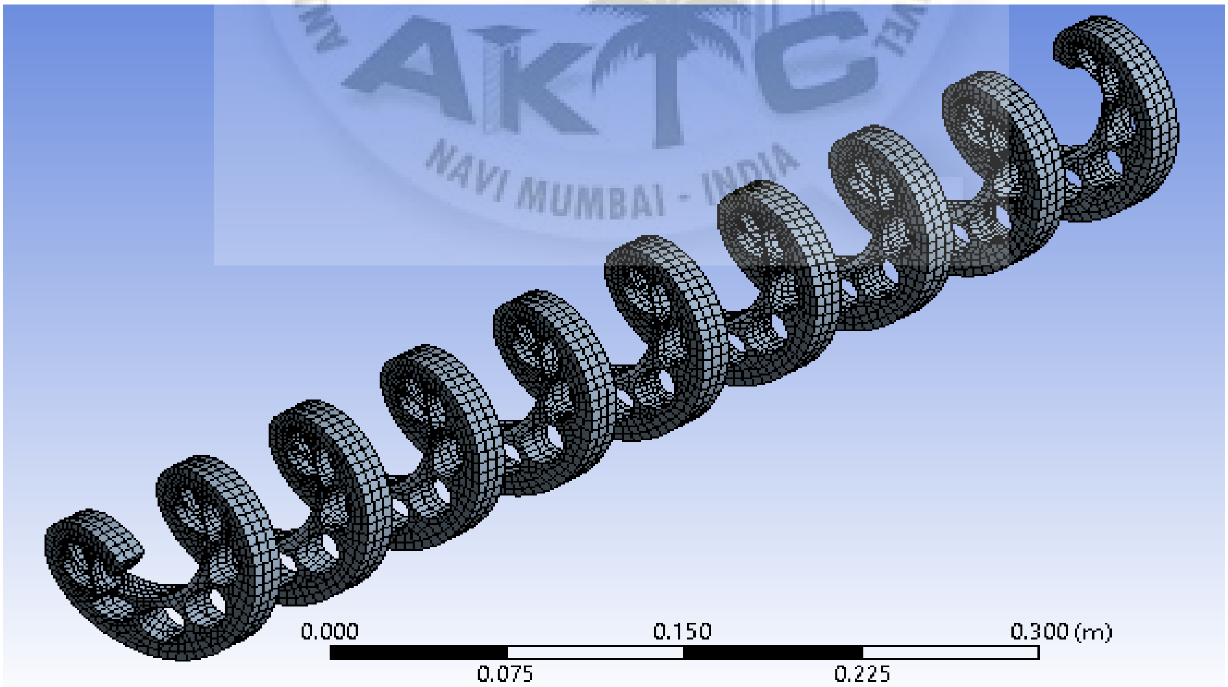

Figure 3.8: Meshed Continuous Helical Baffle.

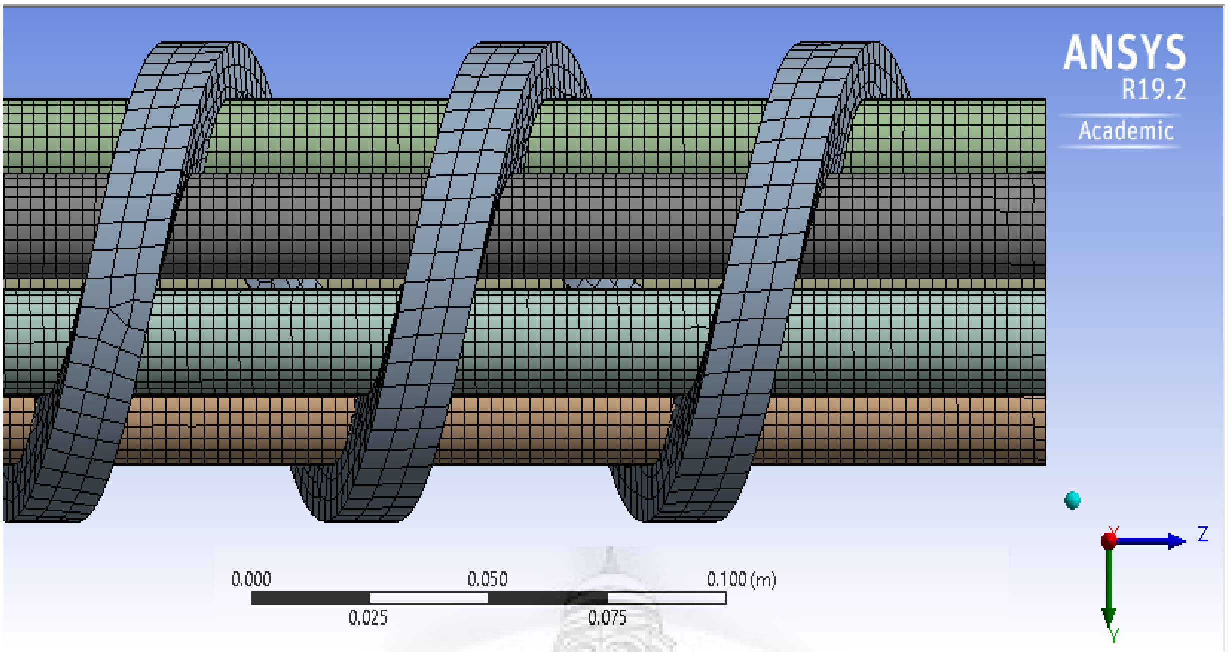

Figure 3.9: Meshed Tubes and Baffle Assembly.

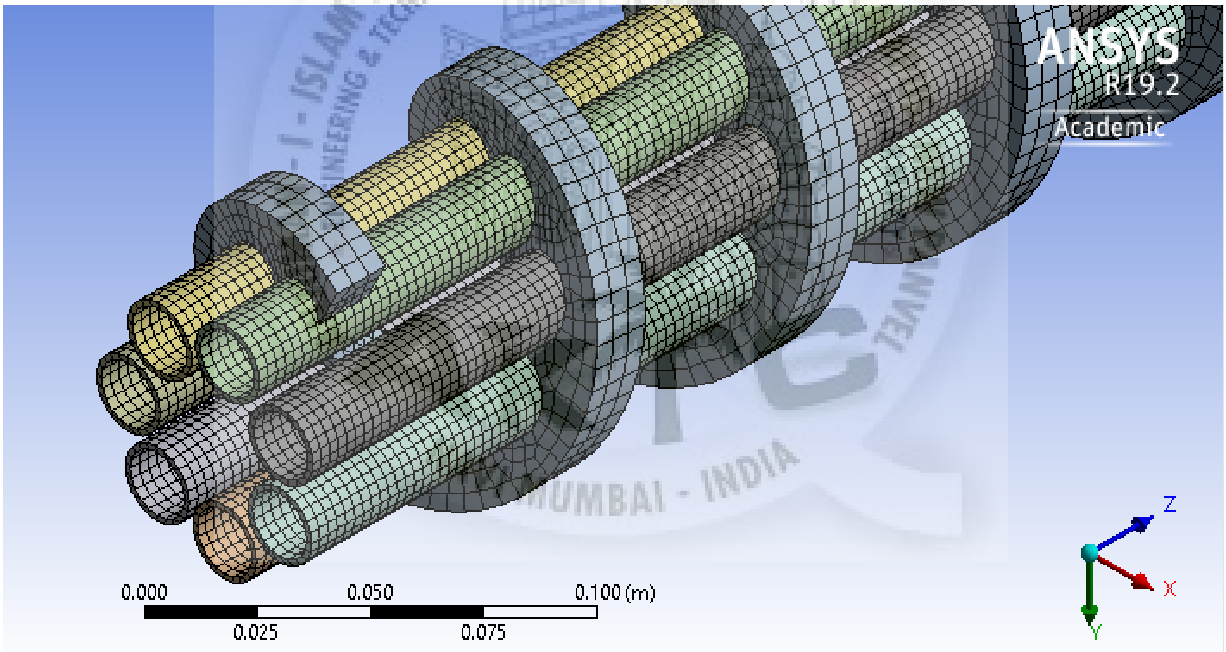

Figure 3.10: Meshed Tubes and Baffle Assembly.

Hexahedral elements were created onto the continuous helical baffle and the tubes in order to capture the proper flow insight as shown in the Fig 3.8, Fig 3.9 and Fig 3.10.

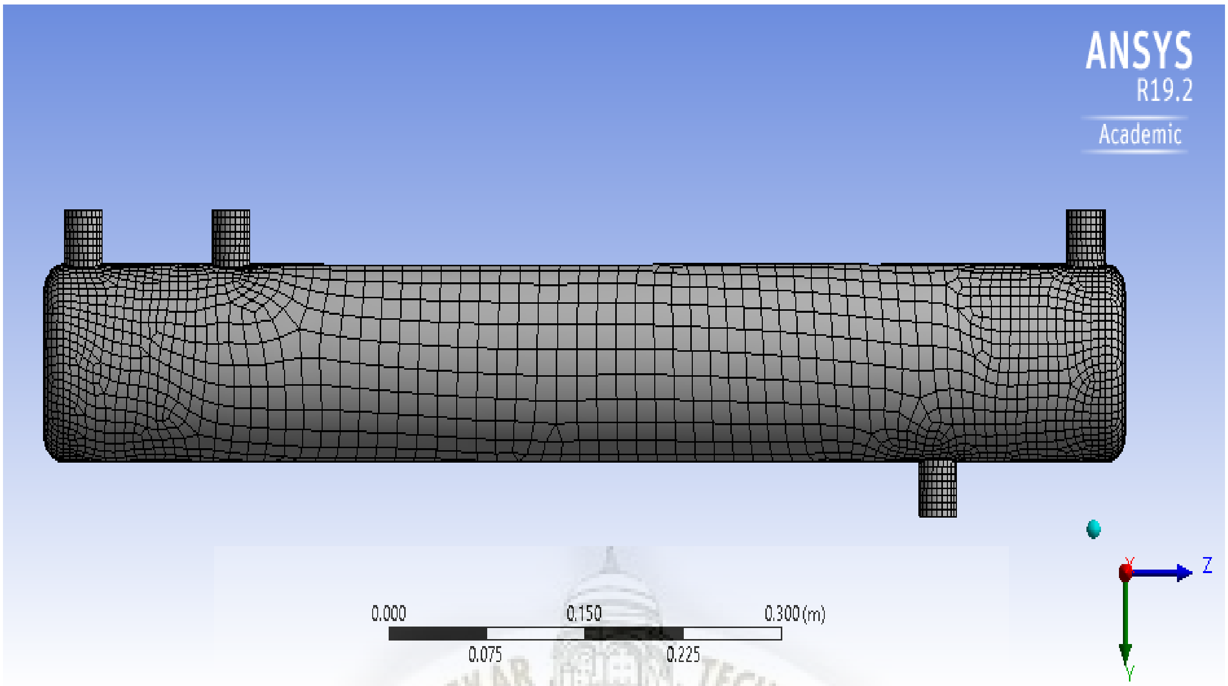

Figure 3.11: Front View of the final Mesh.

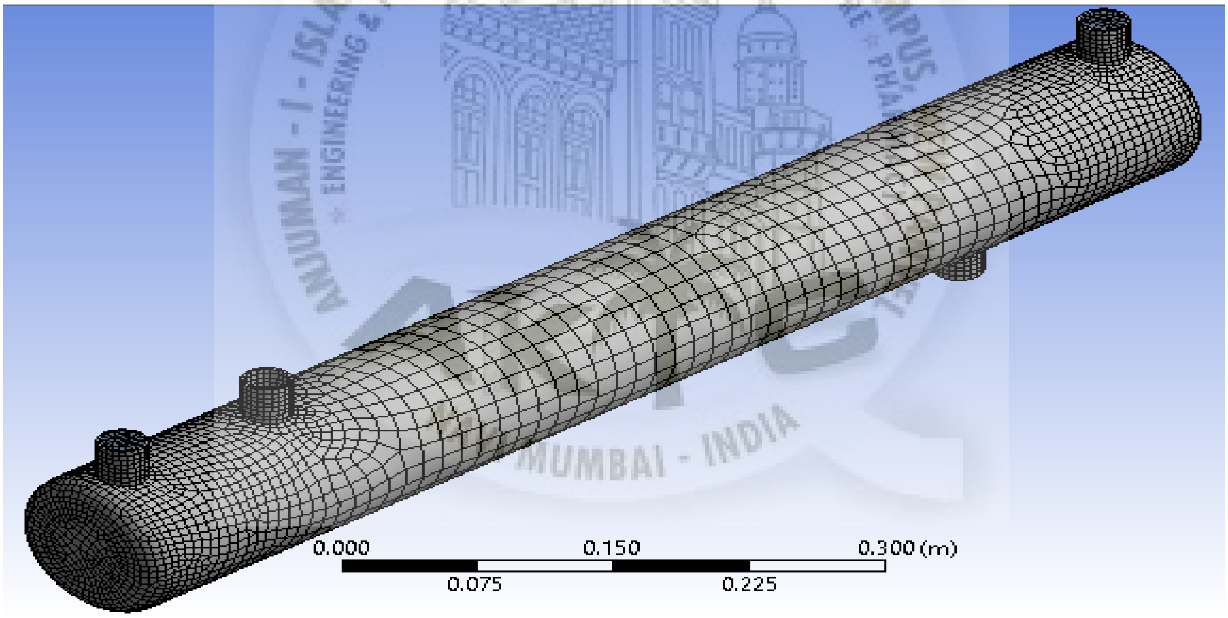

Figure 3.12: Isometric View of the final Mesh.

The result of it is as shown in Fig.3.11, Fig 3.12 and the number of cells were reduced to 457200. After the meshing was checked and confirmed for error free, it was sent to ANSYS Pre-Processor for setup and initialization.

3.7 PROBLEM SETUP

Simulation was carried out in ANSYS FLUENT v19.2. The quality of the mesh was evaluated using built-in Mesh Metrics in ANSYS Pre-processer.

The momentum boundary condition of no slip and no penetration is set for all the solid walls.

The thermal boundary condition of zero heat flux is set for shell wall and inlet and outlet nozzle walls, while the wall of tubes, baffles, and tube bundle, which also represent the solid-fluid interfaces between the two fluid domain and the solid domain, have the thermal boundary condition of coupling heat transfer (two interfaces with coupled wall).

The inlets for the shell and tube sides are set as boundary conditions of the velocity-inlet, the outlets are set as pressure-outlet. The outlets are assumed to have a pressure of zero so the inlet pressure is equal to the pressure drop on both shell and tube sides.

In the Fluent solver Pressure based type, absolute velocity formation and steady time was selected for the simulation .

In the model option energy equation was set ON and the viscous was set as standard k-e, standard wall function (k-epsilon 2 eqn).

In cell zone, fluid water-liquid, copper and aluminum was selected as materials for simulation.

The Inlet Temperature was set as 300K.

3.8 SOLUTION INITIALIZATION

The commercial ANSYS FLUENT is used to calculate the fluid flow and heat transfer in the computational domains.

The governing equations are iteratively solved by the finite-volume formulation with the SIMPLE algorithm.

The second order upwind scheme is adopted for the momentum, energy, turbulence and its dissipation rate.

The pressure term is treated with the standard scheme. Default under relaxation factors of the solvers are used, which are 0.3, 0.7, 0.8 and 0.8 for the pressure, momentum, turbulent kinetic energy, and turbulent dissipation energy, respectively.

The convergence criterion is that the normalized residuals are less than 1e-4 for the flow equations and 1e-8 for the energy equations.

Solution initialization was standard method and solution was initialized from inlet with 300K temperature.

Chapter 4

RESULTS AND DISCUSSION

Under the above boundary condition and solution initialization condition, the simulation was set for 100 iterations.

4.1 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 1e-4 and the energy should be less than 1e-8. If these all values are in the same manner then the solution will be converged.

The scaled residuals and the solution convergence is as shown in the Fig.4.1.

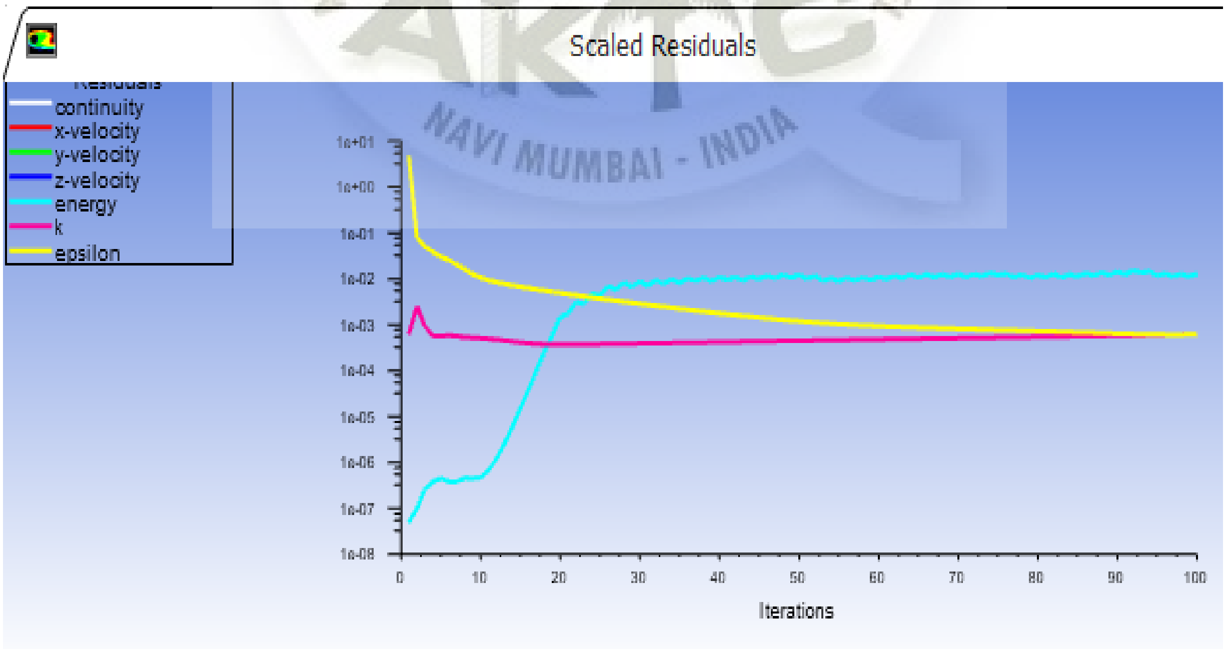

Figure 4.1: Residual.

4.2 METHODS OF THERMO-HYDRAULIC PERFORMANCES

Using the computed results, heat transfer coefficients on both shell and tube side are calculated by Newton's cooling law from the numerical temperature field. As specified in the boundary conditions in the FLUENT, the shell and tube side outlets are assumed to have a pressure of zero, thus, the pressure drop is equal to the inlets pressure for both shell and tube side. The overall thermo-hydraulic performance on the shell side is expressed by the heat transfer coefficient per unit pressure drop, i.e., h/dp.

It is noted that the comparision between the segmental and continuous helical baffles STHX are based on the assumptions that the initial conditions are the same.

4.3 FLOW FIELD CHARACTERISTICS

The flow behavior in the shell side for the STHX is different depending on the baffle type. For the conventional segmental baffle, as shown in Fig. 4.2, the zig-zag flow pattern causes large dead zones, eddy formation and fluid recirculation at the back of the baffles. A large amount of the fluid energy is spent in areas where there are few tubes, resulting in an inefficient conversion of pressure drop in heat transfer.

Figure 4.2: Segmental.

Figure 4.3: Helical.

With the helical baffles, as shown in the Fig. 4.3 , the flow pattern in the shell side is rotational, and the shell fluid passes through the tube bundles close to an ideal helical pattern. Fluid flow with the helical baffles is continuous and the dead zones do not occur near the helical baffle. The spiral motion brings about better mixing, and the heat transfer in this region is significantly enhanced while the pressure drop is reduced.

4.4 VARIATION OF SHELL SIDE VELOCITY AND OUTLET TEMPERATURE

The Fig. 4.4. shows the variation of outlet temperature for different shell side inlet velocity.

Inlet Velocity (m/s)	Inlet Temperature(K)	Outlet Temperature (K)
0.25	300	351.72
0.35	300	349.54
0.45	300	346.23

Figure 4.4: Temperature Variation at the Outlet.

It can be conferred from the data that as the shell side inlet velocity is increasing for the given velocities as mentioned, the outlet temperature goes on decreasing. But this decrease is not at the expense of lower heat transfer as the drop in pressure increases in the outer shell because of the inner helical baffle.

The Fig. 4.5 shows the Velocity contour in the STHX.

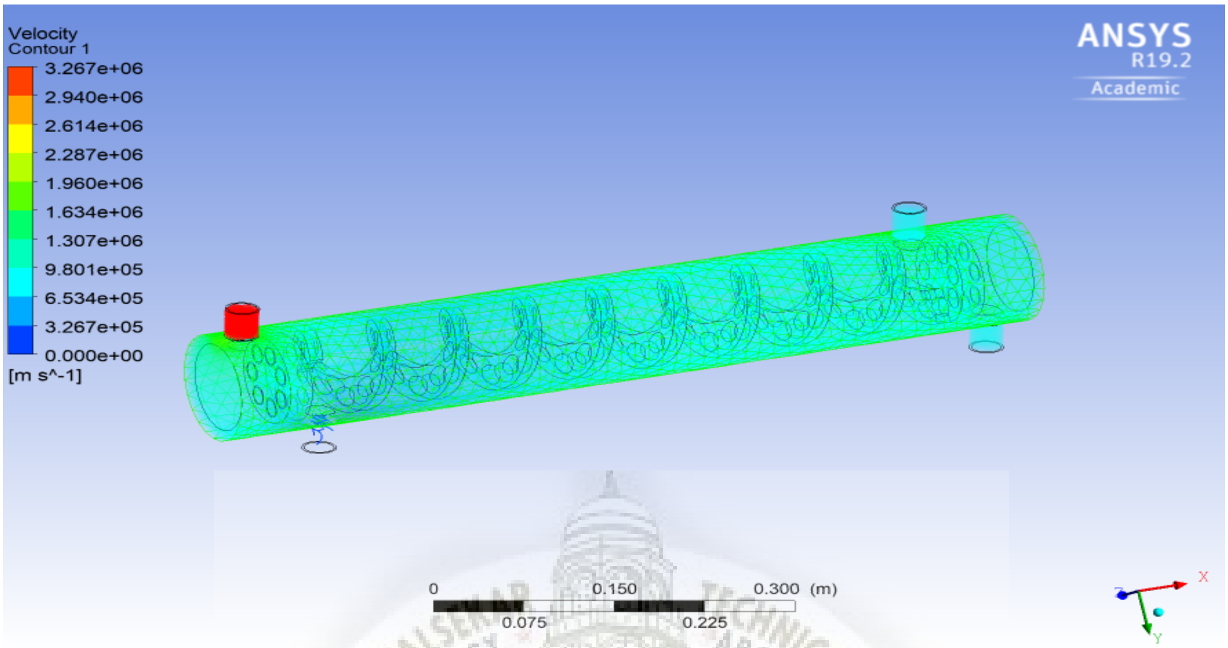

Figure 4.5: Velocity contour.

4.5 PRESSURE DROP

The pressure drop is of great importance in the design of shell-and-tube heat exchangers, pumping cost are highly related to pressure drop, and therefore lower pressure drop results in lower operating costs.

The pressure contour for the continuous helical baffle is as shown in the Fig.4.6.

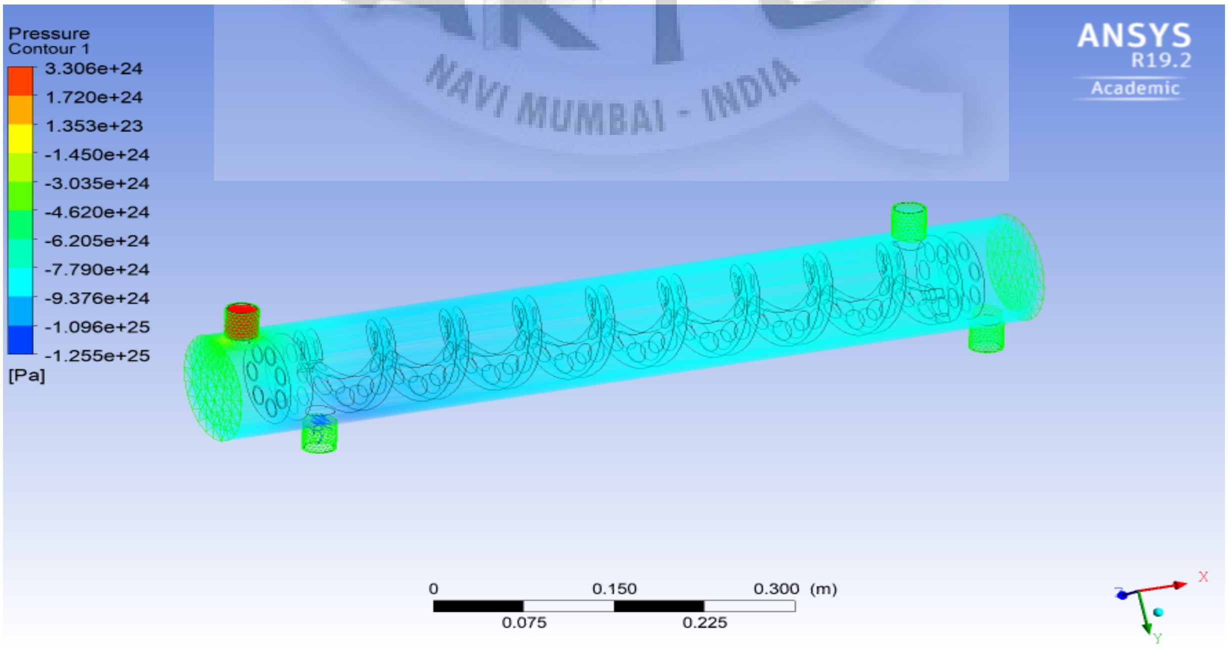

Figure 4.6: Pressure contour.

Helical baffles produce the lowest maximal velocities, a better flow distribution, without dead zones and fluid re-circulation areas. Higher maximal velocities, dead zones, and fluid re-circulation areas always produce an increase in frictional pressure drops.

Fig. 4.7. depicts the variation of the pressure drop versus the shell side inlet velocity for the segmental and continuous helical baffle STHX.

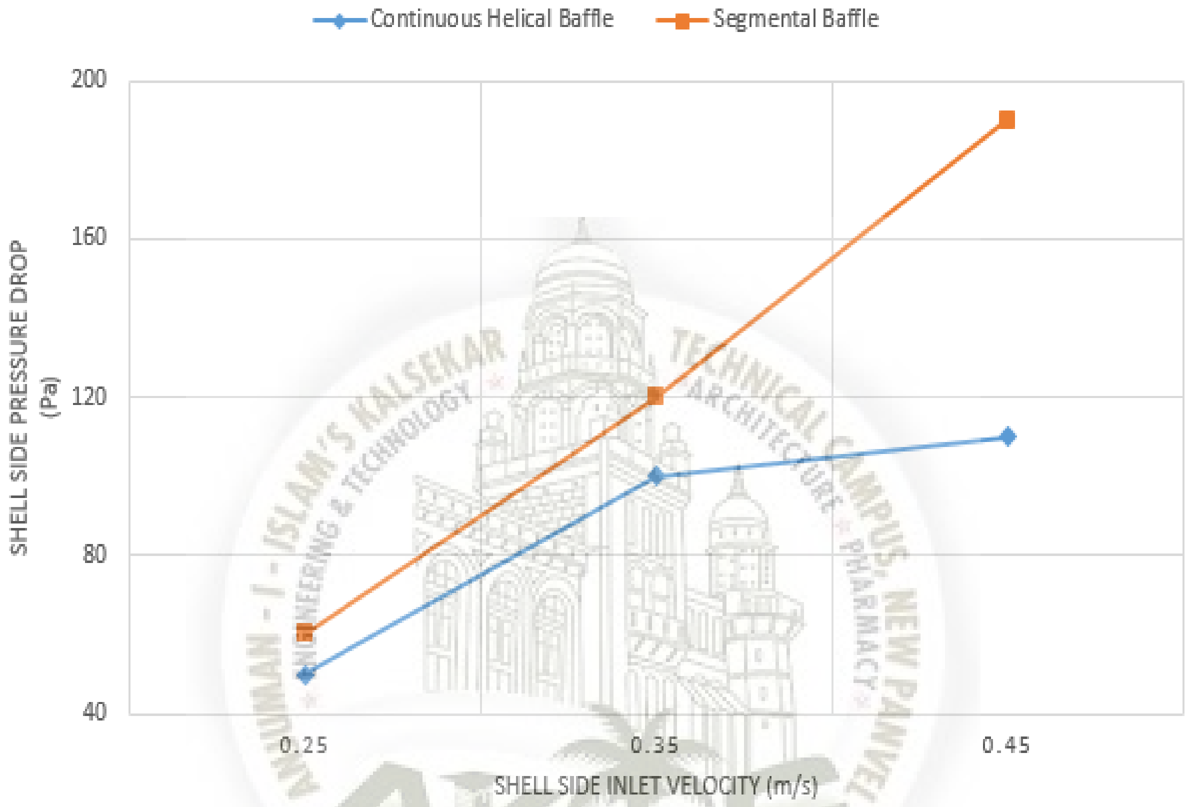

Figure 4.7: Graph of Pressure Drop v/s Inlet Velocity.

The pressure drop increases proportional to the inlet velocity. It can be inferred from the graph that the pressure drop for the STHX with helical baffles is lower than the pressure drop for the STHX with segmental baffles. The reason is that the flow distribution with segmental baffles on the shell side is zigzag, flow separation at the edge of baffles causes abrupt momentum change and severe pressure drop. Whereas the primary flow direction of helical baffles does not change dramatically.

4.6 HEAT TRANSFER PERFORMANCE

The heat transfer rate is given by :- Q = m * Cp * T
m=mass flow rate
Cp = Specific Heat of Water
T = Temperature Difference Between Tube Side

Fig. 4.8 represents the comparison of the shell side heat transfer coefficient of continuous helical and segmental baffle.

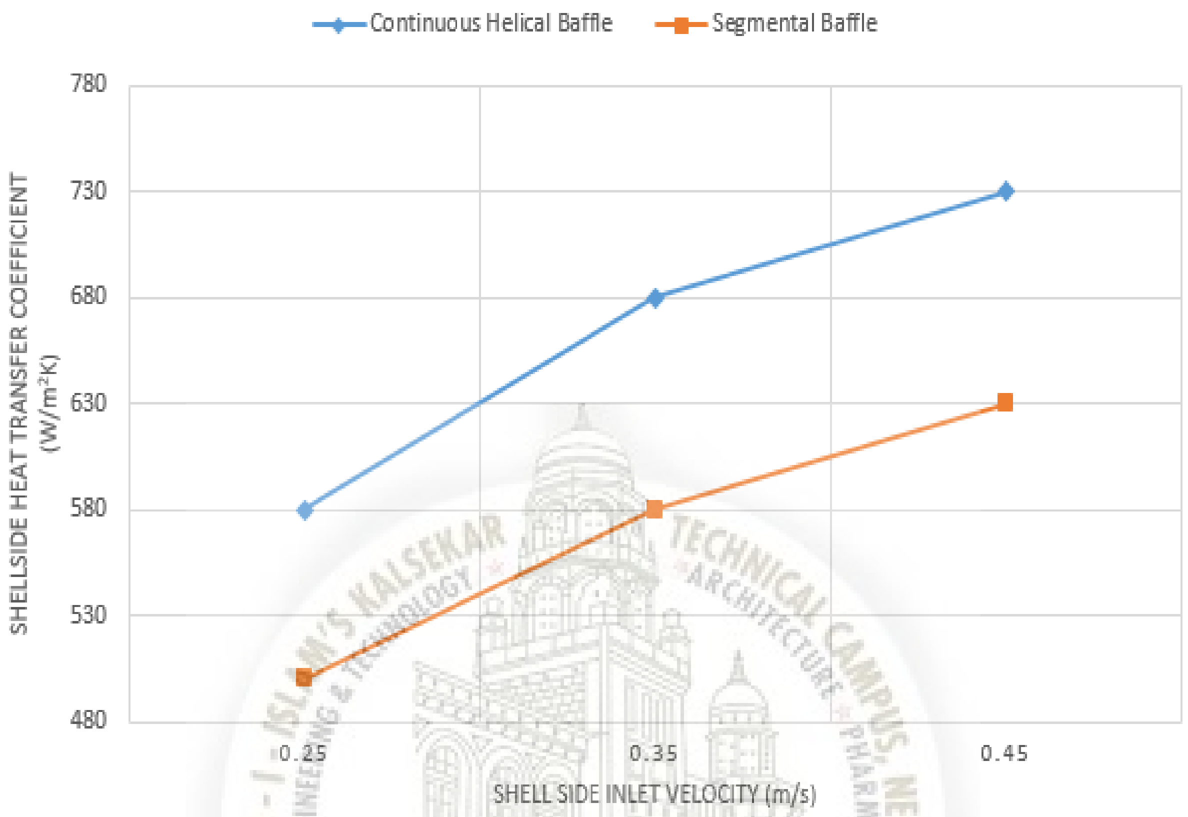

Figure 4.8: Graph of Heat Transfer Coefficient v/s Inlet Velocity.

It can be observed that the shell side heat transfer coefficient increases with increase in the shell side mass flow rate. Heat transfer coefficient for helical baffle is maximum and minimum for segmental baffle. The higher maximum velocities generated by the turbulence caused in due to the helical baffle results in notable thermal enhancement on the shell side.

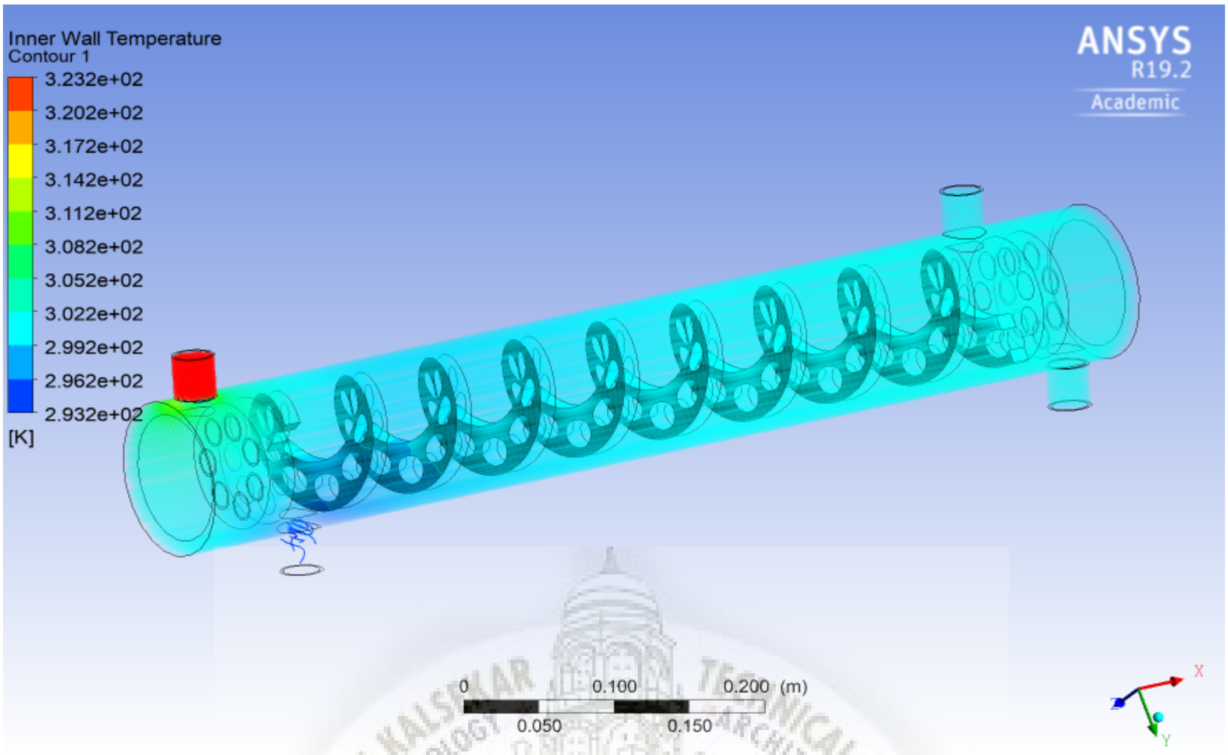

Figure 4.9: Inner Wall Temperature Contour.

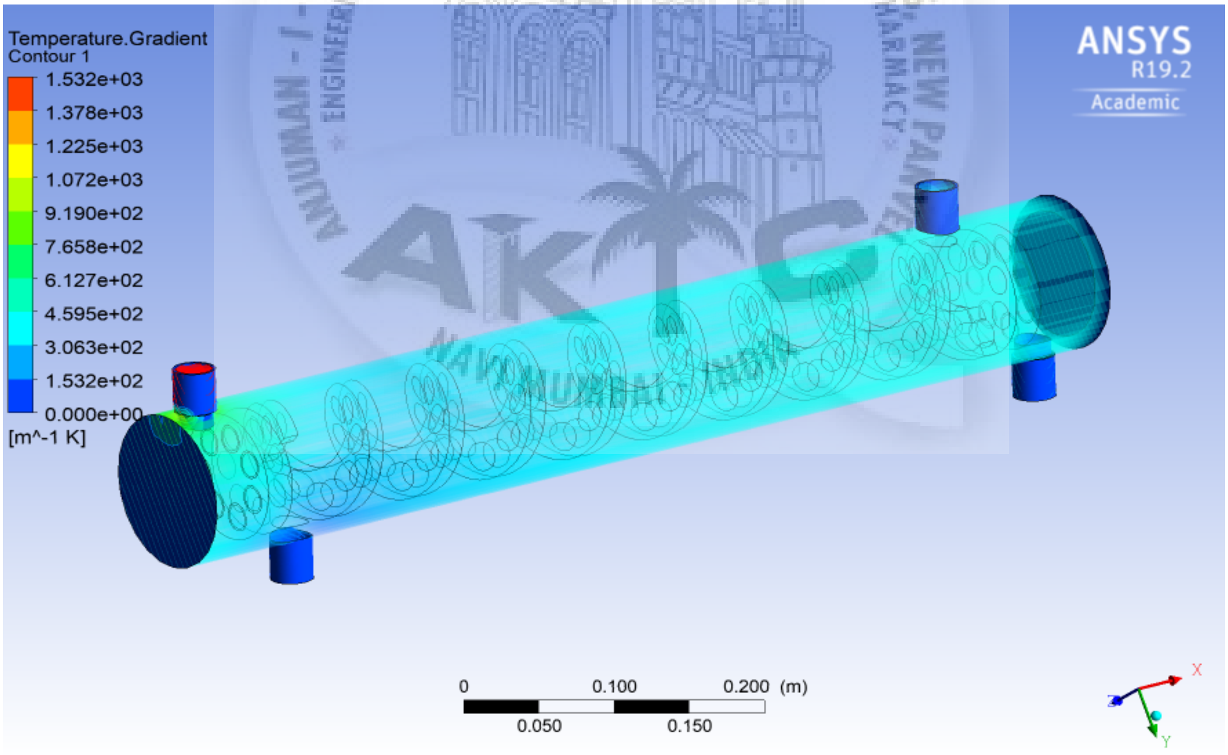

Figure 4.10: Shell Side Temperature Gradient Contour.

Fig. 4.9 and Fig. 4.10 shows the Temperature Gradient Contour of the shell side and the Inner Wall Temperature Contour respectively.

4.7 COMPREHENSIVE PERFORMANCE ANALYSIS (h/dp)

The design of a shell-and-tube heat exchanger is a compromise between higher heat transfer coefficients and lower pressure drop in the fluids, since the two parameters are highly dependent on each other. In order to improve the heat transfer coefficient, it is necessary to increase the fluid velocities. This always produces an increase in frictional pressure drops. Heat transfer coefficient per unit pressure drop at the shell side, h/dp is adopted to evaluate the optimal ranges for both parameters. In practical application the pressure drop of the heat exchangers are usually limited, the goal is to find the design parameter combination that results in the highest heat transfer coefficient within the pressure drop limitations. Thus, the ratio h/dp should be a more reasonable comparison quantity.

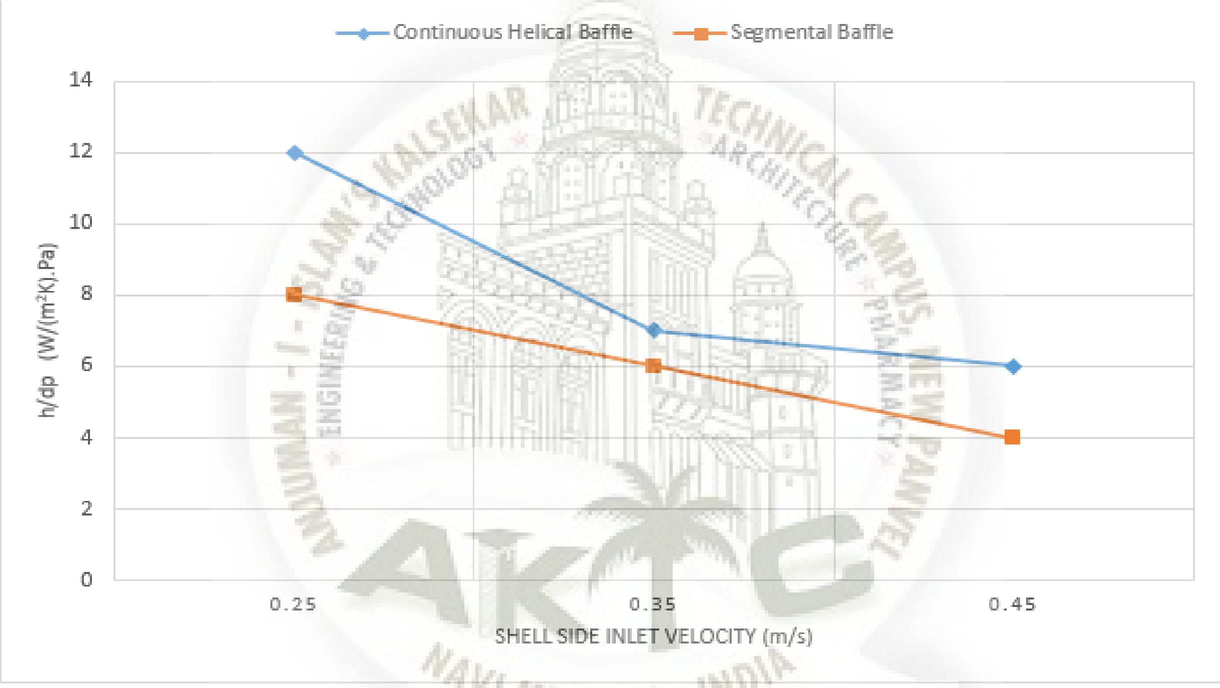

Figure 4.11: Graph of h/dp v/s Inlet Velocity.

From Fig. 4.11, it can be seen that the performance parameter h/dp decreases with increase in the shell side flow rate. It is clearly seen that the STHX with helical baffles has the best performance ratio h/dp.

Based on the results discussed above and for the interval of flow rate used, it is clear that STHX with helical baffles performs better for the flow rates interval studied.

If a new STHX is to be designed to replace an existing one, if the two heat exchangers have equal pressure drop, the new STHX must have a larger heat transfer capacity, and if the two heat exchanger have equal heat transfer capacity, the new STHX must have a lower pressure drop, thus, saving much pumping power.

Chapter 5

CONCLUSION

In the present study, a numerical model is used to compute and compare the thermo-hydraulic performances of shell-and-tube heat exchangers with different baffle types: segmental and helical baffles.

Flow analysis in shell side, showed that velocity distribution in helical baffles is more uniform and homogeneous compared to segmental baffles. This leads to less dead zones and less fluid re-circulation areas inside the shell.

The results indicate that, compared to the conventional segmental baffles, the use of helical baffles provide a good balance between heat transfer and pressure drop characteristics. Thus, the highest Thermo-hydraulic performance is achieved by using helical baffles.

Chapter 6

FUTURE SCOPE

1. CFD eliminates the need of prototype.
2. The comparison between baffle types on induced fouling resistances, flow vibrations and leakages.
3. The Software provided with images and data can predict the performance of the design easily.
4. The study of the effect of varying other design parameters that have an effect on the shell-side Thermo-hydraulic performances.
5. For Large Pressure flow, Direct Numerical Simulation (DNS) method can be used extensively.
6. Based on the analysis of CFD, certain modifications can be done on the existing models, so as to increase their efficiency.

Chapter 7

REFERENCES

1. Mukharji R. (1988). Effective design of shell and tube heat exchanger, American Institute of Chemical Engineering, Vol. 3, No. 11, pp. 17200-17204.

   DOI: 10.15680/IJIRSET.2014.0311016

2. Serna M., Jimenez A. (2005). A compact formulation of the Bell Delaware method for Heat Exchanger design and optimization, Chemical Engineering Research and Design, Vol. 83, No. A5, pp. 539-550.

   DOI: 10.1205/cherd.03192

3. Lei G.Y., He Y.L., Li R. Gao Y.F. (2008). Effects of baffle inclination angle on flow and heat transfer of a heat exchanger with helical baffles, Science DirectChemical Engineering and Processing, pp. 1-10.

   DOI: 10.1016/j.ijheatmasstransfer

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

   DOI: 10.1155/2011/839468

5. Kumar P., Kumar V., Nain S. (2014). Experimental study on heat enhancement of helix changer with grooved tubes, IJLTET, Vol. 3, No. 4.

6. Shinde S., Pancha M.H. (2012). Comparative thermal performance analysis of segmental baffle heat exchanger with continuous helical baffle heat exchanger using kern method, IJERA, Vol. 2, No. 4.

7. Jian-Feng Yang, Min Zeng, Qiu-Wang Wang. Numerical Investigation on shell side performances of combined parallel and serial two shell pass shell and tube heat exchangers with continuous helical baffles.

   Applied Thermal Eng. 2014 Vol 11.029

ACHIEVEMENTS

1. PUBLICATIONS

   (a) NAME OF PAPER PUBLISHED :

   ”CFD ANALYSIS ON CONTINUOUS HELICAL BAFFLE OF SHELL AND TUBE HEAT EXCHANGER.”
   NAME OF AUTHOR :

   SINGH VISHAL, RUMANE ABDULLAH
   SHETTY SANJOTH, GUPTA VIPIN.
   NAME OF JOURNAL : IJRAR JOURNAL
   DATE OF PUBLICATION : APRIL 2019
   WEBSITE : http://ijrar.com

2. PROJECT COMPETITIONS

   (a) NAME OF PAPER:

   ”CFD ANALYSIS ON CONTINUOUS HELICAL BAFFLE OF SHELL AND TUBE HEAT EXCHANGER.”
   NAME OF AUTHORS :

   SINGH VISHAL, RUMANE ABDULLAH
   SHETTY SANJOTH, GUPTA VIPIN.
   NAME OF EVENT:

   CALIBRE NATIONAL LEVEL TECHNICAL PAPER PRESENTATION.
   DATE AND VENUE OF THE EVENT : MARCH 2019(VASHI).