International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

IJIEMR Transactions, online available on 15th FEB 2019. Link :
http://www.ijiemr.org/main/index.php?vol=Volume-08&issue=ISSUE-02

Title: CFD ANALYSIS OF A SHELL AND TUBE HEAT EXCHANGER

Volume 08, Issue 02, Pages: 19–31.

Paper Authors

DOMMETI VASU KIRAN,P. CHANDRA SEKHAR

AIMS College of engineering, Mummidivaram

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

CFD ANALYSIS OF A SHELL AND TUBE HEAT EXCHANGER

DOMMETI VASU KIRAN[1] ,P. CHANDRA SEKHAR[2]
[1] PG scholar, Dept. of M.E., AIMS College of engineering, Mummidivaram
[2] Professor& Head, Dept. of M.E., AIMS College of engineering, Mummidivaram

ABSTRACT: Heat exchanger has a variety of applications in different industries and in this study one such heat exchanger is taken in to account. The heat exchanger is designed as per the commercial needs of the industry.This work is concerned about the analysis of the performance of a shell and tube type heat exchanger and variable load conditions this is accomplished by many of equating the geometric model followed by the detailed analysis using the simulation analysis of process.The warmth switch and drift distribution is discussed in element and proposed man equation shall be checked for various inclination of the helical shuffles both single and double the nature of insolence and other flow parameters are nearly modeled and thus the performance of the shell and tube heat exchanger is evaluated all this is being accomplished through the usage of FEM packages of CATIA and ANSYS the model is completed using CATIA and the model generated is exported to ANSYS and the detailed flow analysis is computed using the fluent domain of ANSYS.The variations in the performance of the shell and tube heat exchanger are formed and the appropriate parameter is identified and the fame is suggested for an optional performance of the heat exchange.

1.0 INTRODUCTION

Heat exchangers are devices used to transfer heat energy from one fluid to another. Typical heat exchangers experienced by us in our daily lives include condensers and evaporators used in air conditioning units and refrigerators. Boilers and condensers in thermal power plants are examples of large industrial heat exchangers. There are heat exchangers in our automobiles in the form of radiators and oil coolers. Heat exchangers are also abundant in chemical and process industries. Different heat exchangers are named according to their applications. For example, heat exchangers being used to condense are known as condensers; similarly heat exchangers for boiling purposes are called boilers. Performance and efficiency of heat exchangers are measured through the amount of heat transferred using least area of heat transfer and pressure drop. A 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. A good design is referred to a heat exchanger with least possible area and pressure drop to fulfill the heat transfer requirements.Heat exchangers are one of the mostly used equipment in the process industries. Heat exchangers are used to switch warmness between two approach streams. You possibly can realize their usage that any approach which involves cooling, heating, condensation, boiling or evaporation would require a heat exchanger for these rationale. Procedure fluids quite often are heated or cooled earlier than the system or endure a section exchange. One of a kind warmness exchangers are named in step with their application. For illustration, warmness exchangers getting used to condense are referred to as condensers, in an identical manner warmth exchanger for boiling functions are known as boilers. Efficiency and efficiency of warmness exchangers are measured by means of the quantity of warmness switch making use of least field of warmness transfer and stress drop. A higher presentation of its effectiveness is completed by way of calculating over all warmth switch coefficient. Stress drop and field required for a targeted quantity of heat switch, presents an insight concerning the capital rate and vigor necessities (going for walks price) of a warmness exchanger. Traditionally, there may be tons of literature and theories to design a heat exchanger consistent with the standards.

Heat exchangers are of two types:-

- The place each media between which heat is exchanged are in direct contact with each and every one of a kind is Direct contact warmness exchanger,
- The place each media are separated through a wall by means of which warmness is transferred so that they on no account mix, oblique contact warmness exchanger.

1.1 Classification of Heat Exchangers

A variety of heat exchangers are used in industry and in their products. The objective of this chapter is to describe most of these heat exchangers in some detail using classification schemes. Starting with a definition, heat exchangers are classified according to transfer processes, number of fluids, and degree of surface compactness, construction features, flow arrangements, and heat transfer mechanisms. With a detailed classification in each category, the terminology associated with a variety of these exchangers is introduced and practical applications are outlined. A brief mention is also made of the differences in design procedure for the various types of heat exchangers.A heat exchanger is a device that is used to transfer thermal energy (enthalpy) between two or more fluids, between a solid surface and a fluid, or between solid particulates and a fluid, at different temperatures and in thermal contact. In heat exchangers, there are usually no external heat and work interactions. Typical applications involve heating or cooling of a fluid stream of concern and evaporation or condensation of single- or multi component fluid streams. In other applications, the objective may be to recover or reject heat, or sterilize, pasteurize, fractionate, distill, concentrate, crystallize, or control a process fluid. In a few heat exchangers, the fluids exchanging heat are in direct contact. In most heat exchangers, heat transfer between fluids takes place through a separating wall or into and out of a wall in a transient manner. In many heat exchangers, the fluids are separated by a heat transfer surface, and ideally they do not mix or leak. Such

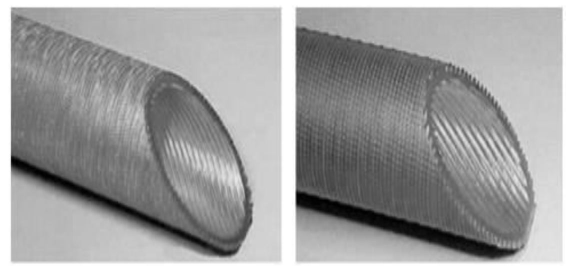
International Journal for Innovative
Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

exchangers are referred to as direct transfer type, or simply recuperates. In contrast, exchangers in which there is intermittent heat exchange between the hot and cold fluids via thermal energy storage and release through the exchanger surface or matrix are referred to as indirect transfer type, or simply regenerators. Such exchangers usually have fluid leakage from one fluid stream to the other, due to pressure differences and matrix rotation/valve switching.

Common examples of heat exchangers are shell-and-tube exchangers, automobile radiators, condensers, evaporators, air preheaters, and cooling towers. If no phase change occurs in any of the fluids in the exchanger, it is sometimes referred to as a sensible heat exchanger. There could be internal thermal energy sources in the exchangers, such as in electric heaters and nuclear fuel elements. Combustion and chemical reaction may take place within the exchanger, such as in boilers, fired heaters, and fluidized-bed exchangers.

1.2 MAJOR COMPONENTS OF SHELL-AND-TUBE EXCHANGERS

1.2.1 Tubes: Round tubes in various shapes are used in shell-and-tube exchangers. Most common are the tube bundles with straight and U-tubes used in process and power industry exchangers. However, sine-wave bend, J-shape, L-shape or hockey sticks, and inverted hockey sticks are used in advanced nuclear exchangers to accommodate large thermal expansion of the tubes. Some of the enhanced tube geometries used in shell-and-tube exchangers is shown in Fig. Serpentine, helical, and bayonet are other tube shapes that are used in shell-and-tube exchangers.



Turbo - EHP (a) Turbo - CDI (b)

fig.1.1 tube geometries used in shell and tube exchangers

1.2.2 Tube sheets

These are used to hold tubes at the ends. A tube sheet is generally a round metal plate with holes drilled through for the desired tube pattern, holes for the tie rods (which are used to space and hold plate base), grooves for the gaskets, and bolt holes for flanging to the shell and channel.

1.3 SHELL AND TUBE HEAT EXCHANGER

A Shell and tube heat exchanger is a class of heat exchanger. It is the most common type of heat exchanger in oil refineries and other large chemical processes. As its name implies, this type of heat exchanger consists of a shell (a large vessel) with a bundle of tubes inside it.

Theory and application

Two fluids, of different starting temperatures, flow through the heat exchanger. One flows through the tubes (the tube side) and the other flows outside the tubes but inside the shell (the shell side). Heat is transferred from one fluid to the other through the tube walls, either from tube side to shell side or vice versa. The fluids can be either liquids or gases on either the shell or the tube side. In order to transfer heat efficiently, a large heat transfer area should be used, so there are many tubes. In

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

this way, waste heat can be put to use. This is a great way to conserve energy.Heat exchangers with only one phase (liquid or gas) on each side can be called one-phase or single-phase heat exchangers. Two-phase heat exchangers can be used to heat a liquid to boil it into a gas(vapor), sometimes called boilers, or cool a vapor to condense it into a liquid (called condensers),with the phase change usually occurring on the shell side. Boilers in steam engine locomotives are typically large, usually cylindrically-shaped shell-and-tube heat exchangers. In large power plants with steam-driven turbines, shell-and-tube (see Condenser (steam turbine) condensers are used to condense the exhaust steam exiting the turbine into condensate water which can be recycled back to be turned into steam, possibly into a shell-and-tube type boiler.

1.3.1 Shell and tube heat exchanger design

There can be many variations on the shell and tube design. Typically, the ends of each tube are connected to plenums (sometimes called water boxes) through holes in tube sheets. The tubes may be straight or bent in the shape of a U, called U-tubes.

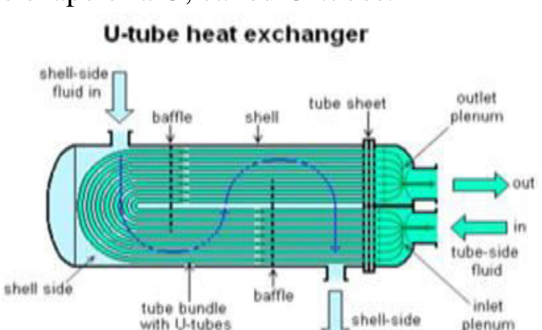


fig.1.2 U – tube heat exchanger

1.3.2 Selection of tube material

To be able to transfer heat well, the tube material should have good thermal conductivity. Because heat is transferred from a hot to a cold side through the tubes, there is a temperature difference through the width of the tubes. Because of the tendency of the tube material to thermally expand differently at various temperatures, thermal stresses occur during operation. This is in addition to any stress from high pressures from the fluids themselves. The tube material also should be compatible with both the shell and tube side fluids for long periods under the operating conditions (temperatures, pressures, pH, etc.) to minimize deterioration such as corrosion. All of these requirements call for careful selection of strong, thermally-conductive, corrosion-resistant, high quality tube materials, typically metals. Poor choice of tube material could result in a leak through a tube between the shell and tube sides causing fluid cross-contamination and possibly loss of pressure.

2.0 LITERATURE REVIEW

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 modeling. In addition, assessments of alternative relevant research reports are made to furnish more knowledge with a view to understand more on this research.

2.2 Purpose of Use of Helical Baffle:

A new type of baffle, called the helical baffle, provides further improvement. This type of baffle was first developed by Lutcha and Nemcansky. They investigated the flow field patterns produced by such

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

helical baffle geometry with different helix angles. They Determined that these waft patterns have been very close to the plug drift , which1 was anticipated to lower shell-side stress drop and to fortify heat transfer performance.1 Stehlik et al. compared heat transfer and pressure drop correction factors for a heat exchanger with an optimized segmental baffle based on the Bell–Delaware method, with those for a heat exchanger with helical baffles. Kral et al. discussed the performance of heat exchangers with helical baffles based on test results of various baffles geometries.

2.3Computational 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 modeling. This application might also build a digital prototype of the process or device before will also be observe to real-world physics and chemistry to the model, and the application 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 among the branches of fluid mechanics that makes use of numerical ways and algorithm can be used to resolve and analyse problems that involve fluid flows and in addition simulate the go with the flow over a piping, auto or machinery. Computers are used to participate in the millions of calculations required to simulate the interaction of fluids and gases with the complex surfaces used in engineering. Extra correct codes that may effectively and quickly simulate even complicated situations similar to supersonic and turbulent flows are on going research. Onwards the aerospace industry has integrated CFD systems into the design, R & D and manufacture of aircraft and jet engines. More lately the ways were utilized to the design of interior combustion engine, combustion chambers of gas turbine and furnaces also fluid flows and heat transfer in warmness exchanger (figure 1). Moreover, motor automobile manufactures now in many instances predict drag forces, under bonnet air flows and surrounding vehicle atmosphere with CFD. More and more CFD is becoming a vital factor within the design of commercial products and techniques.

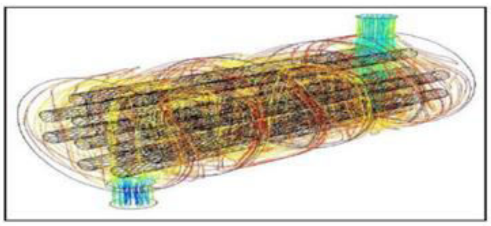


Fig 2.1 Fluid flow simulation for a shell andtube exchanger.

2.4 ANSYS:

Ansys is the finite aspect evaluation code broadly use in computer aided engineering (CAE) subject. ANSYS software aid us to construct laptop units of constitution, computer, accessories or process, practice running hundreds and other design criteria, be trained physical response equivalent to stress stage temperature distribution, stress and many others.

In Ansys following Basic step is followed:

•In the course of pre processing the geometry of the concern is defined. Quantity

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

occupied by using fluid is divided into discrete cells (the mesh). The bodily modeling is defined. This entails specifying the fluid behavior of the obstacle. For transient main issue boundary situation are also outlined.

•The simulation is begun and the equation is solved iteratively as constant state or transient.

•Finally a post procedure is used for the evaluation and visualization of the ensuing main issue.

3.0 COMPUTATIONAL FLUID DYNAMICS (CFD)

Computational Fluid Dynamics (CFD) is the simulation of fluids engineering systems using modeling (mathematical physical problem formulation) and numerical methods (discretization methods, solvers, numerical parameters, and grid generations, etc.). The process is as figure

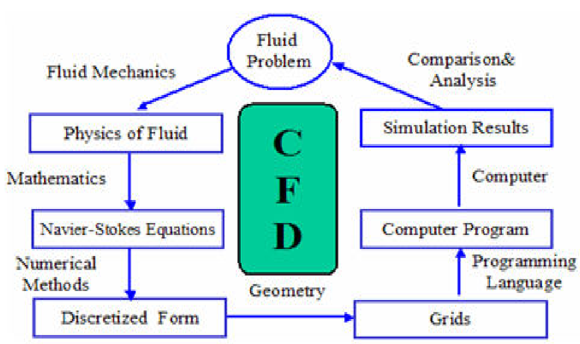


Fig. 3.1 Process of Computational Fluid Dynamics

Firstly, we have a fluid problem. To solve this problem, we should know the physical properties of fluid by using Fluid Mechanics. Then we can use mathematical equations to describe these physical properties. This is Navier-Stokes Equation and it is the governing equation of CFD. As the Navier-Stokes Equation is analytical, human can understand it and solve them on a piece of paper. But if we want to solve this equation by computer, we have to translate it to the discretized form. The translators are numerical discretization methods, such as Finite Difference, Finite Element, Finite Volume methods. Consequently, we also need to divide our whole problem domain into many small parts because our discretization is based on them. Then, we can write programs to solve them. The typical languages are Fortran and C. Normally the programs are run on workstations or supercomputers. At the end, we can get our simulation results. We can compare and analyze the simulation results with experiments and the real problem. If the results are not sufficient to solve the problem, we have to repeat the process until find satisfied solution. This is the process of CFD.

3.1 Importance of Computational Fluid Dynamics

There are three methods in study of Fluid: theory analysis, experiment and simulation (CFD). As a new method, CFD has many advantages compared to experiments. Please refer table 1

	Simulation (CFD)	Experiment
Cost	Cheap	Expensi ve
Time	Short	Long
Scale	Any	Small/Middle
Information	All	Measured Point
Repeatable	Yes	Some
Safety	Yes	Some Dangerous

Table 3.1 Comparison of Simulation and Experiment

3.2 MODEL FOR COMPUTATION

3.2.1 COMPUTATIONAL MODEL FOR HEAT EXCHANGER

The computational model of an experimental tested Shell and Tube Heat Exchanger(STHX) with 10 helix angle is

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

shown in fig. 2, 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 way of the interior side of the shell and the whole lot in the shell contained in the area. The inlet and out let of the domain are linked with the corresponding tubes.

3.2.2Navier-Stokes Equation:

It is named after Claude-Louis Navier and Gabriel Stokes , He described the motion of fluid components. Its also a essential equation being used by ANSYS and even in the reward task work. These equation come up from making use of 2nd regulation of newton to fluid movement, along facet the belief that the fluid stress is sum of a diffusing viscous time interval ,plus a stress time period. The derivation of the Navier Stokes equation starts with an application of 2nd law.

3.2.3 Geometry and Mesh:

The model is designed according to TEMA

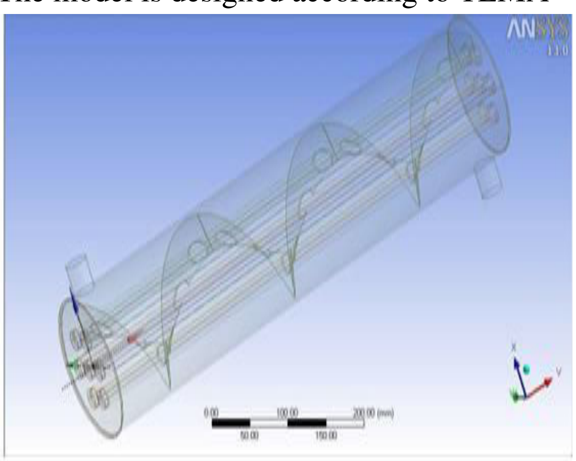


Fig 3.3 Isometric view of arrangement of baffles and tubes of shell and tube heat exchanger with baffle inclination.

Table :3.2 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

3.2.4 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 accurateand involvemuch less computation effort. Satisfactory manipulate on the hexahedral mesh close the wall floor allows for taking pictures the boundary layer gradient safely

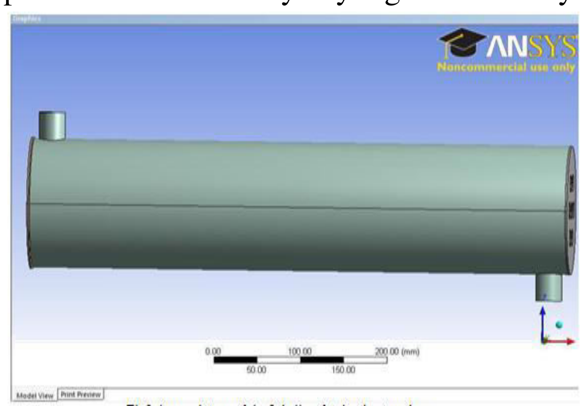


Fig3.4 complete model of shell and tube heat exchanger

3.2.5 Meshing :

Initially a rather coarser mesh is generated with 1.8 Million cells. This mesh contains combined cells (Tetra and Hexahedral cells) having each triangular and quadrilateral faces at the boundaries. Care is taken to use structured cells (Hexahedral) As so much as

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

possible, as a result the geometry is split into a couple of materials for utilising computerized methods to be had in the ANSYS meshing consumer.

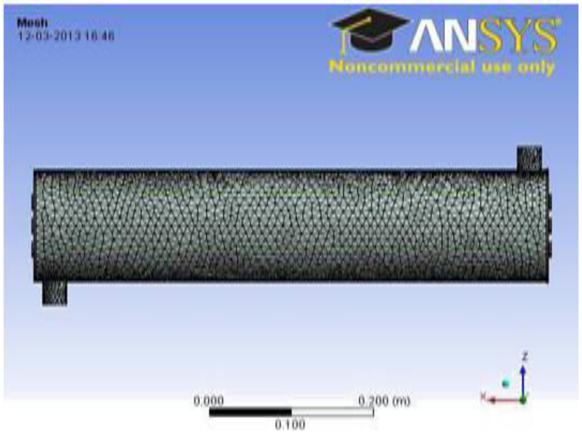


Fig 3.5 Meshing diagram of shell and tube heat exchange

3.2.6 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 mannequin option power calculation was once on and the viscous was set as ordinary k-e, regular wall perform(k-epsilon 2 eqn).

In mobilephone zone fluid water-liquid was once selected. Water-liquid and cupper, aluminum was once selected as substances for simulation. Boundary condition was once selected for inlet,outlet. In inlet and outlet 1kg/s velocity and temperature was once set at 353k.Throughout every tube zero.05kg/s velocity and 300k temperature used to be set. Mass float used to be chosen in each and every 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 .

3.2.7 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.

4.0 RESULTS

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

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 , epsilon are the part of scaled residual which have to converge in a specific region. For the continuity-velocity ,Y-velocity, Z-velocity , k, epsilon 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 answer used to be converged at a hundred and seventieth new release. The next figure indicates the residual plot for the above iterations

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

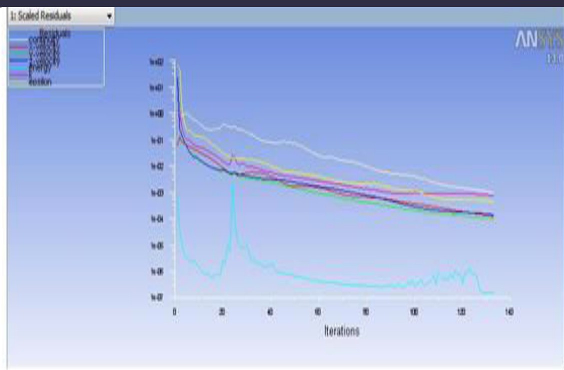


Figure 4.1 For Conversion 0^0 Baffle inclination after 170^{th} iteration

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

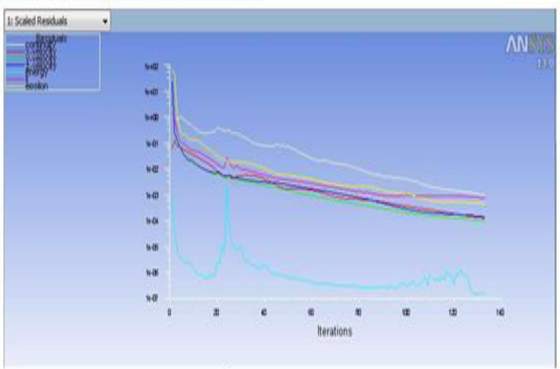


Figure 4.2 Converge simulation of 10^0 baffle inclination at 133th iteration.

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

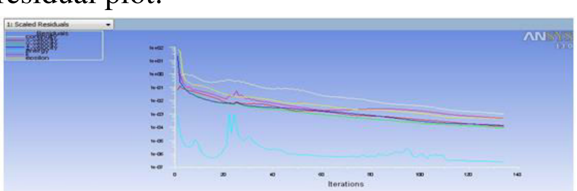


Figure 4.3 Convergence of 20^0 baffle inclination at 138^{th} iteration

4.2 Variation of Temperature:The temperature Contours plots across the cross section at extraordinary inclination of baffle along the length of warmth exchanger will supply an proposal of the glide in element. Three one-of-a-form plots of temperature profile are taken in evaluation with the baffle inclination at 00, 100 , 200 .

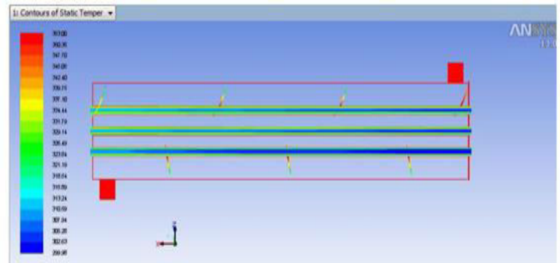


Figure 4.4 Temperature Distribution across the tube and shell

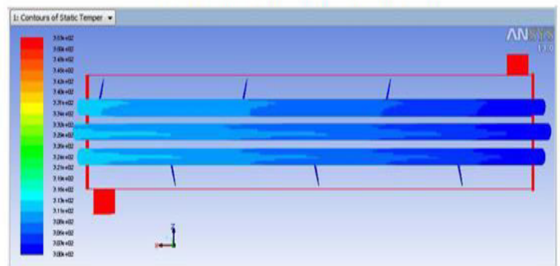


Figure 4.5 Temperature Distribution for 10^0 baffle inclination

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

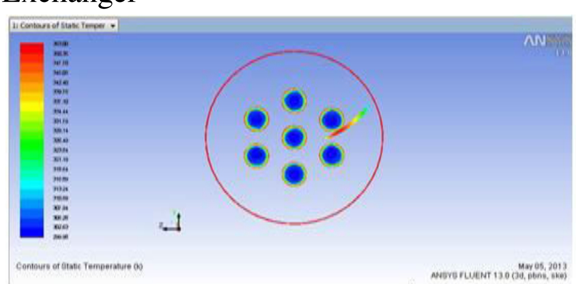


Figure 4.6 Temperature Distribution across Tube outlet in 0^0 baffle inclination

4.3 Variation Of Velocity:

Percent profile is examined to respect the go with the flow distribution throughout the move part at unique positions in warmness exchanger. Under in verify (12) (13) (14) is the speed profile of Shell and Tube warmness exchanger at precise Baffle inclination. It must be saved in intellect that the warmness exchanger is modeled on account that the aircraft symmetry. The % profile at inlet is identical for all three

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

inclination of baffle viewpoint i.e 1.44086 m/s. Outlet speed fluctuate tube to helical baffle and turbulence come up within the shell neighborhood.

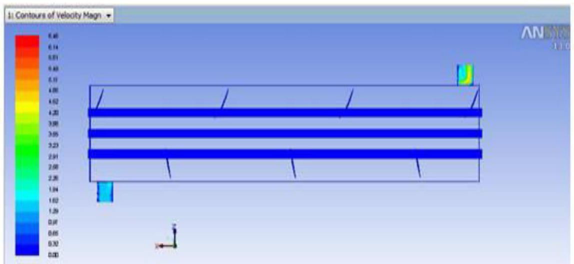


Figure 4.7 Velocity profile across the shell at 0 ^{0} baffle inclination.

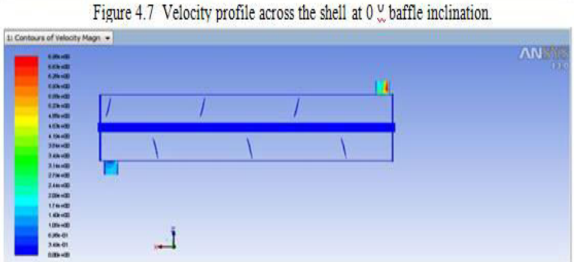


Figure 4.8 Velocity profile across the shell at 10^0 baffle inclination.

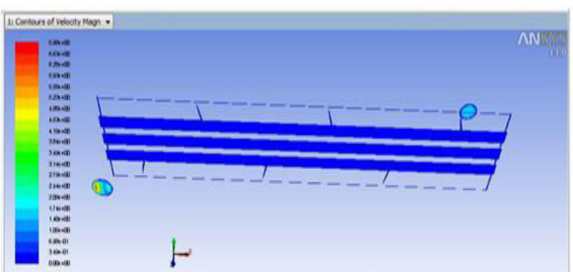


Figure 4.9 Velocity profile across the shell at 20^0 baffle inclination.

4.4 Variation Of Pressure: Pressure Distribution across the shell and tube warmth exchanger is given below in Fig. (14) (15) (sixteen) .With the expand in Baffle inclination perspective strain drop inside the shell is lower . Stress fluctuate generally from inlet to outlet. The contours of static pressure is shown in the entire figure to offer a detail idea

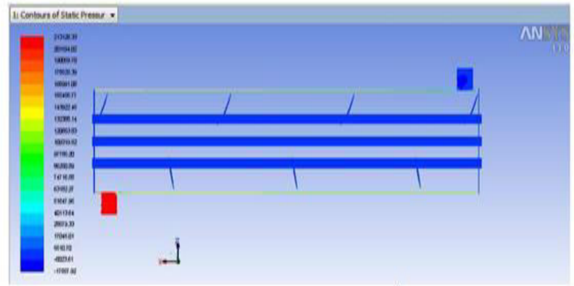


Figure 4.10 Pressure Distribution across the shell at 0^{0} baffle inclination.

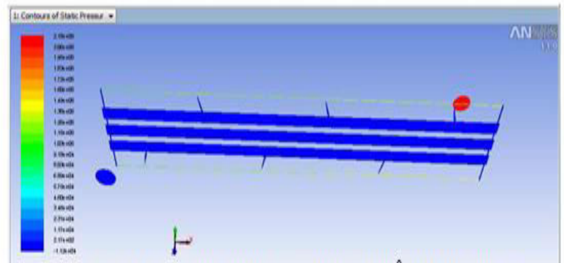


Figure 4.11 Pressure Distribution across the shell at 10^0 baffle inclination

Table 4.1 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

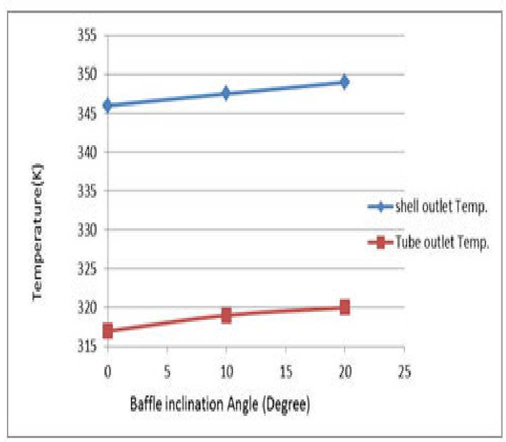


Figure 4.12 Plot of Baffle inclination angle vs Outlet Temperature of shell and tube side

It has been found that there is much effect of outlet temperature of shell side with increasing the baffle inclination angle from 00 to 200.

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

Table 4.2 for the Pressure Drop inside Shell

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

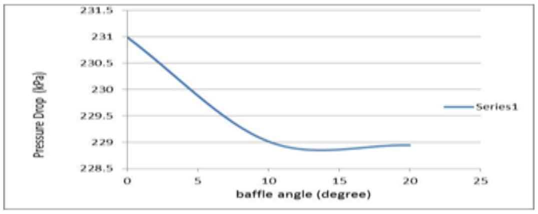


Figure 4.13 Plot of Baffle angle vs Pressure Drop

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. Therefore it may be observed with growing baffle inclination stress drop decreases, in order that it have an effect on in warmness switch cost which is extended

Table 4.3 for Velocity inside Shell

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

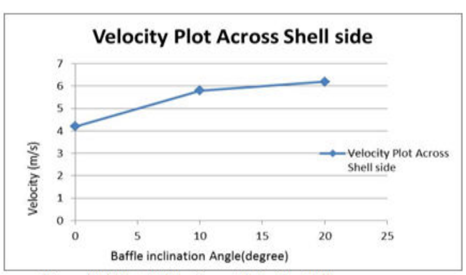


Figure 4.14 Plot of Velocity profile inside shell

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

4.5 Heat Transfer Rate

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

m=mass flow rate

C_p = Specific Heat of Water

ΔT = Temperature Difference Between Tube Side

Table 4.4 for Heat Transfer 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

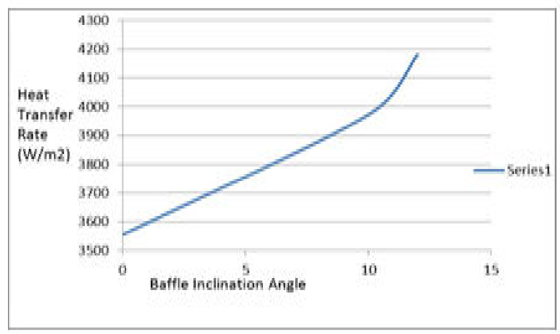


Figure 4.15 Heat Transfer Rate Along Tube side

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.

Table 4.5 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)
0^0	346	317	230.992	3554.7	4.2
10^0	347.5	319	229.015	3972.9	5.8
20^0	349	320	228.943	4182	6.2

The shell facet of a small shell-and-tube heat exchanger is modeled with ample detail to resolve the flow and temperature fields.

The stress drop decreases with increase in baffle inclination. The heat switch fee could be very sluggish in this mannequin so that it influence the outlet temperature of the shell and tube facet.

CONCLUSIONS

The warmth switch and drift distribution is discussed in element and proposed mannequin is in comparison with increasing baffle inclination attitude. The mannequin predicts the warmness switch and stress drop with an traditional error of 20%. The idea labored well in this geometry and meshing count on the outlet and inlet area the place rapid mixing and change in go with the flow course takes situation. As a consequence growth is predicted if the helical baffle used within the model must have complete contact with the outside of the shell, it is going to help in additional turbulence throughout shell aspect and the warmth switch cost will expand. If unique go with the flow fee is taken, it might be support to get better heat transfer and to get higher temperature change between inlet and outlet. In addition the mannequin has furnished the safe results with the aid of due to the fact the usual okay-e and usual wall operate mannequin, however this model over predicts the turbulence in regions with gigantic traditional strain. For that reason this mannequin can be extended with the aid of making use of Nusselt number and Reynolds stress mannequin, however with bigger computational concept. Moreover the enhance wall operate aren't use in this mission, however they may be able to be very valuable. The warmth switch fee is bad since lots of the fluid passes with out the interaction with baffles. Accordingly the design can be modified for higher warmth transfer in two methods both the decreasing the shell diameter, in order that it'll be a proper contact with the helical baffle or by means of growing the baffle so that baffles will likely be right contact with the shell. It's due to the fact the warmth switch subject is not utilized effectually. As a consequence the design can further be extended by way of growing cross-waft regions in this style of signifies that flow doesn’t stay parallel to the tubes. It's going to enable the outer shell fluid to have contact with the inner shell fluid, for this reason heat transfer fee will develop.

REFERENCES

1.Emerson, W.H., “ Shell-side pressure drop and heat transfer with turbulent flow in segmentally baffled shell-tube heat exchangers”, Int. J. Heat Mass Transfer 6 (1963), pp. 649–66.

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

3.Diaper, A.D. and Hesler, L.E., (1990), "Crossflow Pressure Drop and Flow Distributions within a Tube Bundle Using Computational Fluid Dynamic", Proc. 9th Proc. 9th Heat Transfer Conf., Israel, pp. 235-240.

4.Jian-Fei Zhang, Ya-Ling He, Wen-Quan Tao , " 3d numerical simulation of shell and tube heat exchanger with middle-overlapped helical baffle", a journal ,School of energy and power engineering,china.

5.Li, H., Kottke, "Effect of baffle spacing on pressure drop and local heat transfer in shell and tube heat exchangers for staggered tube arrangement", source book on Int. a.Heat Mass Transfer 41 (1998), 10, pp. 1303–1311.

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

7.Usman Ur Ehman , G¨oteborg, Sweden 2011, Master's Thesis 2011:09 on "Heat Transfer Optimization of Shell-and-Tube & Heat Exchanger through CFD".

8.Professor Sunilkumar Shinde, Mustansir Hatim Pancha / International Journal of Engineering Research and Applications (IJERA) ,"Comparative Thermal performance of shell and tube heat Exchanger with continuous helical baffle using ", Vol. 2, Issue4, July-August 2012.

9.KHAIRUN HASMADI OTHMAN, " CFD simulation of heat transfer in shell and tube heat exchnager", A thesis submitted in fulfillment for the award of the Degree of Bachelor in chemical Engineering (Gas Technology),April 2009.

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

11.Diaper, A.D. and Hesler, L.E., (1990), "Crossflow Pressure Drop and Flow Distributions within a Tube Bundle Using Computational Fluid Dynamic", Proc. 9th Proc. 9th Heat

12.Transfer Conf., Israel, pp. 235-240.

13.Jian-Fei Zhang, Ya-Ling He, Wen-Quan Tao , " 3d numerical simulation of shell and tube heat exchanger with middle-overlapped helical baffle", a journal ,School of energy and power engineering,china.

14.Li, H., Kottke, "Effect of baffle spacing on pressure drop and local heat transfer in shell and tube heat exchangers for staggered tube arrangement", source book on Int.a.Heat Mass Transfer 41 (1998), 10, pp. 1303–1311.

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