[Karthikeyan, 3(6): June 2016]
DOI- 10.5281/zenodo.56040

ISSN 2348 – 8034
Impact Factor- 4.022

Global Journal of Engineering Science and Researches

THERMAL ANALYSIS OF SHELL AND TUBE HEAT EXCHANGER USING CFD

Karthikeyan. D
M.E. Thermal Engineering, Department of Mechanical Engg, Muthayammal college of Engineering, Rasipuram-637 408, Namakkal Dt.

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 in solving simulation consists of modeling and meshing the basic geometry of shell and tube heat exchanger using CFD package ANSYS 15.0. 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 100 mm. The helix angle of helical baffle will be varied from 0 to 20. In simulation will show 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*: Thermal Analysis, Shell, heat exchanger etc.*

1. INTRODUCTION

One of the important processes in engineering is the heat exchange between flowing fluids, and many types of heat exchangers are employed in various types of installations, as petro-chemical plants, process industries, pressurized water reactor power plants, nuclear power stations, building heating, ventilating, and air-conditioning and refrigeration systems. As far as construction design is concerned, the tubular or shell and tube type heat exchangers are widely in use.

The shell-and-tube heat exchangers are still the most common type in use. They have larger heat transfer surface area-to-volume ratios than the most of common types of heat exchangers, and they are manufactured easily for a large variety of sizes and flow configurations. They can operate at high pressures, and their construction facilitates disassembly for periodic maintenance and cleaning. The shell-and-tube heat exchangers consist of a bundle of tubes enclosed within a cylindrical shell. One fluid flows through the tubes and a second fluid flows within the space between the tubes and the shell. Typical Shell-and-Tube heat exchanger is shown in Figure 1.1.

Heat exchangers in general and tubular heat exchangers in particular undergo deterioration in performance due to flow maldistribution. The common idealization in the basic tubular heat exchanger design theory is that the fluid is distributed uniformly at the inlet of the exchanger on each fluid side throughout the core. However, in practice, flow maldistribution is more common and significantly reduces the idealized heat exchanger performance. Flow maldistribution can be induced by the heat exchanger geometry, operating conditions (such as viscosity or density- induced maldistribution), multiphase flow, fouling phenomena, etc. Geometry-induced flow maldistribution can be classified into gross flow maldistribution, passage-to-passage flow maldistribution and manifold-induced flow maldistribution.

The flows in shell-and-tube heat exchangers have only been investigated analytically [1,2, and 3] due to their complexity. Ranjit Kumar Sahoo , Wilfried Roetzel [2] and Chakkrit Na Ranong[1] carried out an analysis of the effect of maldistribution on the thermal performance and the temperature distribution in shell and tube heat exchanger using a finite difference method.

Mueller and Chiou [17] summerised various types of flow maldistribution in heat exchangers and discussed the reason leading to flow maldistibution. . Ranganayakulu and Seetharamu [19] carried out an analysis of the effects of inlet fluid flow nonuniformity on the thermal performance and pressure drop in

ISSN 2348 – 8034
Impact Factor- 4.022

crossflow plate-fin heat exchangers by using a finite element method.

Lalot and Florent [4] used the computer code STAR-CD to study the gross flow maldistribution in an electrical heater. They found that reverse flows would occur for the poor header design and the perforated grid can improve the fluid flow distribution. However, few authors studied the fluid flow maldistribution using the computational fluid dynamics (CFD) simulation technique, especially the effects of the configuration of header and distributor on the flow distribution in plate-fin heat exchangers. CFD simulation technique can provide the flexibility to construct computational models that are easily adapted to a wide variety of physical conditions without constructing a large-scale prototype or expensive test rigs. Therefore, CFD can provide an effective platform where various design options can be tested and an optimal design can be determined at a relatively low cost.

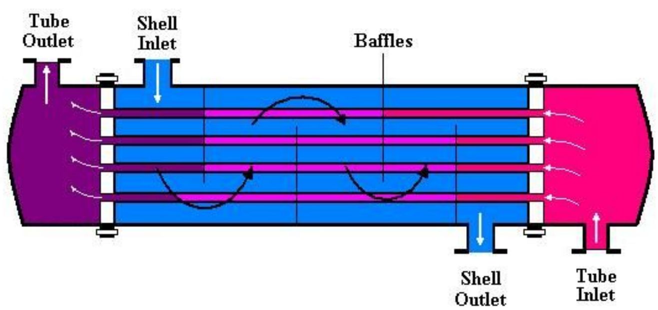


Fig 1.1 Shell-and-tube heat exchanger

1.1 heat exchanger

A heat exchanger is a device built for efficient heat transfer from fluid one to another, whether the fluids are separated by a solid wall so that they never mix, or the fluids are directly contacted. They are widely used in petroleum refineries, chemical plants, petrochemical plants, natural gas processing, refrigeration, power plants, air conditioning and space heating. One common example of a heat exchanger is the radiator in a car, in which a hot engine-cooling fluid, like antifreeze, transfers heat to air flowing through the radiator.

[Karthikeyan, 3(6): June 2016]
DOI- 10.5281/zenodo.56040
ISSN 2348 – 8034
Impact Factor- 4.022

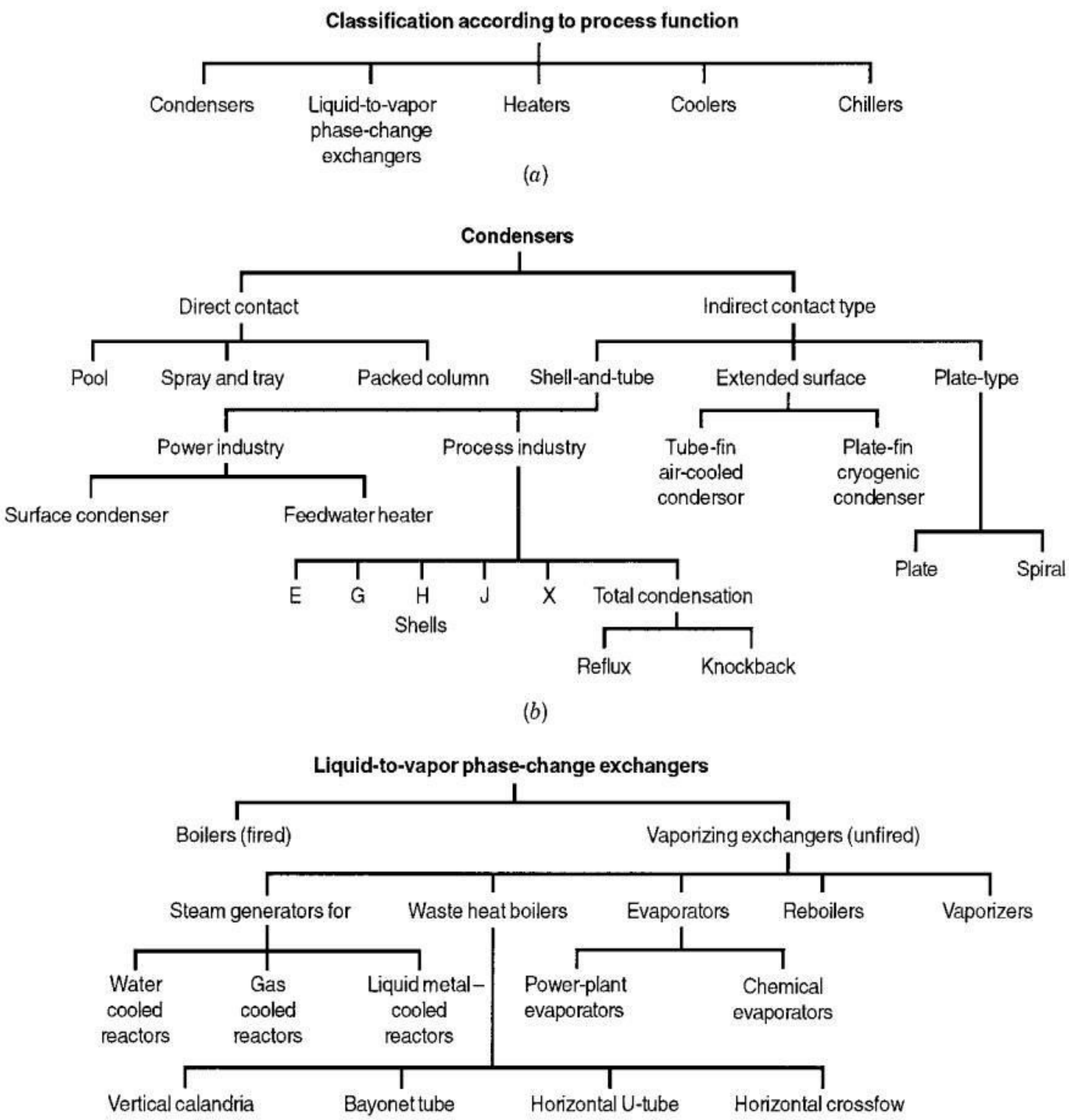


Fig 1.2 Classification of heat exchangers

1.2 Tubular heat exchangers

Tubular heat exchangers are generally built of circular tubes, although elliptical, rectangular or round/flat twisted tubes have also been used in some applications. There is considerable flexibility in design because the core geometry can be varied easily by changing the tube diameter, length, and arrangement. Tubular exchangers can be designed for high pressures relative to environment and high-pressure differences between the fluids. Tubular exchangers are used primarily for liquid-to-liquid and liquid to phase change (condensing or evaporating) heat transfer applications. There are used for gas-to liquid and gas-to-gas heat transfer applications primarily when the operating temperature and /or pressure is very high or fouling is a severe problem on at least one fluid side and no other types of exchangers work.

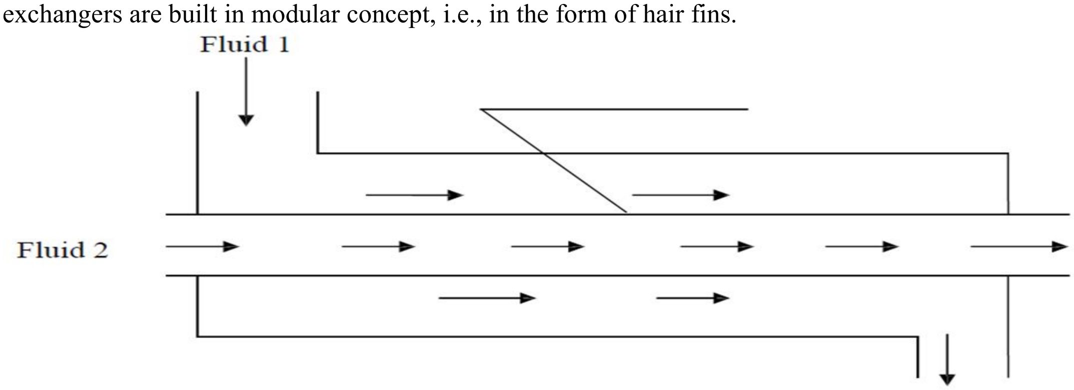
These tubular exchangers may be classified as shell-and-tube, double-pipe, and spiral tube exchangers. There are all prime surface exchangers except for exchangers having fins outside/inside tubes.

ISSN 2348 – 8034
Impact Factor- 4.022

1.2.1 Double pipe heat exchanger:

A typical double-pipe heat exchanger consists of one pipe placed concentrically in side another of larger diameter with appropriate fittings to direct the flow from one section to the next, as shown in figure (1.2). Double-pipe heat exchangers can be arranged in various series and parallel arrangements to meet pressure drop and mean temperature difference requirements. The major use of double-pipes exchangers is for sensible heating or cooling of process fluids where small heat transfer areas (to 50 $m^{2)}$ are required. This configuration is also very suitable.

When one or both fluids is at high pressure. The major disadvantage is that double-pipe heat exchangers are bulky and expensive per unit transfer surface. Inner tube being may be single tube or multi-tubes Fig. (1.3). If heat transfer coefficient is poor in annulus, axially finned inner tube (or tubes) can be used. Double-pipe heat exchangers are built in modular concept, i.e., in the form of hair fins.



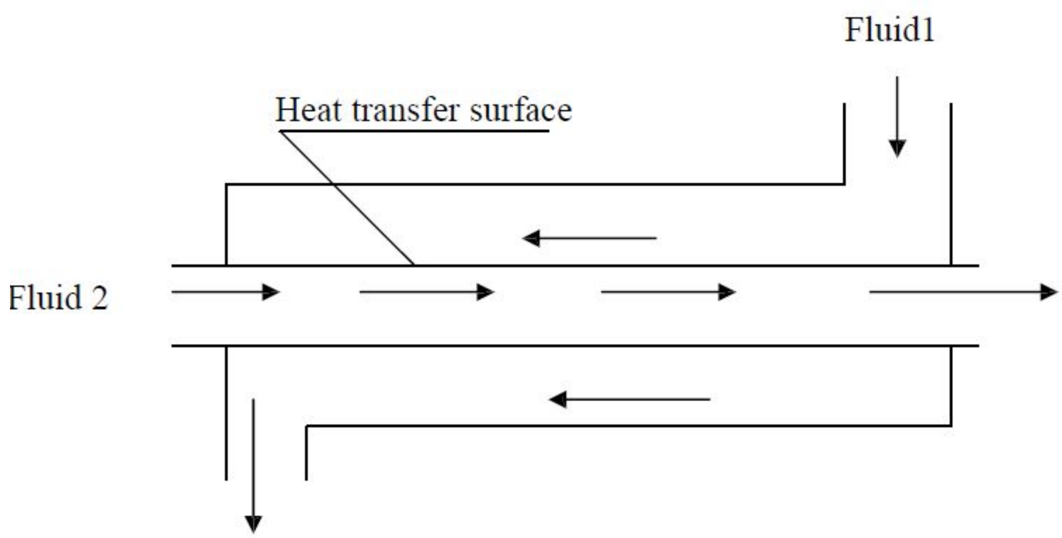


Fig 1.3: Double pipe heat exchanger

ISSN 2348 – 8034
Impact Factor- 4.022

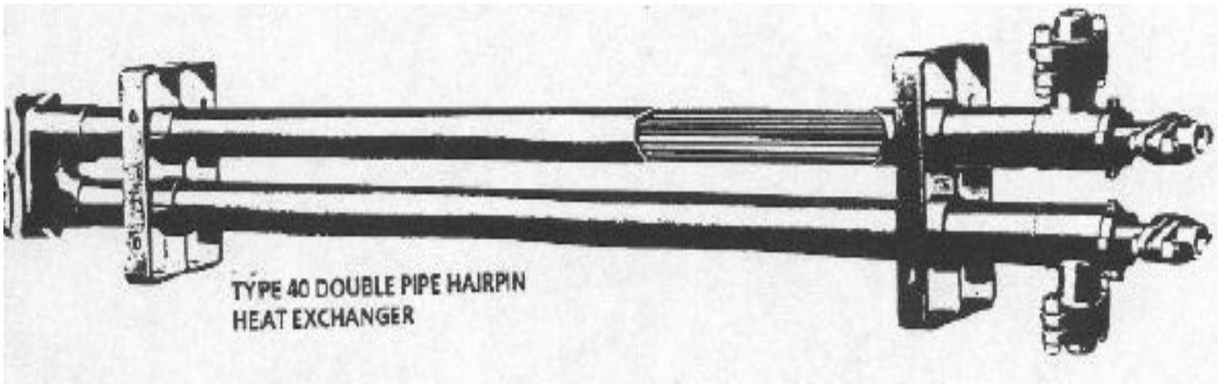


Fig 1.4: Double pipe hair-pin heat exchanger with cross sectional view and return bend housing

2. CHAPTER – 2
LITERATURE REVIEW

2.1. Introduction

The literature reviewed in this chapter can be broadly classified under three categories. The first part of the survey deals with the analytical solution for maldistribution in shell and tube heat exchanger. Second part of the survey deals with the experimental and CFD analysis of maldistribution in heat exchanger, and third part of the survey deals with the analysis of maldistribution in plate heat exchanger.

2.2. Analysis of maldistribution in shell and tube heat exchanger by analytical method

Wilfried Roetzel, Chakkrit Na Ranong., [1] calculated the axial temperature profiles in a shell and tube heat exchanger by numerically for given maldistributions on the tube side. For comparison the same maldistributions are handled with the parabolic and hyperbolic dispersion model with fitted values for the axial dispersion coefficient and third sound wave velocity. The analytical results clearly demonstrate that the hyperbolic model is better suited to describe the steady state axial temperature profiles. For a global consideration of a heat exchanger with maldistribution the parabolic model is satisfactory. The parameter Pepar depends on the NTU of the maldisributed flow stream and on the NTU of the transversely mixed flow stream which makes the model difficult to handle. The hyperbolic model predicts the axial temperature profiles correctly, especially temperature jumps and positive slopes. The third sound Mach number M characterizes the type of flow maldistribution and is independent of both NTUs. For a given type of relative flow maldistribution Pehyp is proportional to the NTU of the maldistributed flow stream but does not depend on the other NTU:

Sahoo, R.K., and Wilfried Roetzel., [2] derived The fundamental equations of hyperbolic model and its boundary conditions in terms of cross-sectional mean temperature from the basic equations of heat exchanger The traditional parabolic model and the proposed hyperbolic model which includes the parabolic model as a special case can be used for dispersive flux formulation. Instead of using the heuristic approach of parabolic or hyperbolic formulation, these models can be quantitatively derived from the axial temperature profiles of heat exchangers. In this paper both the models are derived for a shell-and-tube heat exchanger with pure maldistribution (without back mixing) in tube side flow and the plug flow on the shell side. The Mach number and the boundary condition which plays a key role in the hyperbolic dispersion have been derived and compared with previous investigation. It is observed that the hyperbolic model is the best suited one as it compares well with the actual calculations. This establishes the hyperbolic model and its boundary conditions.

Wilfried Roetzel, and Chakkrit Na Ranong.,[3] tested and compared the newer hyperbolic dispersion model and parabolic model considering the processes with pure maldistribution (with out back mixing) on the tube side of a shell and tube heat exchanger and plug flow on shell side . The boundary conditions of the model equations are discussed in detail for the steady state and equations of the axial temperature profiles are provided in the programmable form. For the hyperbolic model simple relationships between the model parameters are derived. Considering the transient adiabatic processes in the tube bundle a concept for the experimental determination of the model parameter M, the third sound Mach number, is developed. Authors concluded that for an overall consideration of a heat exchanger with maldistribution the parabolic model is satisfactory. The parameter Pe_{par} depend on both NTUs of the heat exchanger which makes the model difficult to handle. The advantage of the parabolic model is that only the only one parameter is needed. The hyperbolic model is superior to the parabolic model because it predicts the axial temperature profiles correctly, especially temperature jumps and (positive) slopes.

Yimin xuan, wilfried roetzel, [4] Applied the dispersion model is to the description of the effects of shell and tube side flow maldistribution. By means of this model, an efficient and versatile method of predicting transient response of multi pass shell and tube heat exchangers is developed. The method allows for effect of maldistribution on transient process, influence of heat capacities of fluids and solid components, arbitrary inlet temperature variations and step disturbances of flow rates. General forms of initial conditions and two different flow arrangements are considered. A general form of the solution for steady-state and dynamic simulation is derived. Temperature profiles are determined with numerical inversion of the Laplace transform. Some examples are calculated and the effect of maldistribution is discussed. Flow maldistribution hinders transient responses to any inlet changes and decreases thermal effectiveness of heat exchangers. Its effect becomes more remarkable with increasing NTC'. The Peclet number has been used to quantitatively describe this kind of effect. The calculation has shown that the dispersion model should be applied instead of the plug-flow model if Pe < 55.

Danckwerts, p. v., [5] When a fluid flows through a vessel at a constant rate, either "piston-flow" or perfect mixing is usually assumed. In practice many systems do not conform to either of these assumptions, so that calculations based on them may be in accurate. It is explained how distribution functions for residence-times can be defined and measured for actual systems. Open and packed tubes are discussed as systems about which predictions can be made. The use of the distribution functions is illustrated by showing how they can be used to calculate the efficiencies of reactors and blenders. It is shown how models may be used to predict the distribution of residence-times in large systems.

3. FLOW MALDISTRIBUTION IN HEAT EXCHANGERS

3.1 INTRODUCTION

One of the common assumptions in basic heat exchanger theory is that fluid be distributed uniformly at the inlet of the exchanger on each fluid side and throughout the core. However, in practice, flow maldistribution is more common and significantly reduces the desired heat exchanger performance. Still this influence may be negligible in many cases, and the goal of uniform flow through the exchanger is met reasonably well for performance analysis and design purposes.

Flow maldistribution is defined as of the mass flow rate on one or both sides in any of the heat exchanger ports and/or in the heat exchanger core. The term ideal fluid flow passage/header/heat exchanger would denote conditions of uniform mass flow distribution through an exchanger core.

Flow maldistribution can be induced by heat exchanger geometry (mechanical design features such as the basic geometry, manufacturing imperfections, and tolerances), and heat exchanger operating conditions (e.g., viscosity-or density-induced maldistribution, and fouling phenomena).geometry-induced flow maldistribution can be classified into (1) gross flow maldistribution, (2) passage-to-passage flow

ISSN 2348 – 8034
Impact Factor- 4.022

maldistribution, and (3) manifold-induced flow maldistribution. The most important flow maldistribution and associated flow instability.

3.2 GEOMETRY-INDUCED FLOW MALDISTRIBUTION

One class of flow maldistribution, which is a result of geometrically nonideal fluid flow passages or nonideal exchanger inlet/outlet header/tank/manifold/nozzle design, is referred to as geometry-induced flow maldistribution. This type maldistribution is closely related to heat exchanger construction and fabrication (e.g., header design, heat exchanger core fabrication including brazing in compact heat exchangers).this maldistribution is peculiar to a particular heat exchanger in question and cannot be influenced significantly by modifying operating conditions. Geometry-induced flow maldistribution is related to mechanical design-induced flow nonuniformities such as (1) entry conditions, (2) by pass and leakage streams, (3) fabrication tolerances, (4) shallow bundle effects, and (5) general equipment and exchanger system effects.

The most important causes of flow non uniformities can be divided roughly into three main groups of maldistribution effects: gross flow maldistribution (at the inlet face of the exchanger), passage-to-passage flow maldistribution (non uniform flow in neighboring flow passages), and manifold-induced flow maldistribution (due to inlet/outlet manifold/header design).

3.2.1 Gross flow maldistribution:

The major feature of gross flow maldistribution is that nonuniform flow occurs at the macroscopic level (due to poor header design or blockage of some flow passages during manufacturing, including brazing or operation). The gross flow maldistribution does not depend on the local heat transfer surface geometry. This class of flow maldistribution may cause a significant increase in the exchanger pressure drop, and some reduction in the heat transfer rate.

To predict the magnitude of these effects for some simple exchanger flow arrangements, the non-uniformity will be modeled as one-or two-dimensional as follows, with some specific results.

3.2.2 Passage-to-Passage flow maldistribution:

Compact heat exchangers with uninterrupted (continuous) flow passages, while design for nonfouling applications, are highly susceptible to passage-to-passage flow maldistribution. That is because the neighboring passages are geometrically never identical, due to imperfect manufacturing processes.

It is especially difficult to control the passage size precisely when small dimensions are involved. Since differently sized and shaped passages exhibit different flow resistances and the flow seeks the path of least resistance, a non-uniform flow through the matrix results. This phenomenon usually causes a slight reduction in pressure drop, while the reduction in heat transfer rate may be significant compared to that for nominal size passages.

3.2.3 Manifold-induced flow maldistribution:

Whereas manifolds are integral in plate heat exchangers due to construction features, manifolds are common and attached separately in many other applications. In The PHEs, The fluids enter and exit the manifolds laterally and flow with in the core axially; here the axial direction is defined as the main direction of fluid flow with in the PHE passages. In other applications, the fluids enter and exit the core also axially, or a combination of axial lateral entry and exit. In PHEs, the manifolds are of two basic types: deviding flow and combining flow. In deviding-flow manifolds, a fluid enters laterally and exits the manifold axially. The velocity with in the manifold, parallel to manifold axis, varies from the inlet velocity to zero value. Conversely, in combining-flow manifolds, fluid enters from the PHE core and exits at the end of the manifold varying from zero to the out let velocity.

 ISSN 2348 – 8034
Impact Factor- 4.022

3.3. Governing equations in shell-and-tube heat exchanger with maldistribution in tube side:

A shell and tube heat exchanger with pure axial plug flow on the shell side and maldistribution on the tube side shown in Fig. 3.1

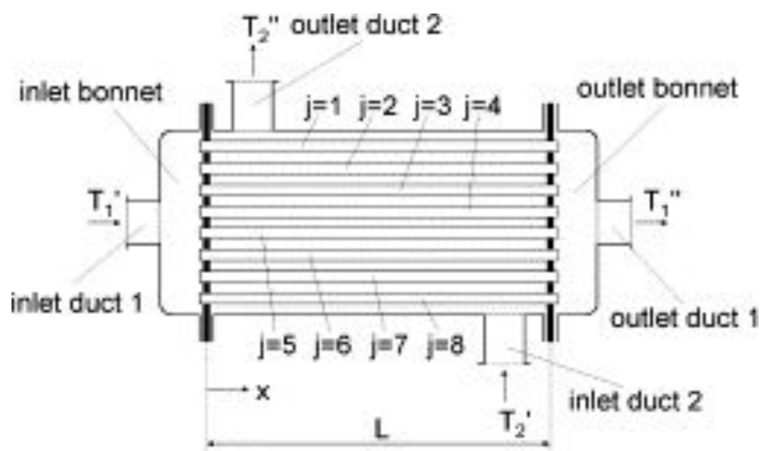


Fig.3.1. Schematic of shell-and-tube heat exchanger with tube channels (j=1,…,8).

In the shell-and-tube heat exchanger, a constant velocity of tube side fluid is usually assumed in all the tubes. Actually, the velocity may vary from tube to tube due to disadvantageous geometry of the inlet duct, outlet duct and bonnets. On the shell side, plug flow is assumed. For simplicity the overall heat transfer coef- ficient is assumed to be uniform. This type of steady state maldistribution due to flow non-uniformity in tube side gives dispersion. Under the realistic assumption of plug flow inside each of the N tubes, the tube side equation given by Ranjit Kumar Sahoo, Wilfried Roetze,l.[2] is,

$$\frac{d\theta_{1j}}{d\xi} + \frac{kA/N}{\left(\dot{W}_1/N\right)\left(W_{1J}/\overline{W}_1\right)}(\theta_{1J} - \theta_2) = 0. \tag{3.1}$$

This equation may be rewritten as

$$\frac{w_{1j}}{\overline{w}_1}\frac{d\theta_{1j}}{d\xi} + NTU_1(\theta_{1J} - \theta_2) = 0. \tag{3.2}$$

The second term of Eq. (3.2) represents the local heat flux at the jth tube due to shell side fluid.

Summing up all N equations and dividing by N gives

$$\frac{1}{N}\sum_{J=1}^{N}\frac{d}{d\xi}\left(\frac{w_{1j}}{\overline{w}_1}\theta_{1j}\right) + NTU_1\left(\frac{1}{N}\sum_{J=1}^{N}\theta_{1J} - \theta_2\right) = 0, \tag{3.3}$$

Where the second term represents the average local heat flux associated with all the tube side flow and this must be equal to the lateral flux associated with the shell side fluid. Thus the shell side energy equation is given as

[Karthikeyan, 3(6): June 2016] ISSN 2348 – 8034
DOI- 10.5281/zenodo.56040 Impact Factor- 4.022

$$\frac{d\theta_2}{d\xi} + NTU_2\left(\frac{1}{N}\sum_{J=1}^{N}\theta_{1J} - \theta_2\right) = 0. \quad (3.4)$$

4. CHAPTER 4
OVERVIEW OF FLUENT

4.1 INTRODUCTION

The availability of affordable high performance computing hardware and the introduction of user-friendly interfaces have led to the development of commercial CFD packages. Several general-purpose CFD packages have been published in past decade. Prominent among them are: PHONICS [21], FLUENT [12], SRAT-CD [19], CFX [20], FLOW -3D and COMPACT. Most
of them are based on the finite volume method.

Among these as mentioned FLUENT is very leading engineering software provides a state of the art computer program for modeling fluid flow and heat transfer in complex geometries. FLUENT provides complete mesh flexibility, solving the flow problems with unstructured meshes that can be generated about complex geometries with relative ease. Supported mesh types include 2D triangular/ quadrilateral, 3D tetrahedral/ hexahedral/ pyramid/ wedge, and mixed (hybrid) meshes.

Fluent also allows refining or coarsening the required mesh based on the flow solution. FLUENT also allows refining or coarsening the required mesh based on the flow solution. FLUENT consists of two main parts. First part is called ICEM CFD and second part is called FLUENT the solver.

One can generate the required geometry and grid using ICEM CFD. Also one can use T grid to generate a triangular, tetrahedral or hybrid volume mesh from the existing boundary mesh.

Once a grid has been read into FLUENT, all refining operations are performed within the solver. These include the setting boundary conditions, defining fluid properties, executing the solution, refining the grid viewing and post processing the results.

4.2 ICEM CFD

Take care to insure that you are in the correct directory. Fire up ICEM CFD from the command prompt by typing ICEM CFD *filename*

Generate a grid

There are two ways of generating a mesh. ICEM CFD calls them 'top down' or 'bottom-up' in the user manuals. These instructions are bottom-up. You will create vertices upon which the edges will be built upon. Connecting edges will create a face. Connecting faces will create a volume (3D). Once the face or volume is created, a mesh can be generated on it. For this example, we will stick to 2D, node -> edge ->face-> mesh. Remember to save and save often.

A mesh can now be created on the face. Under the OPERATION button, click on Mesh Command button. Where the word GEOMETRY used to be, the word MESH will appear with five buttons: *boundary-layer, edge, face, volume* and *group.* You want to mesh the face that you have just created, so click on *face.* Click on the top left button in the FACE menu area, the button is called: Mesh Faces. This will cause the Mesh Faces floating window to pop up. Let everything stay at its default, select the face and click Apply. ICEM CFD may hesitate while it's thinking and then you will see the mesh in yellow. You can play around with mesh spacing but keep the

ISSN 2348 – 8034
Impact Factor- 4.022

elements and type at ICEM CFDs default setting.

Boundary Conditions:

You can set or change the boundary conditions in Fluent but you can also do it in ICEM CFD, in fact, it's a little bit easier. Up in the OPERATIONS menu; click on the Zones button. Under the word ZONES two buttons will appear: Specify Boundary Types and Specify Continuum. Click on the Specify Boundary Types button. A floating window called Specify Boundary Types will appear

Change the Entity pop down menu to edges. Select the edge that will be the velocity inlet and under the Type pop down menu choose Velocity Inlet. It is recommended that you label the different edges. This will help you keep track of them in the Fluent output reports. The labels must be one word, i.e. no spaces or tabs. To finish creating the BC click *Apply.* Now select the edge that will be the outlet and choose Outflow. The top and bottom edges of the airfoil and control volume are Walls. There is a list at the top of this window that should reflect the two BC's that you have created.

4.3 NUMERICAL SOLVING TECHNIQUE

FLUENT in general solve the governing integral equations for the conservation of mass and momentum, and (when appropriate) for energy and other scalars such as turbulence and chemical species. Usually control volume based technique is used that consists of:

Division of the domain into discrete control volumes using a computational grid.

Integration of the governing equations on the individual control volumes to construct algebraic equations for the discrete dependent variables ("unknowns") such as velocities, pressure, temperature and conserved scalars.

Linearization of the discretized equations and solution of the resultant linear equation system to yield updated values of the dependent variables.

Solution Methodology

FLUENT allows choosing either of two numerical methods:

Segregated solver

Coupled solver

The two numerical methods employ a similar discretization process (finite volume), but the approach used to linearize and solve the discretized equation is different**.**

Segregated Method

Using this approach, the governing equations are solved sequentially (i.e., segregated from one another). Because the governing equations are non-linear (and coupled), several iterations of the solution loop must be performed before a converged solution is obtained. Each iteration consists of the steps illustrated below:

1. Fluid properties are updated, based on the current solution. (if the calculation has just begun, the fluid properties will be updated based on the initialized solution).

2. The *u, v and w* momentum equations are each solved in turn using current values for pressure and face

ISSN 2348 – 8034
Impact Factor- 4.022

mass fluxes, in order to update the velocity field.

3. Since the velocities obtained in step 2 may not satisfy the continuity equation locally, a "Poisson-type" equation for the pressure correction is derived from the continuity equation is then solved to obtain the necessary corrections to the pressure and velocity fields and the face mass fluxes such that continuity is
4. Where appropriate, equations for scalars such as turbulence, energy, species and radiation are solved using the previously updated values of the other variables.
5. When inter-phase coupling is to be included, the source terms in the appropriate continuous phase equations may be updated with a discrete phase trajectory calculation.

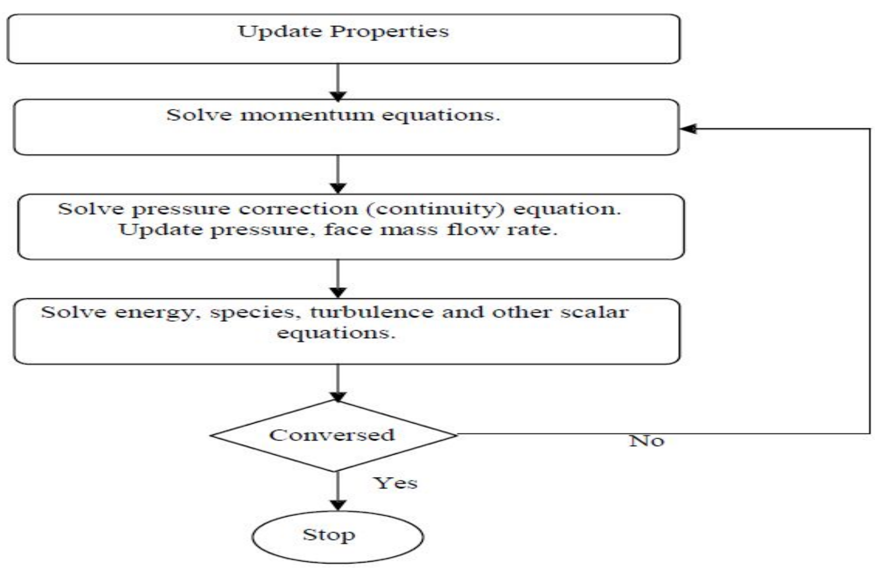
These steps are continued until the convergence criteria are met.

Coupled Method

The coupled solver solves the governing equations of continuity, momentum and (where appropriate) energy and species transport simultaneously (i.e., coupled together). Governing equations for additional scalar will be solved sequentially (i.e., segregated from one another and from the coupled set) using the procedure described for the segregated solver. Because the governing equations are non-linear (and coupled), several iterations of the solution loop must be performed before a converged solution is obtained. Each iteration consists of the steps outlined below:

1. Fluid properties are updated, based on the current solution. (If the calculation has just begun, the fluid properties will be updated based on the initialized solution).
2. The continuity, momentum and (where appropriate) energy and species equations are solved simultaneously.
3. Where appropriate, equations for scalars such as turbulence and radiation are solved using the previously updated values of the other variables.
4. When interphase coupling is to be included, the source terms in the appropriate continuous phase equations may be updated with a discrete phase trajectory calculation.
5. A check for convergence of the equation set is made. These steps are continued until the convergence criteria are met.

Figure4.1: Overview of the Segregated Solution Method



Linearization: Implicit & Explicit

In both the segregated and coupled solution methods the discrete, non-linear governing equations are linearized to produce a system of equations for the dependent variables in every computational cell. The resultant linear system is then solved to yield an updated flow-field solution.

The manner in which the governing equations are linearized may take an "implicit" or "explicit" form with respect to the dependent variables (or set of variables) of interest. By implicit or explicit we mean the following:

Implicit: for a given variable, the unknown value in each cell is computed using a relation that includes both existing and unknown values from neighbouring cells.

Therefore each unknown will appear in more than one equation in the system, and these equations must be solved simultaneously to give the unknown quantities.

Explicit: for a given variable, the unknown value in each cell is computed using a relation that includes only existing values. Therefore each unknown will appear in only one equation in the system, and the equations for the unknown value in each cell can be solved one at a time to give the unknown quantities.

In the segregated solution method each discrete governing equation is linearized only by implicitly with respect to that equations dependent variable. This will result in a system of linear equations with one equation for each cell in the domain. For example, the x-momentum equation is linearized to produce a system of equations in which u velocity is the unknown. Simultaneous solution of this equation system (using the scalar AMG solver) yields an updated u velocity field.

In the coupled solution method user have a choice of using either an implicit or explicit Linearization of the governing equations. Governing equations for additional scalars that are solved segregated from the coupled set, such as for turbulence, radiation etc., linearized and solved implicitly using the same procedures as in the segregated solution method.

If one choose the implicit option of the coupled solver, each equation in the coupled set of governing equations is linearized implicitly with respect to all dependent variables in the set. This will result in a system of linear equations with N equations for each cell in the domain, where N is the number of coupled equations in the set. For example, Linearization of the coupled continuity, x, y, z momentum and energy equation set will produce a system of equations in which $, u, v, w$ *and* T are the unknowns. Simultaneous solution of this equation system (using the block AMG solver) yields at once updated pressure, u, v, w velocity and temperature fields.

If one chooses the explicit option of the coupled solver, each equation in the coupled set of governing equations is linearized explicitly. As in the implicit option, this will result in a system of equations with N equations for each cell in the domain. And likewise, all dependent variables in the set will be updated at once. However, this system of equations is explicit in the unknown dependent variables. For example, the x-momentum equation is written such that the updated x velocity is a function of existing values of the field variables. Because of this, a linear equation solver is not needed. Instead, the solution is updated using a multi stage (Runge-kutta) solver. In summary, the coupled explicit approach solves for all variables ($, u, v, w, T$) one cell at a time.

Discretization

Fluent uses a control volume based technique to convert the governing equations to algebraic equations that can be solved numerically. This control volume technique consists of integrating the governing equations about each control volume, yielding discrete equations that conserve each quantity on a control volume basis.

Initializing the Solution

As because solving is done by iterative method, user must provide FLUENT with an initial "guess" for the solution flow field. In many cases, one must take extra care to provide an initial solution that will allow

[Karthikeyan, 3(6): June 2016]
DOI- 10.5281/zenodo.56040
ISSN 2348 – 8034
Impact Factor- 4.022

the desired final solution to be attained.

There are two methods for initializing the solution:

Initialize the entire flow field (in all cells).

Patch values or functions for selected flow variables in selected cell zones or “registers” of cells.

Convergence and Stability

Under Relaxation

Because of the nonlinearity of the equation set being solved by FLUENT, it is necessary to control the change of . This is typically achieved by under relaxation, which reduces the change of produced during each iteration.

By controlling relaxation factor one can avoid the sudden divergence in the solving process.

Monitoring Residuals

During the solution process one can monitor the convergence dynamically by checking residuals, statistics, force values, surface integrals, and volume integrals.

Judging the convergence

At the end of each iteration, the residual sum for each of the conserved variables is computed. On a computer with infinite precision, these residuals will go to zero as the solution converges. On an actual computer, the residuals decay to some small value (“round off”) and then stop changing (“level out”). For “single precision” computations (the default for workstations and most computers), residuals can drop as many as six orders of magnitude before hitting round off. Double precision residuals can drop up to twelve orders of magnitude. Residual definitions that are useful for one class of problem are sometimes misleading for other classes of problems. Therefore it is a good idea to judge convergence not only by examining residual levels, but also by monitoring relevant integrated quantities such as drag or heat transfer coefficient.

4.4 PROBLEM SOLVING STEPS

After determining the important features of the problem following procedural steps are followed for solving it

1. Create the geometry model and mesh it.

2. Start the appropriate solver for 2D or 3D modeling.

3. Import the grid and check it.

4. Select the solver formulation

5. Chose the basic equation to solved: laminar or turbulent (or in viscid), chemical species or reaction, heat transfer models, etc. Also identify additional models needed: fans, heat exchangers, porous media, etc.

6. Specify the material properties.

7. Specify the boundary properties.

8. Adjust the solution control parameter.

9. Initialize the flow field.
10. Calculate a solution.
11. Examine the results.
12. Save the results.
13. If necessary, refine the grid or consider revisions to the numerical or physical model.

5. CHAPTER 5
GOVERNING EQUATIONS AND NUMERICAL SIMULATION

5.1 Introduction

Due to the advances in computational hardware and available numerical methods, CFD is a powerful tool for the prediction of the fluid motion in various situations, thus, enabling a proper design. CFD is a sophisticated way to analyze not only for fluid flow behavior but also the processes of heat and mass transfer.

Advances in physical models, numerical analysis and computational power enable simulation of the heat transfer characteristics in three-dimensional circumstances. A three dimensional approximation of a turbulent flow is chosen to explore since the three-dimensional approach is considerably greater than two dimensional and moreover, a turbulent flow is fundamentally three-dimensional.

Owing to extremely long computation times, detailed studies on the tubular heat exchanger in three-dimensional flow are very uncommon. Hence, the simulation of the three-dimensional flow field under complex geometrical conditions is seemingly intricate and challenging task.

The available computational fluid dynamics software package FLUENT is used to determine the related problems. FLUENT uses a finite volume method and requires from the user to supply the grid system, physical properties and the boundary conditions. When planning to simulate a problem, basic computation model considerations such as boundary conditions, the size of computational domain, grid topology, two dimensions or three-dimension model, are necessary. For example, appropriate choice of the grid type can save the set up time and computational expense. Moreover, a careful consideration for the selection of physical models and determination of the solution procedure will produce more efficient results. Dependent on the problem, the geometry can be created and meshed with a careful consideration on the size of the computational domain, and shape, density and smoothness of cells. Once a grid has been fed into FLUENT, check the grids and executes the solution after setting models, boundary conditions, and material properties. FLUENT provides the function for post processing the results and if necessary refined the grids is available and solve again as the above procedure.

As described in the objective, the purpose of this study is to investigate numerically the effect of maldistribution in the tubular heat exchanger.

5.2. Governing Equations

The flow and temperature field in the model geometry is determined by the continuity equation, the complete unsteady Navier-Stokes and the energy equation for incompressible fluid with temperature-dependent properties. These three-dimensional equations, to be solved by numerical calculations in the Cartesian coordinates, are as follows:

ISSN 2348 – 8034
Impact Factor- 4.022

Continuity equation:
$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0 \tag{5.1}$$

Momentum equation:
$$\frac{\partial}{\partial t}(\rho u_i u_j) = -\frac{\partial \rho}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} \tag{5.2}$$

Where
$$\tau_{ij} = \mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \frac{2}{3}\frac{\mu \partial u_k}{\partial x_k}\delta_{ij} \tag{5.3}$$

Energy equation :
$$\frac{\partial}{\partial t}(\rho E) + \frac{\partial}{\partial x_i}\left(u_i(\rho E + \rho)\right) = \frac{\partial}{\partial x_i}\left(k\frac{\partial T}{\partial x_i}\right) \tag{5.4}$$

Where *E* is the total energy and *k* is the thermal conductivity.

Navier-Stokes Equations in a Turbulent Flow Regime:

The Navier-Stokes equations mentioned thus far have been developed essentially for laminar flow regimes. However, in practical applications, flow is almost always turbulent. To compensate for this fact a model needs to be utilized to simulate turbulence. Turbulence is fundamentally the presence of velocity fluctuations within flow.

It is characterized by random, three-dimensional motions of fluid particles in addition to the mean motion (Fox & McDonald 1992).

Due to the actuality of the velocity fluctuations being random and high frequency leads to the consequence that the study of turbulence is very difficult. Different methods, which are employed to estimate turbulence, are described in the following sections.

The Standard k–ε Model:

The transport equations for the standard k–ε model are given in Equation . The derivation of these is complex and shall be left to reference books such as Wilcox (1993). An unpretentious statement of the origins of the partial differential equations is that the principle of energy balance is manipulated which is not accounted for in simple algebraic approximations. The main assumption in this model is that turbulence is simulated by an increase in the viscosity of the fluid.

5.3. Scope of the future work

1. Validation of obtained temperature profiles with the numerically solved temperature profiles.
2. Comparison going to be made for various modifications in heat exchangers.
3. The minimization of cost to be studied for the analyzing problems through CFD Software.

ISSN 2348 – 8034
Impact Factor- 4.022

6. MATHEMATICAL MODELING

6.1 INTRODUCTION

CATIA V5 is mechanical design software, addressing advanced process centric design requirements of the mechanical industry. With its feature based design solutions, CATIA proved to be highly productive for mechanical assemblies and drawing generation. CATIA, with its broad range if integrated solutions for all manufacturing organization.

CATIA is the best solution capable of addressing the complete product development process, from product concept specification through product service in a fully integrated and associative manner. CATIA mechanical design solutions provide tools to help you implement a sophisticated standard based architecture. This enables collaborative design and offers digital mockups and hybrid deigns.

- The domain includes-
- Product design & manufacturing.
- Drawing enterprise competitiveness
- Task presentation
- Process improvement.

CATIA V5 is totally compliant with windows presentation standards. CATIA V5 provides a unique two way interoperability with CATIA version4 data. As an open solution, CATIA includes with the most commonly used data exchange industry standards.

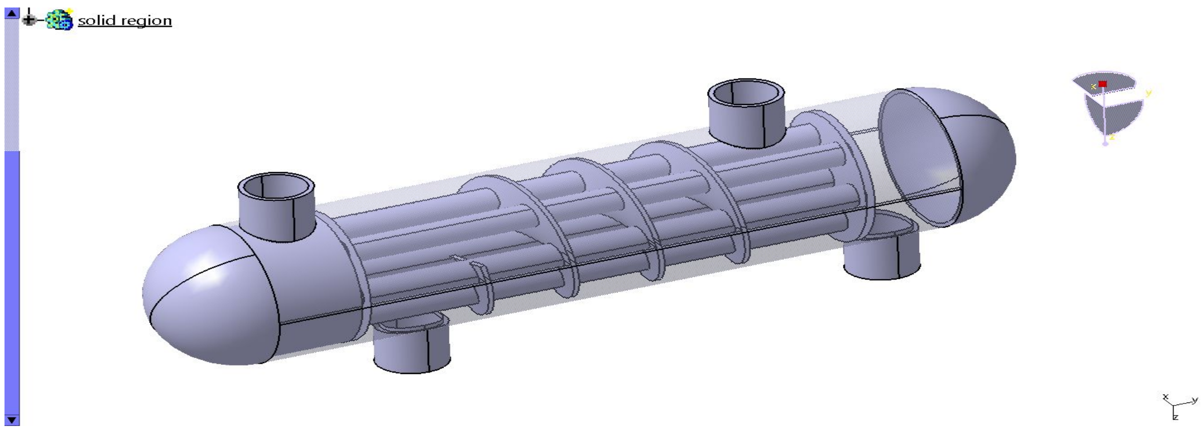
- CATIA V5 R17 extends the power of leading edge engineering practices to include relation design, which results in,
- Higher Quality design
- More opportunities for innovation
- Fewer engineering changes
- CATIA adds value to business in the following areas.
- Power major product programs
- Process expertise
- World class PLM(Product Lifecycle Management)
- Proven openness and standards support.

CATIA V5 offers three platforms (P1, P2, and P3) to choose the most scalable solution for product creation. CATIA V5 P1 users benefit from PLM productivity in an affordable way the security of potential growth. They can conduct associative product engineering based on CATIA V5 product design-in-context, product knowledge reuse, end-to-end-associability, product validation, and collaborative change management capabilities. CATIA V5 P2 users can optimize their PLM processes through knowledge integration, process accelerators, and customized tools.

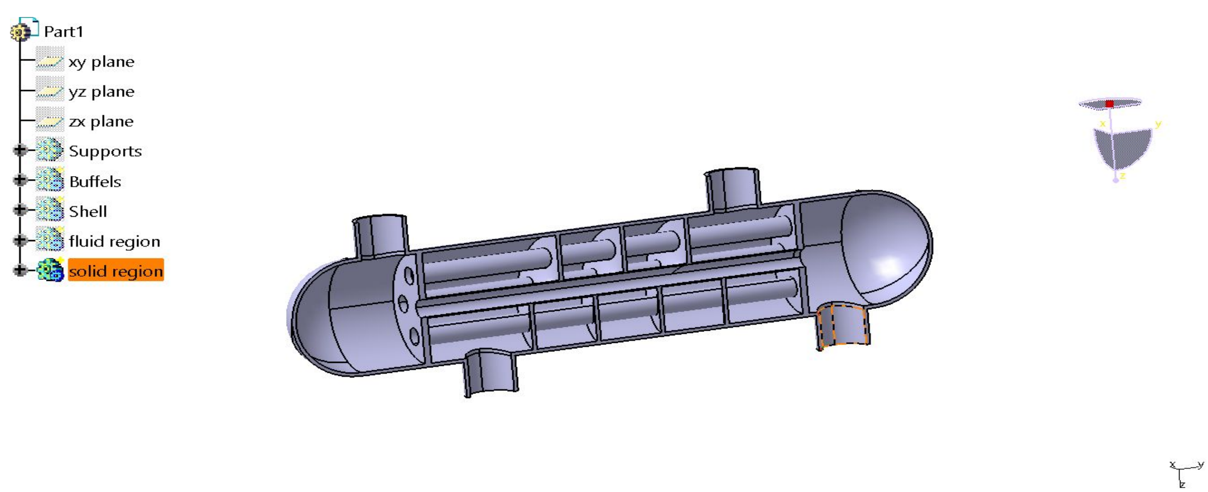
ISSN 2348 – 8034
Impact Factor- 4.022

CATIA V5 P3 users access the highest productivity for specific advanced processes with focused solutions. They can lead expert engineering and advainnovation, relying on unique and very specialized that integrates product and process expertise.

- SKETCHER
- PART DESIGN
- ASSEMBLY DESIGN
- WIREFRAME AND SURFACE DESIGN
- DRAFTING
- REAL

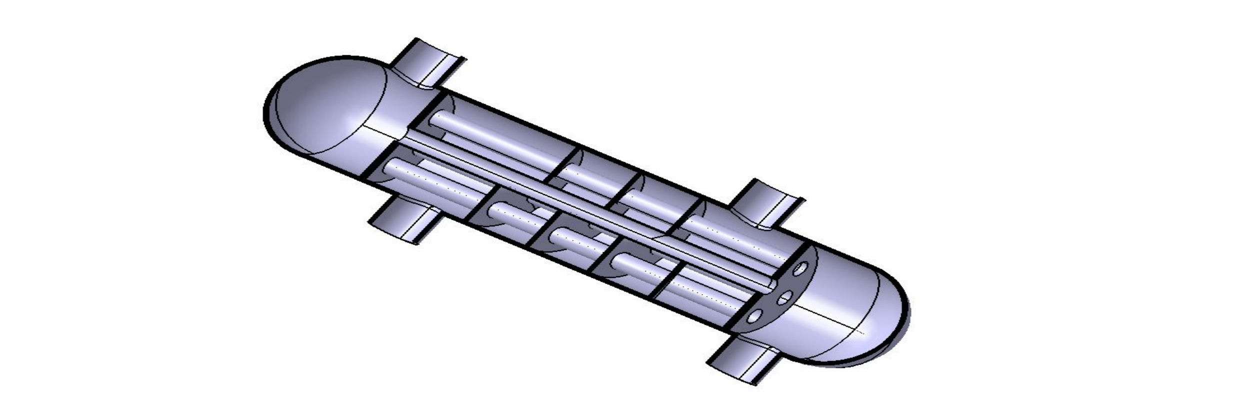


6.1 3D Modeling of shell and Tube heat Exchanger (CATIA)

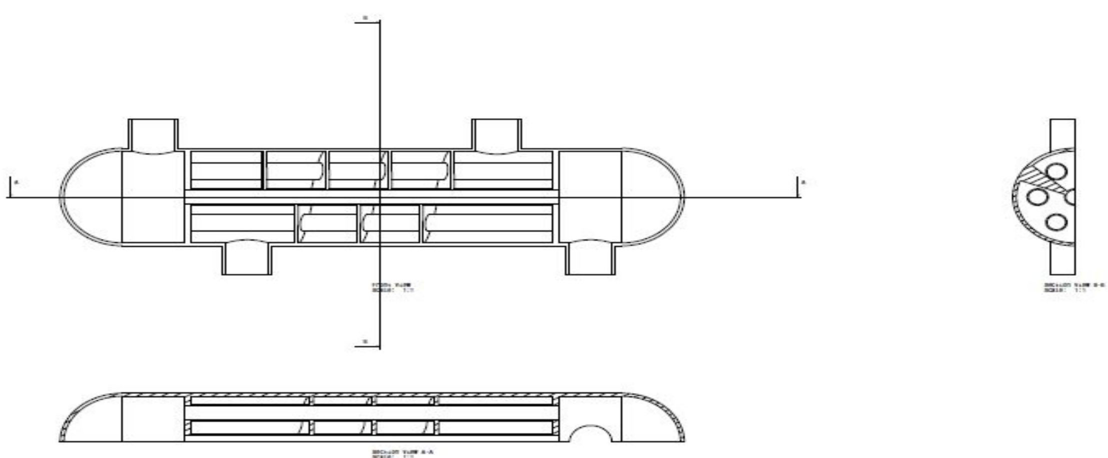


6.2 3D Modeling sectional view of shell and Tube heat Exchanger (CATIA)

DOI- 10.5281/zenodo.56040
ISSN 2348 – 8034
Impact Factor- 4.022



6.3 3D Modeling of shell and Tube heat Exchanger (CATIA)



6.4 2D orthographic sectional views of shell and Tube heat Exchanger (CATIA)

7. CHAPTER - VII
RESULTS AND DISCUSSION
7.1 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,X-velocity ,Yvelocity, 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.

[Karthikeyan, 3(6): June 2016] ISSN 2348 – 8034
DOI- 10.5281/zenodo.56040 Impact Factor- 4.022

7.2 Geometry and Mesh:

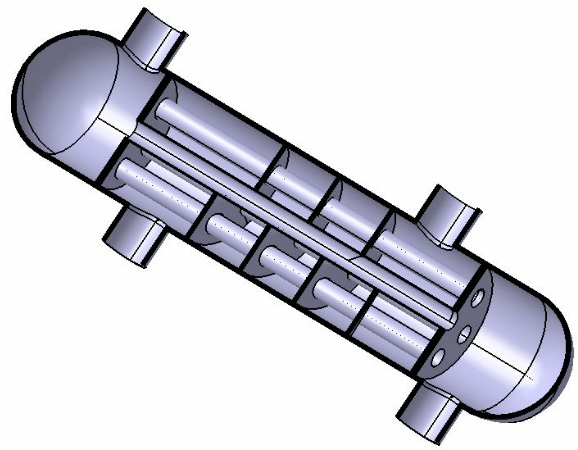


Fig 7.1 Isometric view of arrangement of baffles and tubes of shell and tube heat exchanger with baffle inclination

7.3. Grid Generation

The three-dimensional model is then discretized in ICEM CFD. In order to capture both the thermal and velocity boundary layers the entire model is discretized using hexahedral mesh elements which are accurate and involve less computation effort. Fine control on the hexahedral mesh near the wall surface allows capturing the boundary layer gradient accurately.

The entire geometry is divided into three fluid domains Fluid_Inlet, Fluid_Shell and Fluid_Outlet and six solid domains namely Solid_Baffle1 to Solid_Baffle6 for six baffles respectively. The heat exchanger is discretized into solid and fluid domains in order to have better control over the number of nodes.

The fluid mesh is made finer then solid mesh for simulating conjugate heat transfer phenomenon. The three fluid domains are as shown in Fig. 2. The first cell height in the fluid domain from the tube surface is maintained at 100 microns to capture the velocity and thermal boundary layers.

The discretised model is checked for quality and is found to have a minimum angle of 18° and min determinant of 4.12. Once the meshes are checked for free of errors and minimum required quality it is exported to ANSYS CFX pre-processor

7.4 Meshing

Initially a relatively coarser mesh is generated with 1.8 Million cells. This mesh contains mixed cells (Tetra and Hexahedral cells) having both triangular and quadrilateral faces at the boundaries. Care is taken to use structured cells (Hexahedral) as much as possible, for this reason the geometry is divided into several parts for using automatic methods available in the ANSYS meshing client.

It is meant to reduce numerical diffusion as much as possible by structuring the mesh in a well manner, particularly near the wall region. Later on, for the mesh independent model, a fine mesh is generated with 5.65 Million cells. For this fine mesh, the edges and regions of high temperature and pressure gradients are finely meshed.

[Karthikeyan, 3(6): June 2016]
DOI- 10.5281/zenodo.56040

ISSN 2348 – 8034
Impact Factor- 4.022

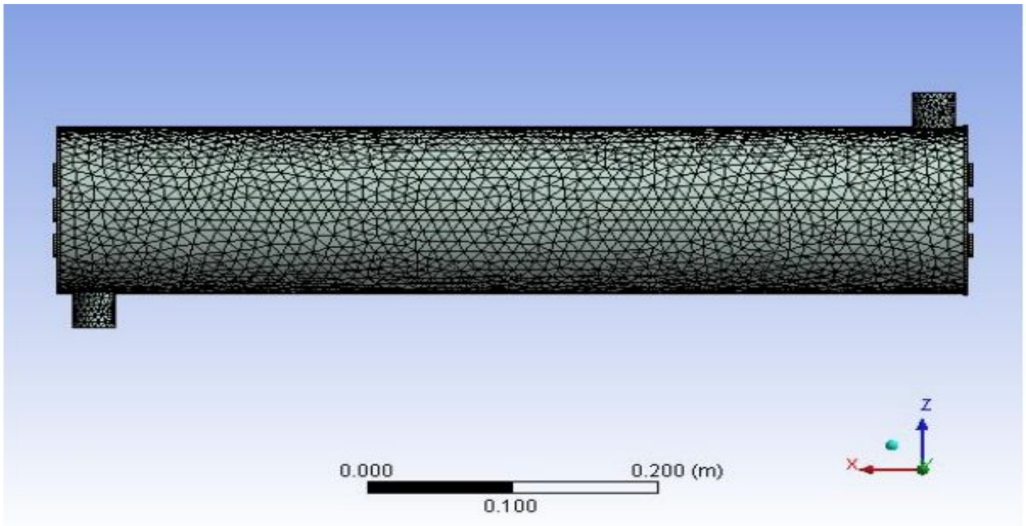


Fig 7.2 Meshing diagram of shell and tube heat exchanger

7.5 Problem Setup

Simulation was carried out in ANSYS® FLUENT® v13. In the Fluent solver Pressure Based type was selected , absolute velocity formation and steady time was selected for the simulation . In the model option energy calculation was on and the viscous was set as standard k-e, standard wall function(k-epsilon 2 eqn). In cell zone fluid water-liquid was selected. Water-liquid and cupper, aluminum was selected as materials for simulation. Boundary condition was selected for inlet,outlet. In inlet and outlet 1kg/s velocity and temperature was set at 353k.Across each tube 0.05kg/s velocity and 300k temperature was set. Mass flow was selected in each inlet. In reference Value Area set as 1m2 ,Density 998 kg/m3 ,enthalpy 229485 j/kg , length 1m , temperature 353k, Velocity 1.44085 m/s , Ration of specific heat 1.4 was considered .

7.6 Solution Initialization:

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

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

7.7 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,X-velocity ,Yvelocity, Z-velocity , k, epsilon should be less than 10-4 and the energy should be less than 10-7 .

[Karthikeyan, 3(6): June 2016] ISSN 2348 – 8034
DOI- 10.5281/zenodo.56040 Impact Factor- 4.022

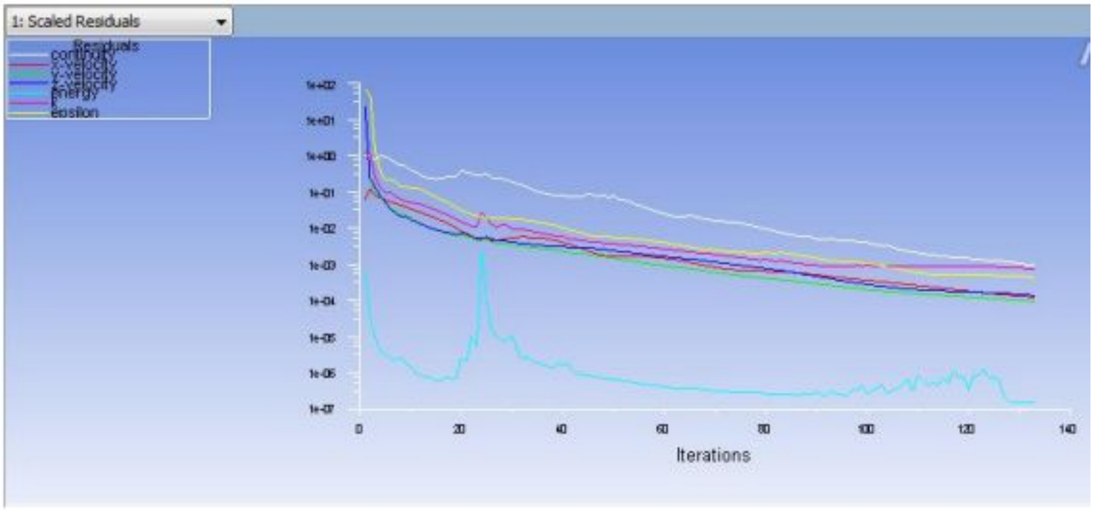


Fig 7.3 Convergence Of Simulation

7.8 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 vertical and helical baffle

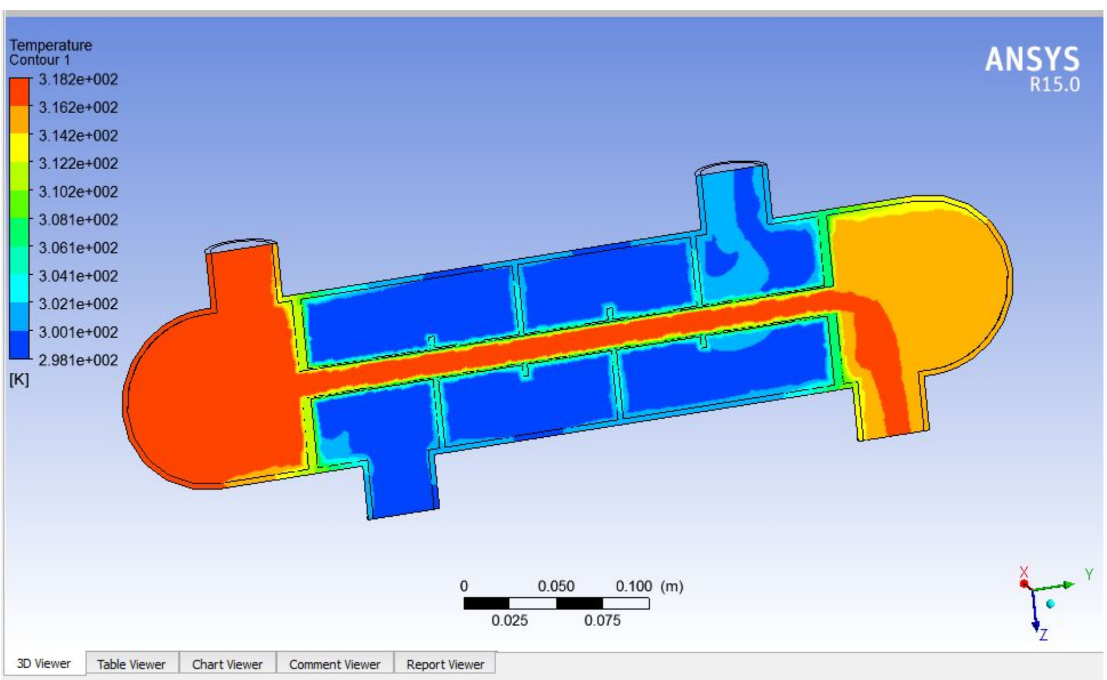


Figure 7.4 Vertical baffle Temperature Distribution across the tube and shell

ISSN 2348 – 8034
Impact Factor- 4.022

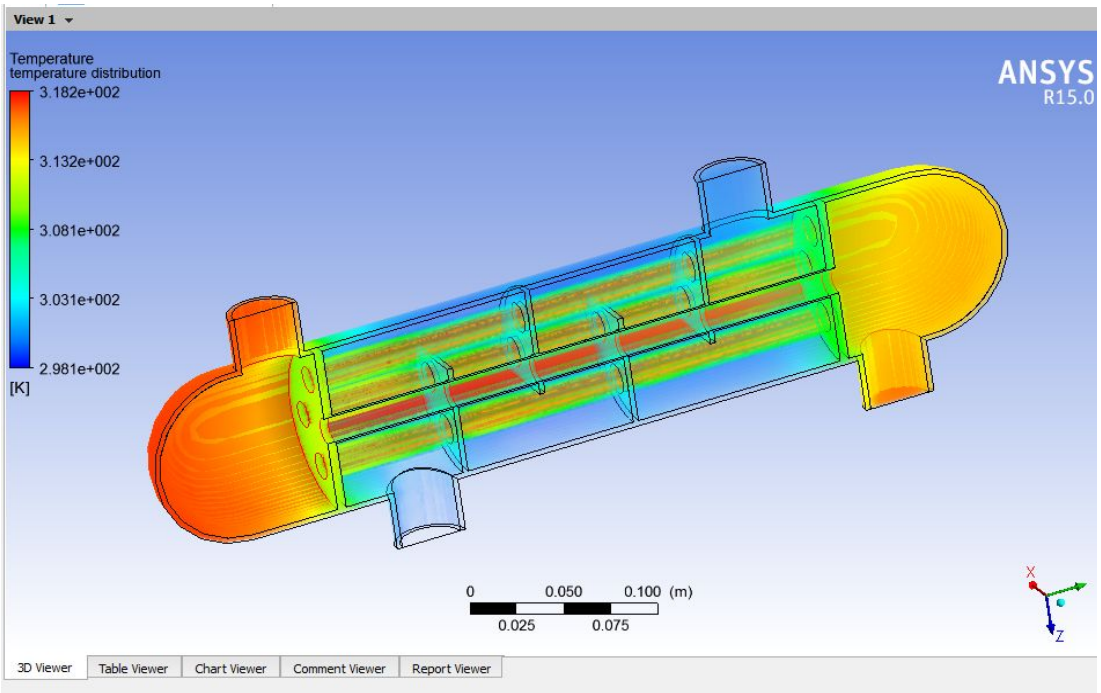


Figure 7.5 Vertical baffle Temperature Distribution across the tube and shell (volume transperant)

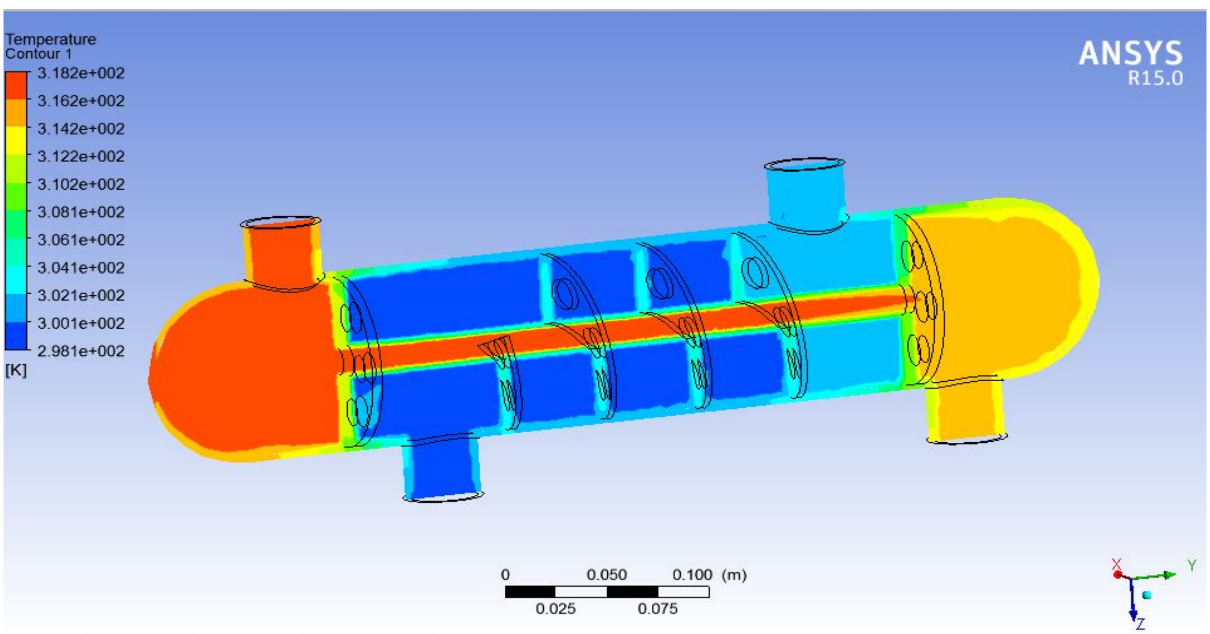


Figure 7.6 helical baffle Temperature Distribution across the tube and shell

ISSN 2348 – 8034
Impact Factor- 4.022

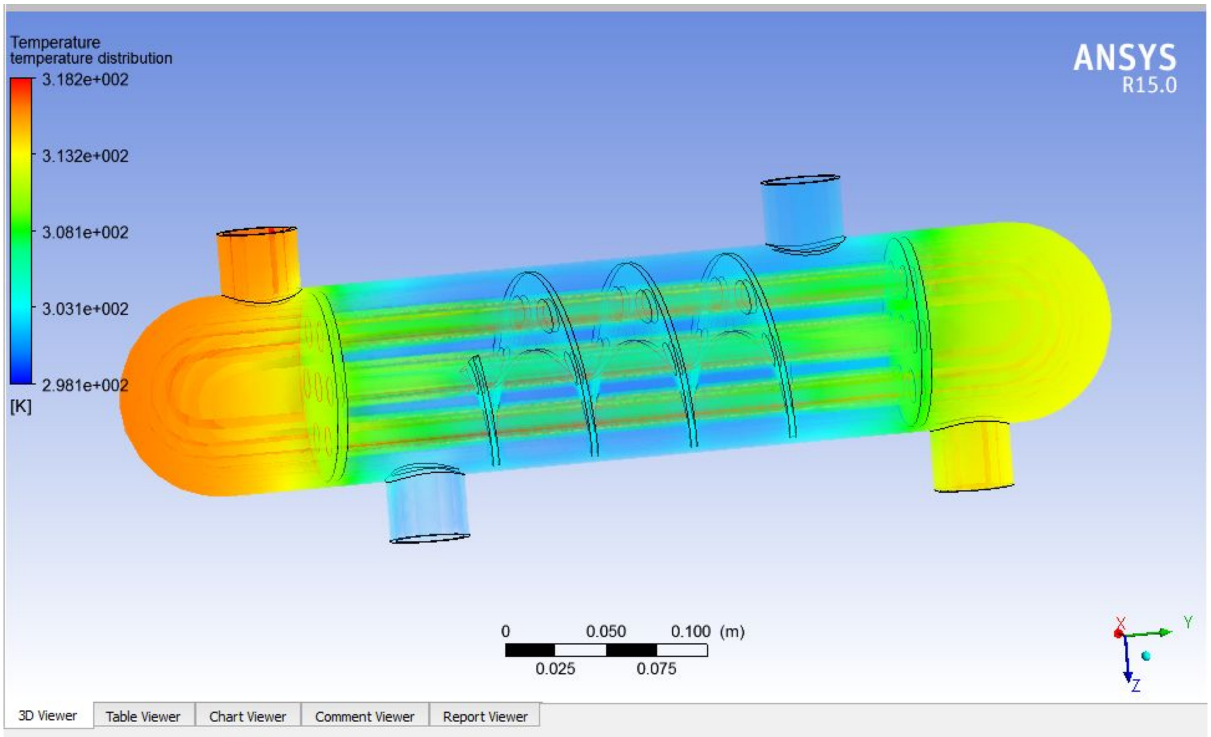


Figure 7.7 Helical baffle Temperature Distribution across the tube and shell (volume transperant) .

7.9 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.

 ISSN 2348 – 8034
Impact Factor- 4.022

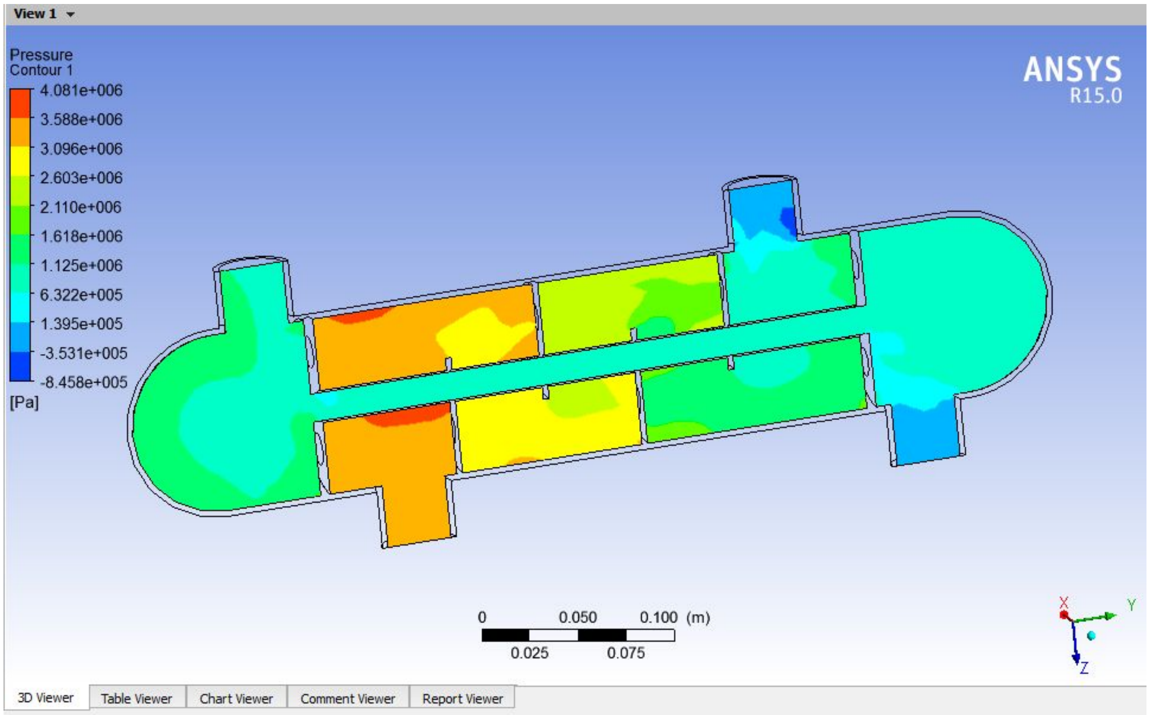


Figure 7.8 vertical baffle -Pressure Distribution across the shell and tube

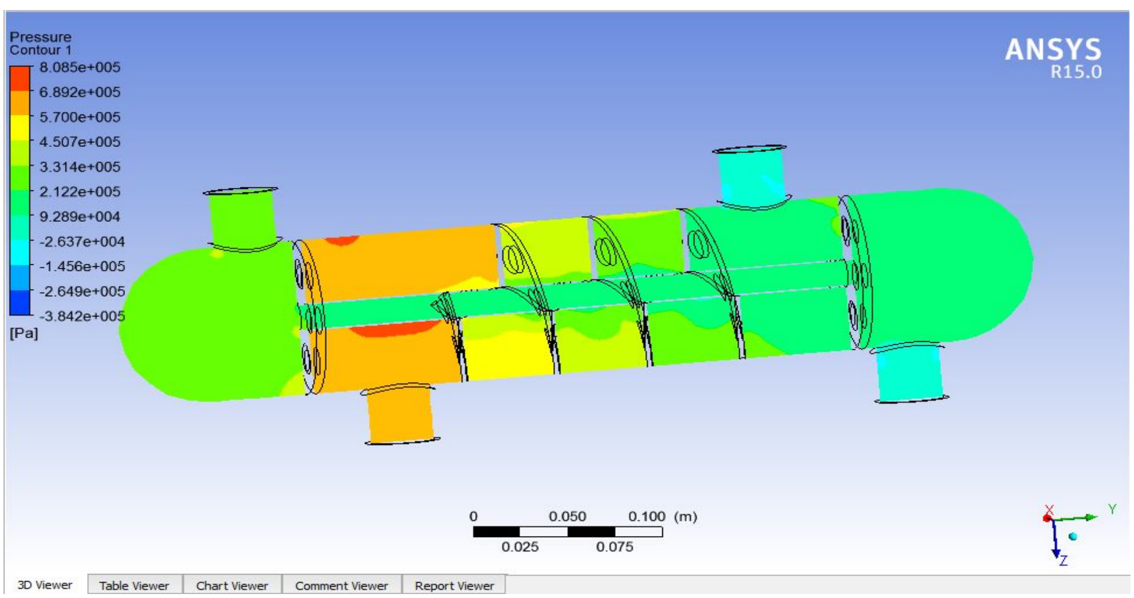


Figure 7.9 Helical baffle Pressure Distribution across the tube and shell

ISSN 2348 – 8034
Impact Factor- 4.022

7.10 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.

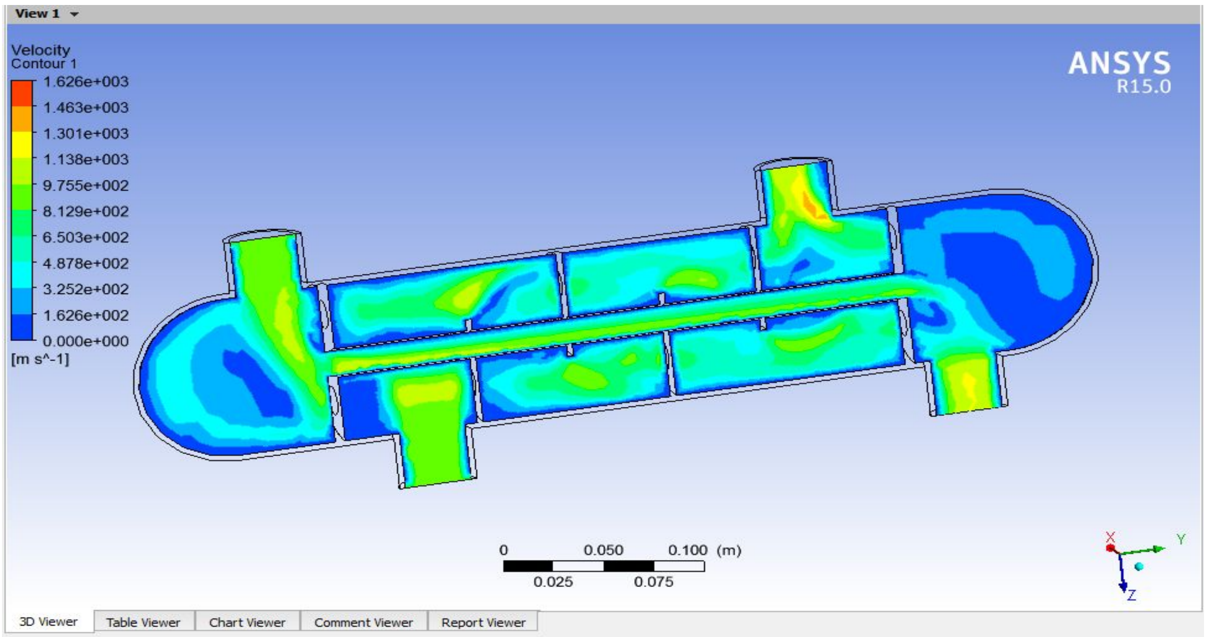


Figure 7.10 vertical velocity Distribution across the tube and shell

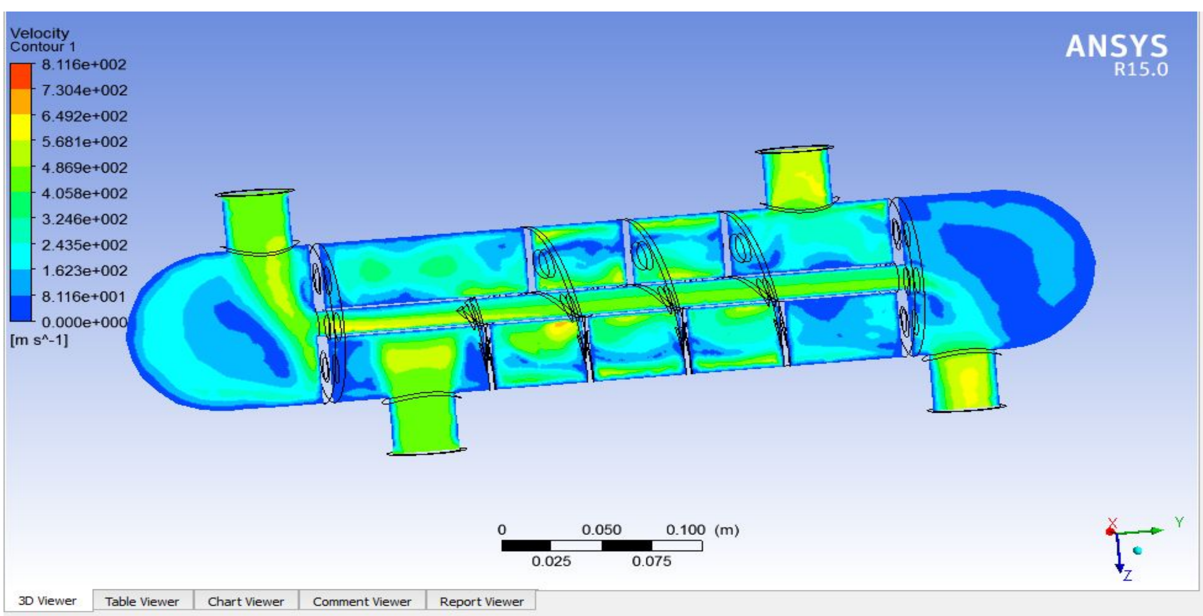


Figure 7.11 helical baffle velocity Distribution across the tube and shell

ISSN 2348 – 8034
Impact Factor- 4.022

Table 7.1 Temperature variation

Baffle Inclination Angle (Degree)	Outlet Temperature Of Shell side	Outlet Temperature Of Tube side
Vertical baffle	346	317
Helical baffle	347.5	319

7.11 Heat Transfer Rate

Q = m * Cp * ΔT

m=mass flow rate

Cp = Speific Heat of Water

ΔT = Temperature Difference Between Tube Side

Table 7.2 for Heat Transfer Rate Across Tube side

Baffle Inclination	Heat Transfer Rate Across Tube side (w/m2)
Vertical baffle	3557.7
Helical baffle	3972.9

Table 7.3 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)
vertical	346	317	230.992	3554.7	4.2
Helical	347.5	319	229.015	3972.9	5.8

8. CONCLUSIONS

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

9. FUTURE WORK

In this paper, a solution method of the shell and tube heat exchanger design optimization problem was proposed based on the utilization of cfd. Referring to the literature test cases, reduction of capital investment up to 7.4% and savings in operating costs up to 93% were obtained, with an overall decrease of total cost up to 52%, showing the improvement potential of the proposed method. Furthermore, the genetic algorithm allows for rapid solution of the design problem and enables to examine.

A number of alternative solutions of good quality, giving the designer more degrees of freedom in the final choice with respect to traditional methods.

As a future work, it intended to deal in detail with issues of mechanical design and Creation of CAD model is done by using CAD modeling software (CATIA V5). Design of the shell and tube heat exchanger with required parameters like external diameter, internal diameter, and baffle space is shown through CATIA V5. By studying the properties of the varied materials, I manage to change the properties of the materials in the shell and tube to increase the heat transfer rate and cost efficient process through CFD Analysis.

REFERENCES

[1] Wilfried Roetzel and Chakkrit Na Ranong,. 1999, "Consideration of maldistribution in heat exchangers using the hyperbolic dispersion model" Chemical Engineering and Processing 38, pp. 675–681.

[2] Sahoo, R.K., and Wilfried Roetzel., 2002, "Hyperbolic axial dispersion model for heat exchangers" International Journal of Heat and Mass Transfer 45, pp 1261–1270.

[3] Wilfried Roetzel , Chakkrit Na Ranong., 2000 "axial dispersion model for heat exchangers" Heat and Technology vol. 18.

[4] Yimin Xuan, and Wilfried Roetzel., "Stationary and dynamic simulation of multipass shell and tube heat exchangers with the dispersion model for both fluids" Int. J. Heat Mass Transfer. Vol. 36, No. 17,4221A231,

[5] Danckwerts, P.V., 1953, Continuous flow systems Distribution of Residence times

ISSN 2348 – 8034
Impact Factor- 4.022

Chemical Engineering science genie chimique Vol. 2.

[6] Lalot, S., P. Florent , Langc, S.K., Bergles, A.E., 1999, "Flow maldistribution in heat exchangers "Applied Thermal Engineering 19, pp 847-863.

[7] Prabhakara Rao Bobbili, Bengt Sunden, and. Das, S.K., 2006, "An experimental investigation of the port flow maldistribution in small and large plate package heat exchangers Applied Thermal Engineering 26, pp 1919–1926.

[8] Zakro Stevanovic, Gradimir, Ilić., Nenad Radojković, Mića Vukić, Velimir Stefanović, Goran Vučković., 2001, "Design of shell and tube heat exchangers by using CFD technique- part one: thermo hydraulic calculation" Facta Universities Series: Mechanical Engineering Vol.1, No 8, pp. 1091 – 1105

[9] Wilfried roetzel and Das, S.K., 1995 "Hyperbolic axial dispersion model: concept and its application to a plate heat exchanger" Int..J. Heat Mass Transfer. Vol. 38, No. 16, pp.3062-3076.

[10] Anindya Roy, and Das, S.K., 2001, "An analytical solution for a cyclic regenerator in the warm-up period in presence of an axially dispersive wave" Int. J. Therm. Sci. 40, pp.21–29

[11] Ping Yuan., 2003 "Effect of inlet flow maldistribution on the thermal performance of a three-fluid crossflow heat exchanger International" Journal of Heat and Mass Transfer 46, pp.3777–3787.

[12] Srihari, N., Prabhakara Rao, B., Bengt Sunden, Das, S.K., 2005 "Transient response of plate heat exchangers considering effect of flow maldistribution" International Journal of Heat and Mass Transfer 48, pp. 3231–3243.

[13] Roetzel, W., Spang, B., Luo, X., and Dash, S.K., 1998 "Propagation of the third sound wave in fluid : hypothesis and theoretical foundation" International Journal of Heat and Mass Transfer 41, pp. 2769.-2780.

[14] Zhe Zhang and YanZhong Li., 2003 "CFD simulation on inlet configuration of plate-fin heat exchangers" Cryogenics 43, pp. 673–678.

[15] Xuan, Y., and Roetzel, W., " Dynamics of shell-and-tube heat exchangers to arbitrary temperature and step flow variations" AIChE Journal Volume 39, Issue 3 , pp.413 – 421.

[16] Wilfried Roetzel, and Frank Balzereit., 2000, "Axial dispersion in shell-and-tube heat exchangers" Int. J. Therm. Sci. 39, 1028–1038.

[17] Mueller, A.C., 1987, "Effects of some types of maldistribution on the performance of heat exchangers" Heat Transfer Engineering Volume 8, Issue 2, pp. 75-86.

[18] Anil Kumar Dwivedi and Sarit Kumar Das., 2007, "Dynamics of plate heat exchangers subject to flow variations" International Journal of Heat and Mass Transfer Volume 50, Issues 13-14, pp. 2733-2743.

[19] Ranganayakulu, Ch., and Seetharamu, K.N., 2000, "The combined effects of wall longitudinal heat conduction and inlet fluid flow maldistribution in crossflow plate-fin heat exchangers Heat and Mass Transfer 36, pp. 247 ± 256.

[20] Karno, A., and Ajib, S., 2006, "Effect of tube pitch on heat transfer in shell-and- tube heat exchangers—new simulation software Heat Mass Transfer 42, pp. 263–272.