NUMERICAL INVESTIGATION ON PERFORMANCE OF VCR SYSTEM USING SHELL AND TUBE HEAT EXCHANGER

SACHU PRASAD[1], MELVIN M THOMAS[2], LINCE MATHEW THOMAS[3], RAHUL REGHU[4] AJAI M[5]

[1,2,3,4]Student, Dept. of Mechanical Engineering, Musaliar College of Engineering and Technology, Pathanamthitta, Kerala, India
[5]Assitant Professor, Dept. of Mechanical Engineering, Musaliar College of Engineering and Technology, Pathanamthitta, Kerala, India

---***---

Abstract - *The production of dry air is getting important in the present scenario. Dry air has several key applications in industries such as food industries, pharmaceutical industries agricultural industries for preserving and maintaining. In many process industries Dry air flow is used to make the raw-materials dry and moisture free for better transportation from one stage to other during the process. This air-flow is obtained from big industrial compressors and the output air of which contains atmospheric moisture. This moisture of air is removed through a Refrigeration system. The efficiency or COP of the refrigeration system depends on the inlet temperature of the air which comes from an air-compressor which remains hot.*

The present system uses an additional double pipe heat exchanger (DPHE) for moisture removing purpose. The low efficiency of the double pipe heat exchanger leads to the gap for modification. We are replacing the DPHE by a Shell and Tube Heat Exchanger (STHE) which has more efficiency and requires less area. Several improvements within the STHE can also done using helical baffles. CFD analysis done on double pipe heat exchanger and shell and tube heat exchanger is done and COP is compared.

Key Words: COP, Shell and tube heat exchanger, Double pipe heat exchanger, CFD analysis.

1. INTRODUCTION

In many process industries Dry air flow is used to make the raw-materials dry and moisture free for better transportation from one stage to other during the process. This air-flow is obtained from big industrial Compressors and the output air of which contains atmospheric moisture. A case study was undertaken at a company viz. HEG Ltd where carbon rods are produced from graphite powder. These graphite powders are dried by flowing dry & moisture free air over them for pneumatic conveying the powder from one process to another. The flow of high pressure air from compressor is generally hot and contains moisture. This moisture of air is removed from the air through a Refrigeration system. Now as the efficiency or COP of the refrigeration system depends on the inlet temperature of the air which comes from an air-compressor which remains hot. The heat of inlet air is removed by using the cold air through the introduction of an additional Heat-exchanger. As a result, the temperature of the inlet air to the refrigeration system is reduced thus improving the efficiency & COP of the Refrigeration system.

The present system uses an additional double pipe heat exchanger (DPHE) for moisture removing purpose. The low efficiency of the double pipe heat exchanger leads to the gap for modification. We are replacing the DPHE by a Shell and Tube Heat Exchanger (STHE) which has more efficiency

Here the model made in CATIA and CFD simulation is used to investigate the heat transfer and fluid flow in shell and tube heat exchanger with helical baffle and double pipe heat exchanger. Numerical investigations on shell side fluid flow and heat transfer are conducted by using commercial CFD software ANSYS Fluent Release 16.0. CFD is the science of predicting fluid flow, heat and mass transfer, chemical reactions and related phenomena by solving numerically the set of governing mathematical equations, which is stated in the ANSYS training module.

2. LITERATURE REVIEW

Prabal Roy [1] in his journal 'Improvement of Efficiency of Air Refrigeration System by Lowering the Inlet Temperature of Air' modified the previous system of production of dry air by using the waste heat from the compressor plant by installing an additional double pipe air to air heat exchanger at the outlet of evaporator.

Shambhu Kumar Rai [2] studied about various types of heat exchangers in his journal 'A Review on Heat Exchanger.'

M M Aslam Bhutta [3] in his journal 'CFD Application in Various Heat Exchangers Design: A Review' found out CFD analysis is the best way to find out the best heat exchanger which can be used unless using the conventional method of creating each heat exchanger without knowing their efficiency at the first place.

Ammar Ali Abd [4] studied about performance of STHE in his journal 'Performance Analysis of Shell and Tube Heat Exchanger: Parametric Study' investigated the effect of change some parameters on heat transfer coefficient and pressure drop for shell and tube heat exchanger concluded that as shell diameter increases the heat transfer coefficient and pressure drop increases.

Gurbir Singh [5] performed CFD technique which is a computer-based analysis to simulate the shell and tube heat exchanger involving fluid flow, heat transfer and stated that CFD helps to design the heat exchanger easily otherwise it is very difficult to done practically. Validation of the software used is done on the basis of this journal,

Pranita Bichkar [6] in his journal 'Study of Shell and Tube Heat Exchanger with the Effect of Types of Baffles' concluded that the use of helical baffles over other two lowers the pressure drop which in turn increases the overall system efficiency. So, it is proved that helical baffles are more advantageous than other two types of baffles.

Ravi Kumar Banjare [7] in his journal 'A Paper on The Analysis of Effect of Material Used in Heat Exchanger and Its Performance' studied about a number of factors need to be considered when selecting a tube material of the heat exchanger found out that copper and copper alloys when used as the tubes of STHE conducts more heat exchange than any other materials.

Li He [8] in his journal 'Numerical Investigation on Double Tube-Pass Shell-And-Tube Heat Exchangers with Different Baffle Configurations' compares the performances of Single Tube Pass STHE and Double Tube Pass STHE with segmental baffles reveals that DTP-STHE can improve recovered heat quality in the shell side without decreasing heat transfer rate.

In a work by Pranita Bichkar [9] on 'Study of Shell and Tube Heat Exchanger with The Effect of Types of Baffles' presents the numerical simulations carried out on different baffles i.e. single segmental, double segmented and helical concluded that use of helical baffles over other two lowers the pressure drop which in turn increases the overall system efficiency.

Usman Salahuddin [10] on his study about Helical baffle in the journal 'A Review of The Advancements Made in Helical Baffles Used in Shell and Tube Heat Exchangers found that effectiveness of heat exchanger with two-layer helical baffles is higher than single layer helical baffles.

N. Piroozfam [11] in his journal 'Numerical Investigation of Three Methods for Improving Heat Transfer in Counter Flow Heat Exchangers' investigated about the methods for improving heat transfer and found out that counter flow heat exchangers have the better efficiency.

Dipankar Del [12] in his journal Helical Baffle Design in Shell and Tube Type Heat Exchanger with CFD Analysis shows the clear idea that the helical baffle heat exchanger has better overall heat transfer coefficient than the straight baffle Heat Exchanger.

3. THEORY

Irjet Template sample paragraph .Define abbreviations and acronyms the first time they are used in the text, even after they have been defined in the abstract. Abbreviations such as IEEE, SI, MKS, CGS, sc, dc, and rms do not have to be defined. Do not use abbreviations in the title or heads unless they are unavoidable.

3.1. HEAT EXCHANGERS

A Heat exchanger is a piece of equipment built for efficient heat transfer from one medium to another. The media may be separated by a solid wall, so that it never mixes or may be in direct contact. The heat exchangers widely used in space heating, refrigeration, air conditioning, power plants, chemical plants, petrochemical plants, petroleum refineries, and natural gas processing and sewage treatment. One common example of a heat exchanger is the radiator in a car, in which the heat source, being a hot engine-cooling fluid, water transfer heat to air flowing through the radiator.

3.1.1. TYPES OF HEAT EXCHANGERS

Double Pipe Heat Exchanger

A double pipe heat exchanger (also sometimes referred to as a 'pipe-in-pipe' exchanger) is a type of heat exchanger comprising a 'tube in tube' structure. As the name suggests, it consists of two pipes, one within the other. One fluid flows through the inner pipe (analogous to the tube-side in a shell and tube type exchanger) whilst the other flows through the outer pipe, which surrounds the inner pipe (analogous to the shell-side in a shell and tube exchanger).

A cross-section of a double pipe exchanger would look something like this:

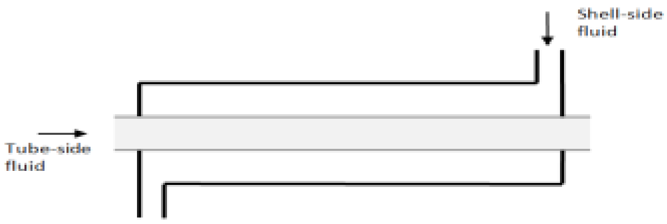

Fig -3.1: Double Pipe Heat Exchanger

A double pipe heat exchanger, in its simplest form is just one pipe inside another larger pipe. One fluid flows through the inside pipe and the other flows through the annulus between the two pipes. The wall of the inner pipe is the heat transfer surface. The pipes are usually doubled back multiple times as shown in the diagram at the left, in order to make the overall unit more compact.

Counter flow and Parallel Flow in a Double Pipe Heat Exchanger

Counter flow

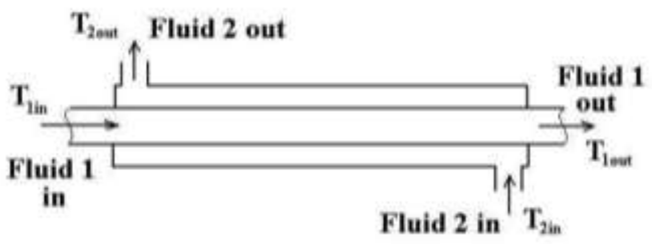

Fig -3.2: Counter flow in Double Pipe Heat exchanger

Two advantages for counter flow, (a) larger effective LMTD and (b) greater potential energy recovery. The advantage of the larger LMTD, as seen from the heat exchanger equation, is that a larger LMTD permits a smaller heat exchanger area, Ao, for a given heat transfer, Q. This would normally be expected to result in smaller, less expensive equipment for a given application.

Parallel flow

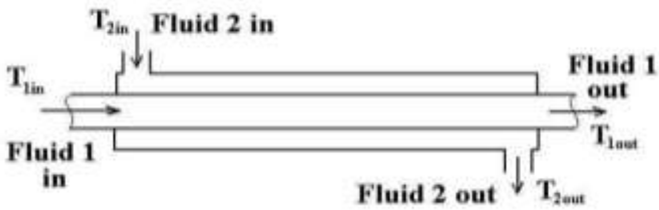

Fig -3.3: Parallel flow in Double Pipe Heat exchangers

Parallel flows are desirable (a) where the high initial heating rate may be used to advantage and (b) where it is required the temperatures developed at the tube walls are moderate. In heating very viscous fluids, parallel flow provides for rapid initial heating and consequent decrease in fluid viscosity and reduction in pumping requirement. In applications where moderation of tube wall temperatures is required, parallel flow results in cooler walls. This is especially beneficial in cases where the tubes are sensitive to fouling effects which are aggravated by high temperature.

Shell and Tube heat exchanger

Shell and tube heat exchangers consist of a series of tubes which contain fluid that must be either heated or cooled. A second fluid runs over the tubes that are being heated or cooled so that it can either provide the heat or absorb the heat required. A set of tubes is called the tube bundle and can be made up of several types of tubes: plain, longitudinally finned, etc. Shell and tube heat exchangers are typically used for high-pressure applications (with pressures greater than 30 bar and temperatures greater than 260 °C). This is because the shell and tube heat exchangers are robust due to their shape several thermal design features must be considered when designing the tubes in the shell and tube heat exchangers: 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.

Classification of Shell and Tube Heat Exchangers

The tubes may be straight or bent in the shape of a U, called U-tubes.

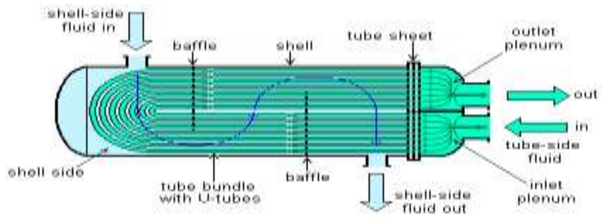

Fig -3.4: U Tube Heat exchanger

In nuclear power plants called pressurized water reactors, large heat exchangers called steam generators are two-phase, shell-and-tube heat exchangers which typically have U-tubes. They are used to boil water recycled from a surface condenser into steam to drive a turbine to produce power. Most shell-and-tube heat exchangers are either 1, 2, or 4 pass designs on the tube side. This refers to the number of times the fluid in the tubes passes through the fluid in the shell. In a single pass heat exchanger, the fluid goes in one end of each tube and out the other.

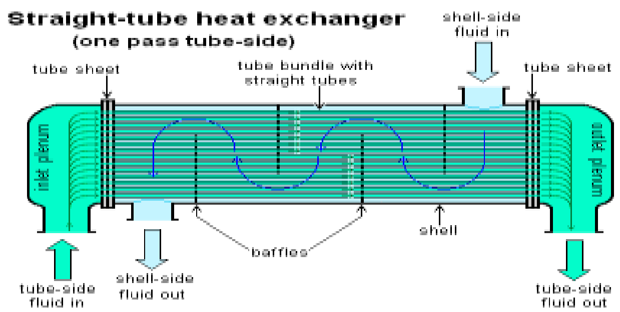

Fig -3.5: Straight-tube (one pass) heat exchanger

Surface condensers in power plants are often 1-pass straight-tube heat exchangers (see surface condenser for diagram). Two and four pass designs are common because the fluid can enter and exit on the same side. This makes construction much simpler.

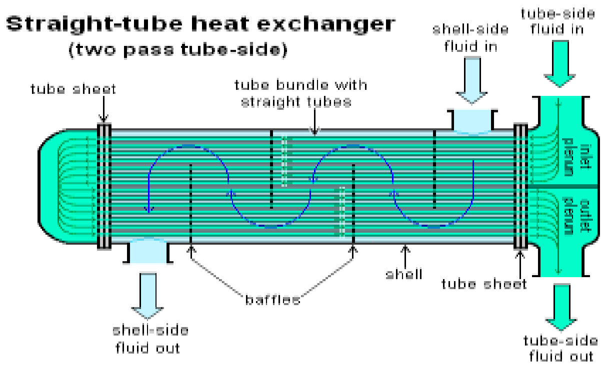

Fig -3.6: Straight-tube (double pass) heat exchanger

3.2. BAFFLES

Baffle is a device used to put down the flow of a fluid, gas etc. Baffles serve two important functions. They support the tubes during assembly and operation and help prevent vibration from flow induced eddies and direct the shell side fluid back and forth across the tube bundle to provide effective velocity and Heat Transfer rates. The diameter of the baffle must be slightly less than the shell inside diameter to allow assembly but must be close enough to avoid the significant performance penalty caused by fluid bypass around the baffles.

Shell roundness is important to achieve effective sealing against excessive bypass. Baffles can be made from a variety of materials compatible with the shell side fluid. They can be punched or machined. Some baffles are made by a punch which provides a lip around the tube hole to provide more surfaces against the tube and eliminate tube wall cutting from the baffle edge.

Baffles may be classified as transverse and longitudinal types. The purpose of longitudinal baffles is to control the overall flow direction of the shell fluid such that a desired overall flow arrangement of the two fluid streams is achieved. For example, two-pass shell with longitudinal baffle, split flow, double split flow. Transverse baffles may be classified as plate baffles and grid. Plate baffles may be single segmental, double-segmental, and triple-segmental, non-tubes-in-window segmental baffle and disk-and-doughnut baffle.

3.2.1. HELICAL BAFFLES

The Helical Baffle Heat Exchanger is also known as a Helix changer solution that removes many of the deficiencies of Segmental Baffle Heat Exchanger. It is very effective where heat exchanger is predicted to be faced with vibration condition Quadrant shaped baffle segment are arranged right angle to the tube axis in a sequential pattern that guide the shell side flow in a helical path over the tube bundle. The Helical flow provides the necessary characteristics to reduce flow dispersion and generate near plug flow conditions. The shell side flow configuration offers a very high conversion of pressure drop to heat transfer. Advantages over segmental STHE are increased heat transfer rate, reduced bypass effects, reduced Shell Fouling Factor, Prevention of flow induced vibration & Reduces Pumping cost. Shell and tube type heat exchanger with helical baffle diagram is shown in Figure 3.7.

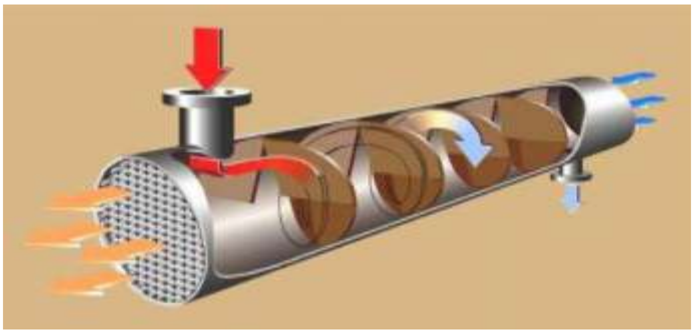

Fig -3.7: Shell and tube type heat exchanger with helical Baffle

3.2.2. SEGMENTED BAFFLES

Classified into two:

- Single segmental

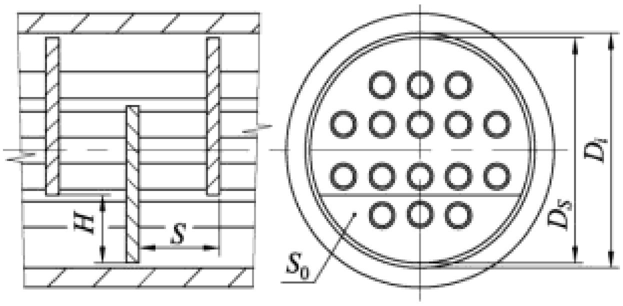

Fig -3.8: Single segmental baffles

- Double Segmental

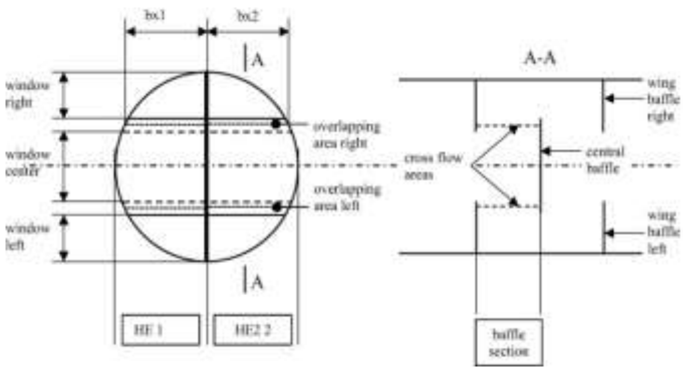

Fig -3.9: Double segmental baffle

3.3. GOVERNING EQUATIONS

The fluid flow assumed study-state turbulent model with incompressible fluid. The shell side fluid flows only through the hexagonal vents and it is assumed that three are no other leakages. Finite volume method is adopted to solve the model equations like Continuity equation (1) momentum equation (2) and energy equation (3).
The equations are given below:

$$\frac{\partial ui}{\partial xi} = 0 \quad (1)$$

$$\frac{\partial uiuj}{\partial xi} = -\frac{1}{\rho}\frac{\partial \rho}{\partial xi} + \frac{\partial}{\partial xi}\{(v + vturb)(\frac{\partial ui}{\partial xi} + \frac{\partial uj}{\partial xi})\} \quad (2)$$

$$\frac{\partial uiT}{\partial xi} = \rho\frac{\partial\{(\frac{v}{pr}+\frac{v}{prturb})\frac{\partial uj}{\partial xi}\}}{\partial xi} \quad (3)$$

3.4. VISCOSITY

Viscosity is a measure of the resistance of a fluid which is being deformed by either shear stress or extensional stress. In other words, Viscosity is a measure of a fluid's resistance to flow. It describes the internal friction of a moving fluid. A fluid with large viscosity resists motion because it's molecular make up gives it a lot of internal friction. In general, in any flow, layers move at different velocities and the fluid's viscosity arises from the shear stress between the layers that ultimately oppose any applied force. Isaac Newton postulated that, for straight, parallel and uniform flow, the shear stress, τ, between layers is proportional to the velocity gradient ∂u /∂y, in the direction perpendicular to the layers. In fact, due to the air is a viscous fluid airplane can fly. If the air had no viscosity, the fluid layers not travel attached to the wing surface, and not show the forces that allow the flight

3.5. REYNOLDS NUMBER

The Reynolds number relates the density, viscosity, speed and size of a typical flow in a dimension less expression, which is involved in many fluid dynamics problems. This dimensionless number or combination appears in many cases related to the fact that laminar flow can be seen (small Reynolds number) or turbulent (Reynolds number largest). From a mathematical point of view the Reynolds number of a problem or situation is defined by the following equation:

$$Re = \frac{\rho VL}{\mu} = \frac{VL}{v} = \frac{QL}{vA}$$

where:
V is the mean fluid velocity (m/s)
L is a characteristic linear dimension (m)
μ is the dynamic viscosity of the fluid (Pa*s or N・s/m2 or kg/(m*s))
ν is the kinematic viscosity (ν = μ / ρ) (m2/s)
ρ is the density of the fluid (kg/m3)

3.6. BERNOULLI'S PRINCIPLE

Bernoulli's Principle, also known as the triad of Bernoulli or Bernoulli's equation describes the behavior of a fluid moving along a streamline. It states that an ideal fluid without viscosity or friction, running through a closed pipeline, total energy remains constant throughout its length. This means an increase in flow speed lead to a reduction of pressure, and conversely, if the flow speed is reduced the pressure increases.
The Bernoulli equation is given by: V^2

$$\frac{V^2 \times \rho}{2} + p + \rho \times g \times z = constsnt$$

4. EXISTING SYSTEM

The existing system consist of a double pipe heat exchanger at the VCR evaporator output. Temp of the compressed air at the entry to the Evaporator is higher.

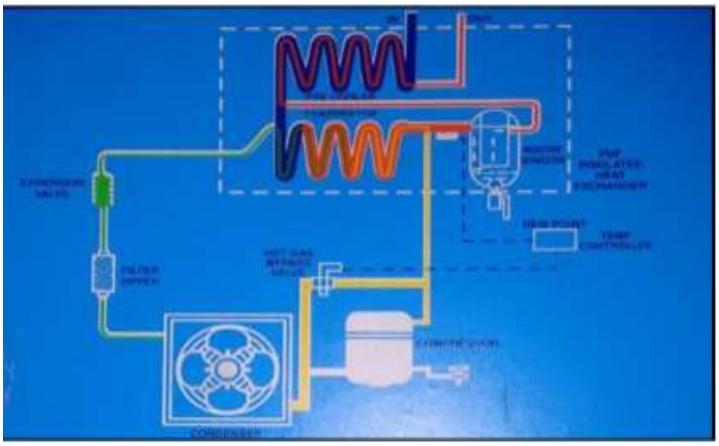

Fig -4.1: Schematic diagram of existing system

4.1 WORKING

The following are the steps takes place during the working of existing system:

1. The compressed air from the compressor is allowed in and passes through the air-to-refrigerant heat exchanger via additional double pipe air-to-air heat exchanger. Thus the air in the air-to-refrigerant heat exchanger (same as the evaporator of the VCR system) there are two fluids:
   - The hot compressed air from outside compressor

- The cold refrigerant in the VCR system

2. The cold refrigerant absorbs heat from the hot compressed air and thus condensation takes place and the moisture present in the compressed air is converted into droplets.
3. Then the cold compressed air is allowed to pass through the moisture separator where the droplets are removed and cold dry compressed air is the output.
4. This cold dry compressed air then allowed to pass through the additional air-to-air heat exchanger. Thus, the air-to-air heat exchanger has two fluids:
    - The outside hot compressed air from the compressor
    - The cold dry compressed air from the moisture separator
5. This cold dry compressed air absorbs heat from the hot compressed air in and attains ambient temperature and this ambient hot dry compressed air is passed out as the final output.
6. At the same time the heat of the compressed air from compressor (outside) will decrease and thus pre-cooled wet compressed air reaches the air-to-refrigerant heat exchanger and Step 2 and 3 repeats.
7. But the difference is, the compressed air inlet to the evaporator is already pre-cooled so the amount of heat transferred from pre-cooled wet compressed air-to-refrigerant is less.
8. Then Step 4 and 5 repeats.

4.2 DRAWBACKS

1. The output air is not perfectly dry.
2. The COP of the system is low.
3. Heat transfer in the double pipe air to air heat exchanger is low.

4.3 OBJECTIVE

1. Experiments are done to reduce the moisture content of dry air at VCR evaporator output.
2. To improve the COP of the existing system
3. To increase the amount of heat transferred inside the air to air heat exchanger.

4.4 MODIFICATIONS

1. Replace the double pipe air to air heat exchanger by a shell and tube heat exchanger (STHE).
2. Increase the number of coil tubes inside the heat exchanger using steel tubes
3. Use helical baffles
4. Modification is analyzed using CFD analysis

5. COMPUTATIONAL FLUID DYNAMICS

5.1 INTRODUCTION

CFD is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the millions of calculations required to simulate the interaction of fluids and gases with the complex surfaces used in engineering. However, even with simplified equations and high-speed supercomputers, only approximate solutions can be achieved in many cases. More accurate codes that can accurately and quickly simulate even complex scenarios such as supersonic or turbulent flows are an ongoing area of research.

The physical aspects of any fluid flow are governed by three fundamental principles:

1. Conservation of Mass (i.e. Continuity Equation)
2. Newton’s second law (force = rate of change of momentum)
3. Conservation of Energy (Energy equation)

These fundamental principles are expressed in terms of basic mathematical equations, which generally are either integral equations or partial differential equations. CFD is the art of replacing the integrals or the partial derivatives in these equations with discretized algebraic forms, which in turn are solved to obtain numerical values for the flow field at discrete points in time and/or space.

5.2 DISCRETIZATION METHODS

There are three discretization methods in CFD:

1. Finite difference method (FDM)
2. Finite volume method (FVM)
3. Finite element method (FEM)

5.2.1 FINITE DIFFERENCE METHOD (FDM)

A finite difference method (FDM) discretization is based upon the differential form of the partial differential equation to be solved. Each derivative is replaced with an approximate difference formula (that can generally be derived from a Taylor series expansion). The computational domain is usually divided into hexahedral cells (the grid), and the solution will be obtained at each nodal point. The FDM is easiest to understand when the physical grid is Cartesian, but through the use of curvilinear transforms the method can be extended to domains that are not easily represented by brick-shaped elements. The discretization results in a system of equation of the variable at nodal points, and once a solution is found, then we have a discrete representation of the solution.

5.2.2 FINITE VOLUME METHOD (FVM)

A finite volume method (FVM) discretization is based upon an integral form of the partial differential equation to be solved (e.g. conservation of mass, momentum, or energy). The partial differential equation is written in a form which can be solved for a given finite volume (or cell). The computational domain is discretized into finite volumes and then for every volume the 12 governing equations are solved. The resulting system of equations usually involves fluxes of the conserved variable, and thus the calculation of fluxes is very important in FVM. The basic advantage of this method over FDM is, it does not require the use of structured grids, and the effort to convert the given mesh in to structured numerical grid internally is completely avoided. As with FDM, the resulting approximate solution is a discrete, but the variables are typically placed at cell centers rather than at nodal points. This is not always true, as there are also face- centered finite volume methods. In any case, the values of field variables at non-storage locations (e.g. vertices) are obtained using interpolation.

5.2.3 FINITE ELEMENT METHOD (FEM)

A finite element method (FEM) discretization is based upon a piecewise representation of the solution in terms of specified basis functions. The computational domain is divided up into smaller domains (finite elements) and the solution in each element is constructed from the basic functions. The actual equations that are solved are typically obtained by restating the conservation equation in weak form: the field variables are written in terms of the basis functions, the equation is multiplied by appropriate test functions, and then integrated over an element. Since the FEM solution is in terms of specific basis functions, a great deal more is known about the solution than for either FDM or FVM. This can be a double-edged sword, as the choice of basic functions is very important and boundary conditions may be more difficult to formulate. Again, a system of equations is obtained (usually for nodal values) that must be solved to obtain a solution.

Comparison of the three methods is difficult, primarily due to the many variations of all three methods. FVM and FDM provide discrete solutions, while FEM provides a continuous (up to a point) solution. FVM and FDM are generally considered easier to program than FEM, but opinions vary on this point. FVM are generally expected to provide better conservation properties, but opinions vary on this point also.

5.3 CFD PROCEDURE

CFD codes are structured around the numerical algorithms that can be tackle fluid problems. In order to provide easy access to their solving power all commercial CFD packages include sophisticated user interfaces input problem parameters and to examine the results. Hence all codes contain three main elements:

1. Pre-processing
2. Solver
3. Post-processing

5.3.1 PRE-PROCESSING

This is the first step in building and analyzing a flow model. Preprocessor consist of input of a flow problem by means of an operator –friendly interface and subsequent transformation of this input into form of suitable for the use by the solver. The user activities at the Pre-processing stage involve:

- Definition of the geometry of the region: The computational domain.
- Grid generation the subdivision of the domain into a number of smaller, non overlapping sub domains (or control volumes or elements Selection of physical or chemical phenomena that need to be modeled).
- Definition of fluid properties.
- Specification of appropriate boundary conditions at cells, which coincide with or touch the boundary. The solution of a flow problem (velocity, pressure, temperature etc.) is defined at nodes inside each cell. The accuracy of CFD solutions is governed by number of cells in the grid. In general, the larger numbers of cells better the solution accuracy. Both the accuracy of the solution & its cost in terms of necessary computer hardware & calculation time are dependent on the fineness of the grid. Efforts are underway to develop CFD codes with a (self) adaptive meshing capability. Ultimately such programs will automatically refine the grid in areas of rapid variation.

5.3.2 SOLVER

The CFD solver does the flow calculations and produces the results. FLUENT, FloWizard, FIDAP, CFX and POLYFLOW are some of the types of solvers. FLUENT is used in most industries. FloWizard is the first general-purpose rapid flow modeling tool for design and process engineers built by Fluent. POLYFLOW (and FIDAP) are also used in a wide range of fields, with emphasis on the materials processing industries. FLUENT and CFX two solvers were developed independently by ANSYS and have a number of things in common, but they also have some significant differences. Both are control-volume based for high accuracy and rely heavily on a pressure-based solution technique for broad applicability. They differ mainly in the way they integrate the fluid flow equations and in their equation solution strategies. The CFX solver uses finite elements (cell vertex numeric), similar to those used in mechanical analysis, to discretize the domain. In contrast, the FLUENT solver uses finite volumes (cell centered (coupled algebraic multigrid), while the

FLUENT product offers several solution approaches (density-, segregated- and coupled-pressure-based methods). The FLUENT CFD code has extensive interactivity, so we can make changes to the analysis at any time during the process. This saves time and enables to refine designs more efficiently. Graphical user interface (GUI) is intuitive, which helps to shorten the learning curve and make the modeling process faster. In addition, FLUENT's adaptive and dynamic mesh capability is unique and works with a wide range of physical models. This capability makes it possible and simple to model complex moving objects in relation to flow. This solver provides the broadest range of rigorous physical models that have been validated against industrial scale applications, so we can accurately simulate real-world conditions, including multiphase flows, reacting flows, rotating equipment, moving and deforming objects, turbulence, radiation, acoustics and dynamic meshing. The FLUENT solver has repeatedly proven to be fast and reliable for a wide range of CFD applications. The speed to solution is faster because suite of software enables us to stay within one interface from geometry building through the solution process, to post-processing and final output. The numerical solution of Navier–Stokes equations in CFD codes usually implies a discretization method: it means that derivatives in partial differential equations are approximated by algebraic expressions which can be alternatively obtained by means of the finite-difference or the finite-element method. Otherwise, in a way that is completely different from the previous one, the discretization equations can be derived from the integral form of the conservation equations: this approach, known as the finite volume method, is implemented in FLUENT, because of its adaptability to a wide variety of grid structures. The result is a set of algebraic equations through which mass, momentum, and energy transport are predicted at discrete points in the domain. In the freeboard model that is being described, the segregated solver has been chosen so the governing equations are solved sequentially. Because the governing equations are non-linear and coupled, several iterations of the solution loop must be performed before a converged solution is obtained and each of the iteration is carried out as follows:

- Fluid properties are updated in relation to the current solution; if the calculation is at the first iteration, the fluid properties are updated consistent with the initialized solution.
- The three momentum equations are solved consecutively using the current value for pressure so as to update the velocity field.
- Since the velocities obtained in the previous step may not satisfy the continuity equation, one more equation for the pressure correction is derived from the continuity equation and the linearized momentum equations: once solved, it gives the correct pressure so that continuity is satisfied. The pressure–velocity coupling is made by the SIMPLE algorithm, as in FLUENT default options.
- Other equations for scalar quantities such as turbulence, chemical species and radiation are solved using the previously updated value of the other variables; when inter-phase coupling is to be considered, the source terms in the appropriate continuous phase equations have to be updated with a discrete phase trajectory calculation.
- Finally, the convergence of the equations set is checked and all the procedure is repeated until convergence criteria are met.

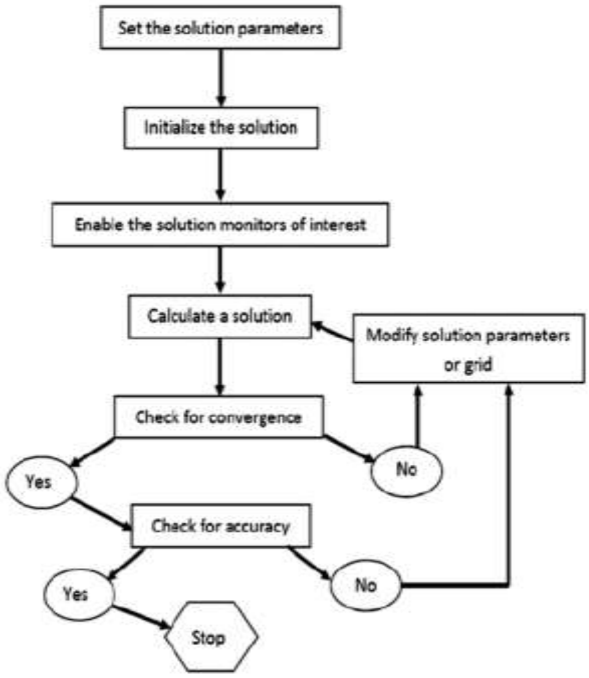

Fig -5.1: Algorithm of numerical approach used by simulation software

The conservation equations are linearized according to the implicit scheme with respect to the dependent variable: the result is a system of linear equations (with one equation for each cell in the domain) that can be solved simultaneously. Briefly, the segregated implicit method calculates every single variable field considering all the cells at the same time. The code stores discrete values of each scalar quantity at the cell centre; the face values must be interpolated from the cell centre values. For all the scalar quantities, the interpolation is carried out by the second order upwind scheme with the purpose of achieving high order accuracy. The only exception is represented by pressure interpolation, for which the standard method has been chosen.

5.3.3 POST-PROCESSING

This is the final step in CFD analysis, and it involves the organization and interpretation of the predicted flow data and the production of CFD images and animations. Fluent's software includes full post processing capabilities. FLUENT exports CFD's data to third-party post-processors and visualization tools such as Ensight, Fieldview and TechPlot as well as to VRML formats. In addition, FLUENT CFD solutions are easily coupled with structural codes such as ABAQUS, MSC and ANSYS, as well as to other engineering process simulation tools. Thus, FLUENT is general-purpose computational fluid dynamics (CFD) software ideally suited for incompressible and mildly compressible flows. Utilizing a pressure-based segregated finite-volume method solver, FLUENT contains physical models for a wide range of applications including turbulent flows, heat transfer, reacting flows, chemical mixing, combustion, and multiphase flows. FLUENT provides physical models on unstructured meshes, bringing you the benefits of easier problem setup and greater accuracy using solution-adaptation of the mesh. FLUENT is a computational fluid dynamics (CFD) software package to simulate fluid flow problems. It uses the finite-volume method to solve the governing equations for a fluid. It provides the capability to use different physical models such as incompressible or compressible, inviscid or viscous, laminar or turbulent, etc. Geometry and grid generation is done using GAMBIT which is the preprocessor bundled with FLUENT. Owing to increased popularity of engineering work stations, many of which has outstanding graphics capabilities, the leading CFD are now equipped with versatile data visualization tools. These include:

- Domain geometry & Grid display.
- Vector plots.
- Line & shaded contour plots.
- 2D & 3D surface plots.
- Particle tracking.
- View manipulation (translation, rotation, scaling etc.)

5.4 DIFFERENTIAL EQUATION USED IN CFD

5.4.1 NAVIER-STOKES EQUATIONS:

The ''Navier-Stokes equations'', named after Claude-Louis Navier and George Gabriel Stokes, describe the motion of fluid substances. These equations arise from applying Newton's second law to fluid motion, together with the assumption that the fluid stress is the sum of a diffusing viscous term (proportional to the gradient of velocity), plus a pressure term.

The fundamental of almost all CFD problems are the Navier–Stokes equations, which define any single-phase fluid flow. These equations can be simplified by removing terms describing viscosity to yield the Euler equations.

The Navier-Stokes equations dictate not position but rather velocity. A solution of the Navier-Stokes equations is called a velocity field or flow field, which is a description of the velocity of the fluid at a given point in space and time. Once the velocity field is solved for, other quantities of interest (such as flow rate or drag force) may be found.

The derivation of the Navier–Stokes equations begins with an application of Newton's second law: conservation of momentum (often alongside mass and energy conservation) being written for an arbitrary portion of the fluid.
Navier- Stokes equations in conservation form: -
In x direction as

$$\frac{\partial(\rho u)}{\partial t}+\frac{\partial(\rho u^2)}{\partial x}+\frac{\partial(\rho uv)}{\partial y}+\frac{\partial(\rho uw)}{\partial z}=-\frac{\partial p}{\partial x}+\frac{\partial}{\partial x}\left(\lambda\nabla\cdot\mathbf{V}+2\mu\frac{\partial u}{\partial x}\right)+\frac{\partial}{\partial y}\left[\mu\left(\frac{\partial v}{\partial x}+\frac{\partial u}{\partial y}\right)\right]+\frac{\partial}{\partial z}\left[\mu\left(\frac{\partial u}{\partial z}+\frac{\partial w}{\partial x}\right)\right]+\rho f_x$$

Fig -5.2: Navier-Stocks Equation

Similarly we can write in y and z direction also.

The Navier–Stokes equations are nonlinear partial differential equations in almost every real situation. In some cases, such as one-dimensional flow and Stokes flow (or creeping flow), the equations can be simplified to linear equations. The nonlinearity makes most problems difficult or impossible to solve and is the main contributor to the turbulence that the equations model.

The nonlinearity is due to convective acceleration, which is an acceleration associated with the change in velocity over position. Hence, any convective flow, whether turbulent or not, will involve nonlinearity.

The numerical solution of the Navier-Stokes equations for turbulent flow is extremely difficult, and due to the significantly different mixing-length scales that are involved in turbulent flow, the stable solution of this requires such a fine mesh resolution that the computational time becomes significantly infeasible for calculation. Attempts to solve turbulent flow using a laminar solver typically result in a time-unsteady solution, which fails to converge appropriately. To counter this, time-averaged equations such as the Reynolds-averaged Navier–Stokes equations (RANS), supplemented with turbulence models, are used in practical computational fluid dynamics (CFD) applications when modeling turbulent flows. Some models include the Spalart-Allmaras, k-ω (k-omega), k-ε (k-epsilon), and SST models which add a variety of additional equations to bring closure to the RANS equations.

5.4.2 REYNOLDS-AVERAGED NAVIER-STOKES EQUATIONS

The Reynolds-averaged Navier–Stokes equations (or RANS equations) are time-averaged equations of motion for fluid flow. The idea behind the equations is Reynolds decomposition, whereby an instantaneous quantity is decomposed into its time-averaged and fluctuating quantities, an idea first proposed by Osborne Reynolds. The RANS equations are primarily used to describe turbulent flows. These equations can be used with approximations based on knowledge of the properties of flow turbulence to give approximate time-averaged solutions to the Navier–Stokes equations.

The left-hand side of this equation represents the change in mean momentum of fluid element owing to the unsteadiness in the mean flow and the convection by the mean flow. This change is balanced by the mean body force, the isotropic stress owing to the mean pressure field, the viscous stresses, and apparent stress owing to the fluctuating velocity field, generally referred to as the Reynolds stress. This nonlinear Reynolds stress term requires additional modeling to close the RANS equation for solving and has led to the creation of many different turbulence models.

5.5 ADVANTAGES OF CFD

Major advancements in the area of gas-solid multiphase flow modeling offer substantial process improvements that have the potential to significantly improve process plant operations. Prediction of gas solid flow fields, in processes such as pneumatic transport lines, risers, fluidized bed reactors, hoppers and precipitators are crucial to the operation of most process plants. Up to now, the inability to accurately model these interactions has limited the role that simulation could play in improving operations. In recent years, computational fluid dynamics (CFD) software developers have focused on this area to develop new modeling methods that can simulate gas-liquid-solid flows to a much higher level of reliability. As a result, process industry engineers are beginning to utilize these methods to make major improvements by evaluating alternatives that would be, if not impossible, too expensive or time-consuming to trial on the plant floor. Over the past few decades, CFD has been used to improve process design by allowing engineers to simulate the performance of alternative configurations, eliminating guesswork that would normally be used to establish equipment geometry and process conditions. The use of CFD enables engineers to obtain solutions for problems with complex geometry and boundary conditions. Advantages of CFD can be summarized as:

1. It provides the flexibility to change design parameters without the expense of hardware changes. It therefore costs less than laboratory or field experiments, allowing engineers to try more alternative designs than would be feasible otherwise.
2. It has a faster turnaround time than experiments.
3. It guides the engineer to the root of problems and is therefore well suited for trouble-shooting.
4. It provides comprehensive information about a flow field, especially in regions where measurements are either difficult or impossible to obtain.

6. VALIDATION

Validation of software is done using the journal ‘Computational fluid dynamics analysis of shell and tube heat exchanger’ proposed by Gurbir Singh [5].

6.1 Geometry

Dimensions of shell and tube heat exchanger
- Length of pipe = 1610 mm

Inner tube: Material =SS
- Inner diameter = 9.5mm
- Outer diameter = 12.7 mm

Outer tube: Material GI
- Inner diameter = 28 mm
- outer diameter = 33.8 mm
- Minimum distance of cold water nozzle from pipe end = 20 mm
- Cold water nozzle inner diameter = 16.8 mm
- Flow considered: Couter flow.
- Flow passage:
- Shell: Cold fluid
- Tube: Hot fluid
- Notations used
- T_{hi} – Inlet temperature of hot water
- T_{ho}- Outlet temperature of hot water
- T_{ci}- Inlet temperature of cold water
- T_{co}- Outlet temperature of cold water

Boundary conditions
- Flow rate = 2 L/min = 2x0.017 = 0.34 kg/sec
- Shell inlet temperature = 294.1 K
- Tube inlet temperature = 330.4 K

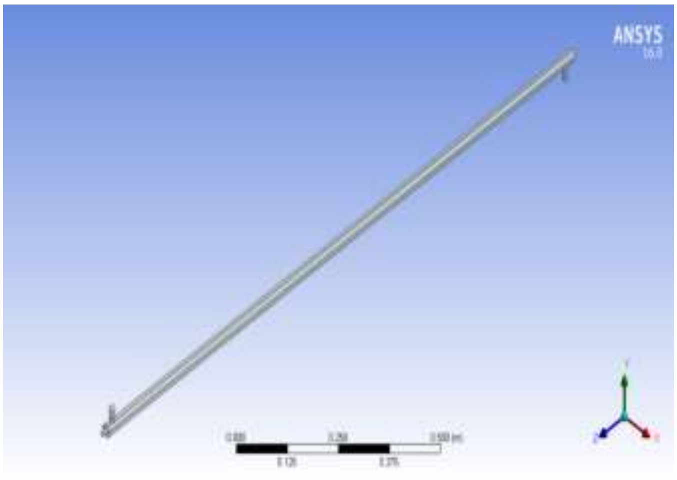

Fig -6.1: Geometry

6.2 MESHING

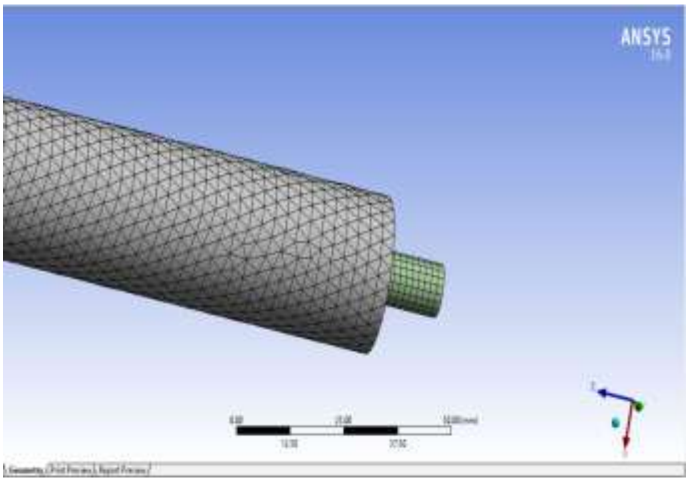

Fig -6.2: Face meshing

Statistics	
Nodes	285067
Elements	541407
Mesh Metric	Orthogonal Quality
Min	0.16349
Max	0.99782
Average	0.78263
Standard Devi...	0.22989

Fig -6.3: Mesh Statistics

6.3 MODELS

Energy equation is turned on and k-epsilon model is selected

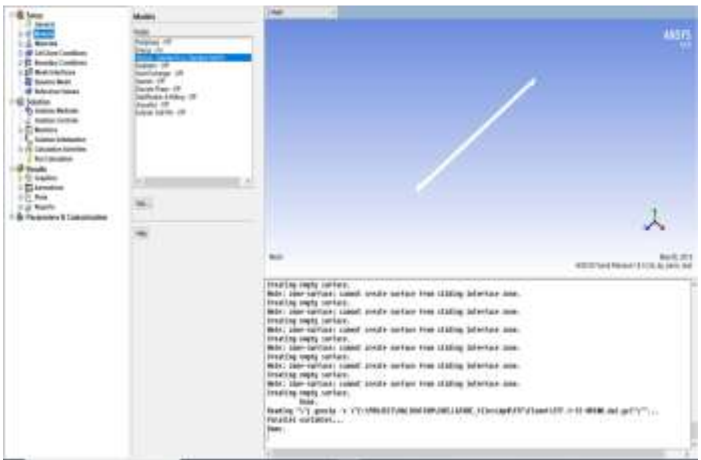

Fig -6.4: Energy and Viscous

6.4 MATERIAL SELECTION

Different material is selected for different cell bodies.

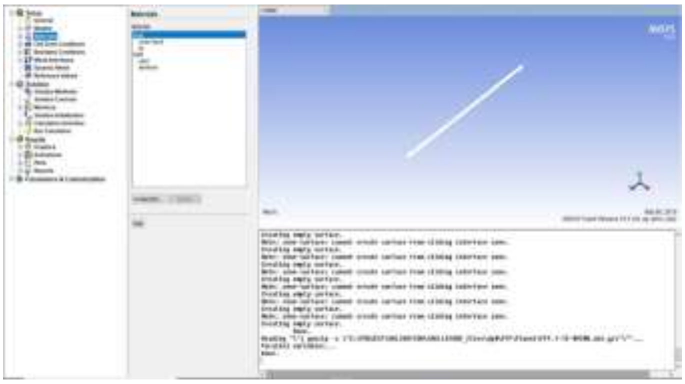

Fig -6.5: Solid and Fluid Material Selection

6.5 ASSIGNING CELL ZONE CONDITIONS

Material is assigned for cell zones.

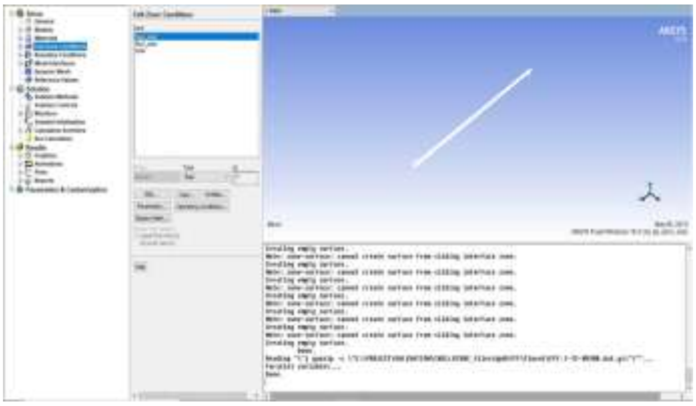

Fig -6.7: Materials Assigned for Different Bodies

6.6 SOLUTION INITIALISATION

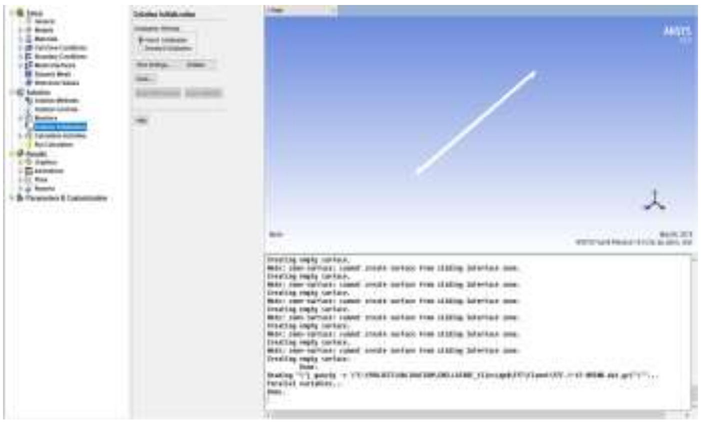

Fig -6.8: Hybrid Initialization

6.7 RUN CALCULATIONS

Iterations passed : 9309

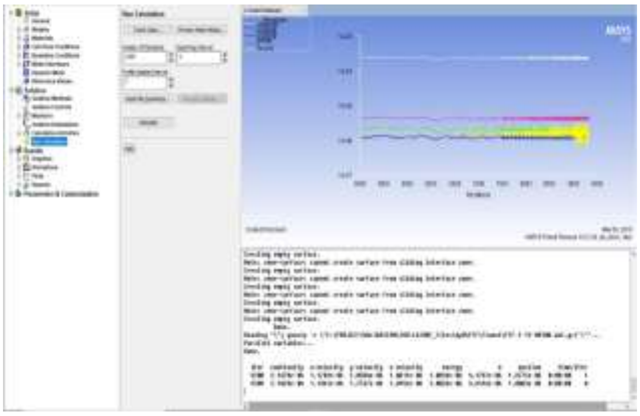

Fig -6.9: Running Iterations

6.8 RESULT

Shell outlet temperature : 301.31 K

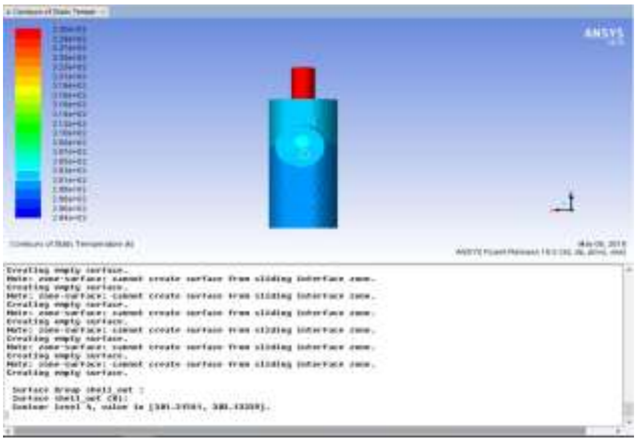

Fig -6.10: Shell outlet Temperature

Tube outlet temperature : 321.31K

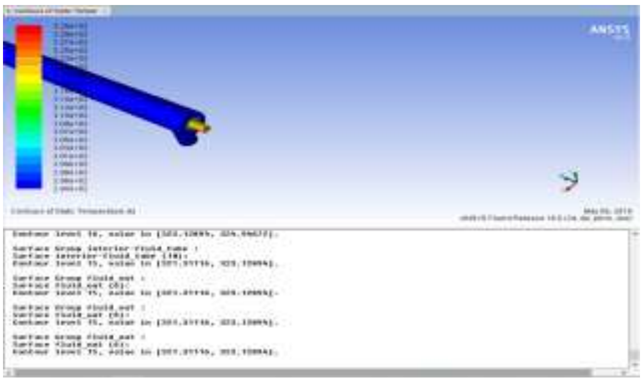

Fig -6.11: Tube outlet Temperatur

TAB -1: Result Comparison

	JOURNEL RESULT	VALIDATION RESULT	ERROR PERCENT AGE
SHELL OUTLET TEMPERATURE (k)	302.5	301.31	0.39 %
TUBE OUTLET TEMPERATURE (K)	320.1	321.3	0.37 %

7. DESIGNING OF SHELL AND TUBE HEAT EXCHANGER

7.1 GEOMETRY

Tab -2: Shell and Tube Parameters

Parameters	Specifications
Material	Stainless steel
Tube internal diameter	4 mm
Tube external diameter	6 mm
Tube arrangements	Triangular
Number of tubes	7
Tube effective length	184 mm
Shell internal diameter	44 mm
Baffle number	4

7.2 MESH

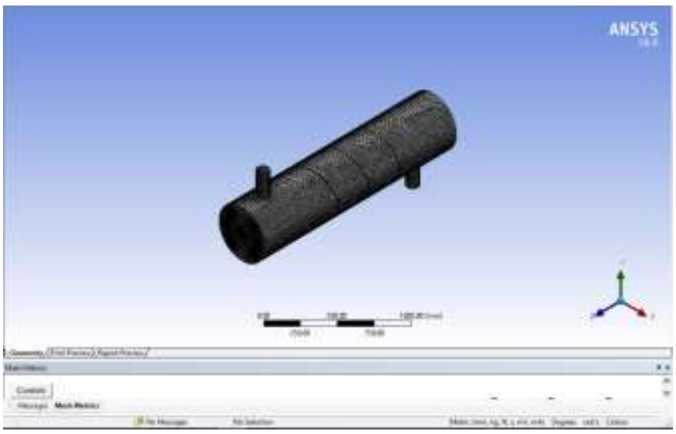

Fig -7.1: Meshing

Statistics	
Nodes	383047
Elements	993080
Mesh Metric	Orthogonal Quality
Min	0.23327
Max	0.99836
Average	0.85772
Standard Devi...	0.10315

Fig -7.2: Mesh Statistics

7.3 MODELS

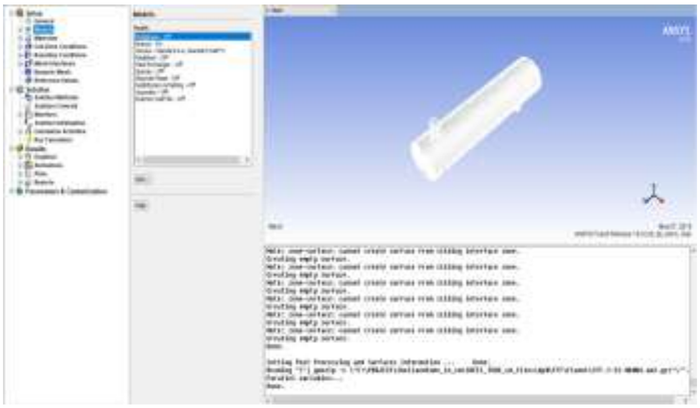

Fig -7.3: Models

7.4 MATERIALS

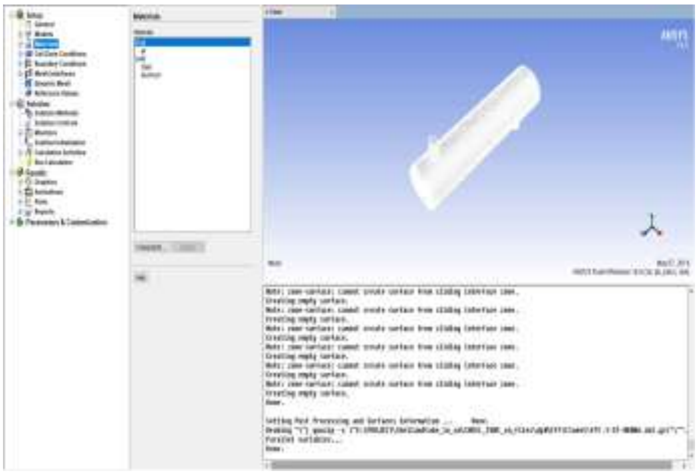

Fig -7.4: Materials

7.5 ASSIGNING CELL ZONE CONDITIONS

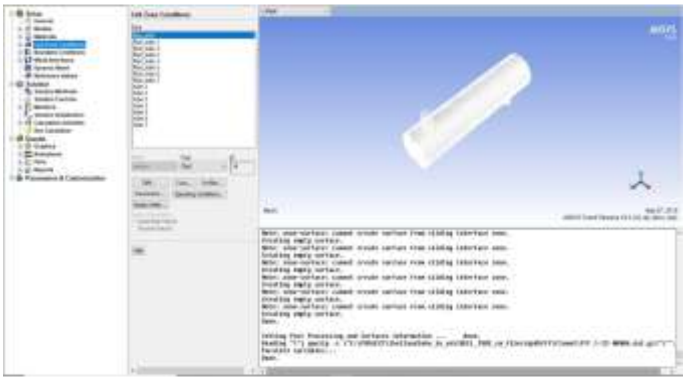

Fig -7.5: Boundary Conditions

- Flow rate :0.0104 kg/sec
- Tube inlet temperature = 278.15 K
- Shell inner temperature = 318.15 K

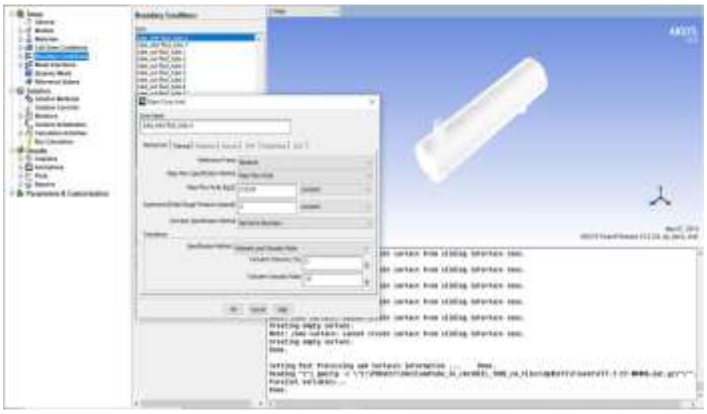

Fig -7.6: Boundary Conditions

7.6 SOLUTION INITIALISATION

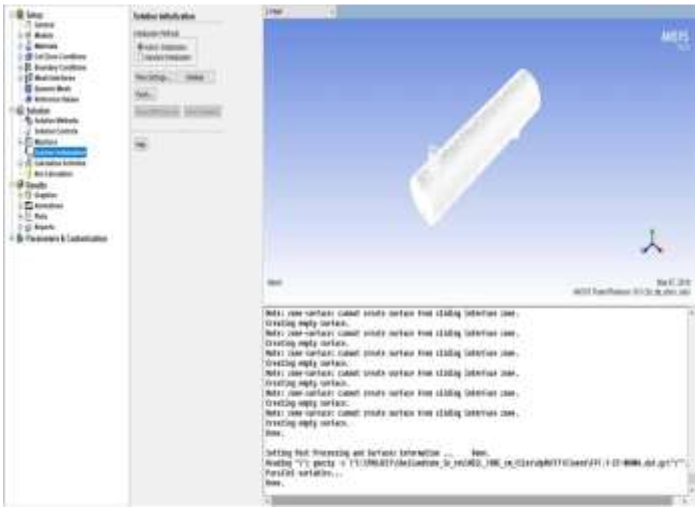

Fig -7.7: Hybrid Initialization

7.7 RUN CALCULATIONS

Iterations passed : 8090

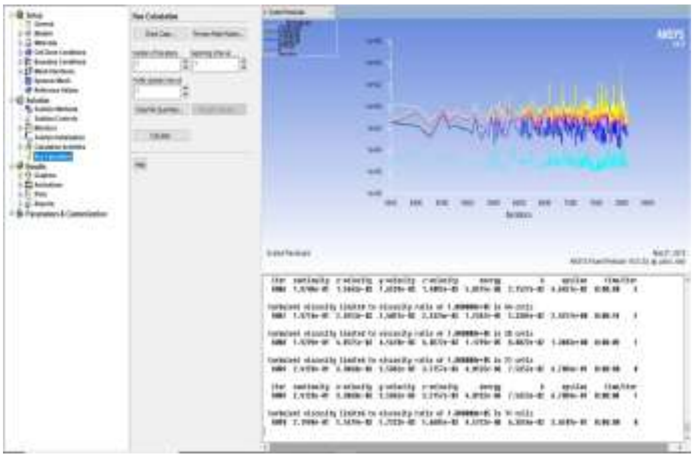

Fig -7.8: Iterations

7.8 RESULTS

Shell outlet temperature : 288.15 K

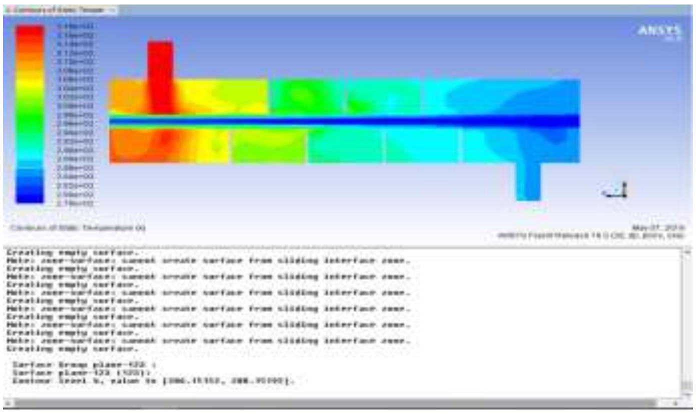

Fig -7.9: Shell Outlet Temperature

Tube outlet temperature : 282.15 K

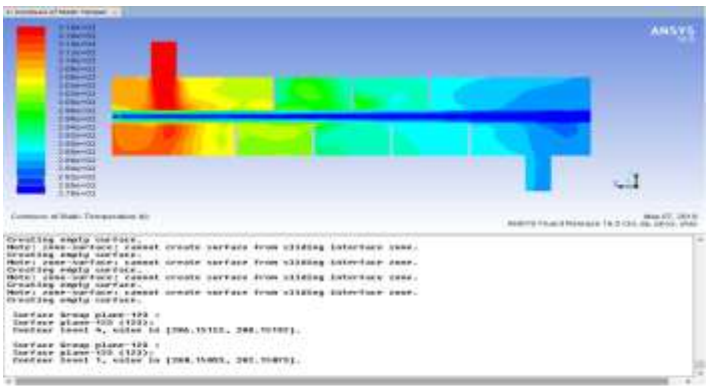

Fig -7.10: Tube Outlet Temperature

8. DESIGNING OF DOUBLE PIPE HEAT EXCHANGER

8.1 PARAMETERS OF SHELL AND TUBE HEAT EXCHANGERS

Parameters of shell and tube heat exchangers required to design double pipe heat exchanger are:

- Inner volume of 7 tubes $= \pi \times 2^2 \times 184 \times 7$
  $= 16185.4853\ cm^3$
- Outer volume of 7 tubes $= \pi \times 3^2 \times 184 \times 7$
  $= 36417.339\ cm^3$
- Shell total volume $= \pi \times 22^2 \times 184$
  $= 279777.675 cm^3$
- Effective shell volume = total shell vol – tube ext vol = 279777.675-36417.339

  $= 243360.336\ cm^3$

8.2 DESIGNING OF DOUBLE TUBE

- Taking inner tube inner radius = 6 cm
- Inner tube inner volume $= \pi \times 3^2 \times L$
  $= 16185.485\ cm^3$
- Free length of tube L = 572.44 cm
- Take inner tube outer radius = 4 cm
- Inner tube outer volume $= \pi \times 4^2 \times 572.444$
  $= 28774.1738\ cm^3$

Outer tube inner radius

- Effective outer tube volume =
  $\pi \times R^2 \times 572.44$ – inner tube external volume
- $\pi \times R^2 \times 572.44 - 28774.1738 = 243360.333\ cm^3$
- Outer tube inner radius R = 12.3 cm
- Double pipe pitch = 64 cm
- Double pipe helix diameter = 34 cm

8.3 GEOMETRY

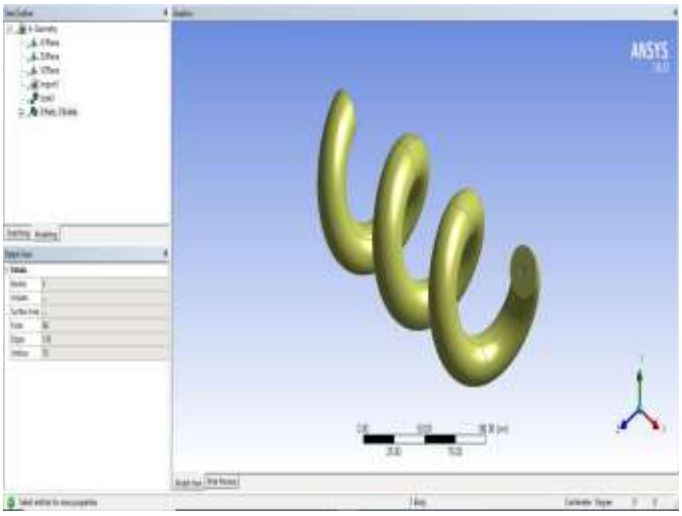

Fig -8.1: Geometry

8.4 MESHING

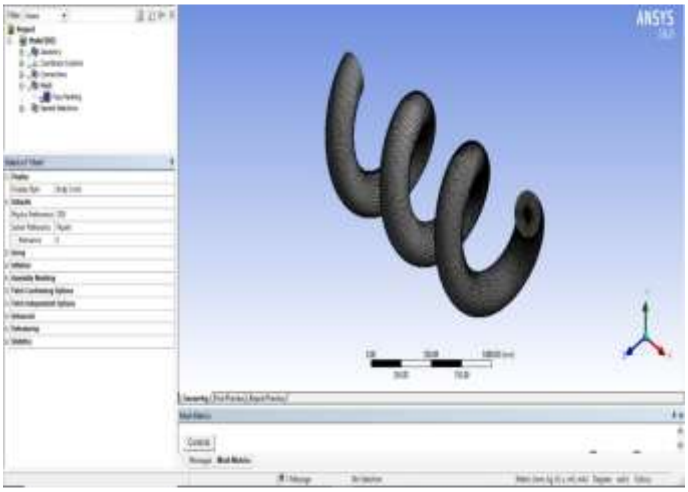

Fig -8.2: Meshing

Statistics	
Nodes	481391
Elements	942154
Mesh Metric	Orthogonal Quality
Min	0.11448
Max	0.99976
Average	0.80833
Standard Devi...	0.17364

Fig -8.3: Mesh Statistics

8.5 MODELS

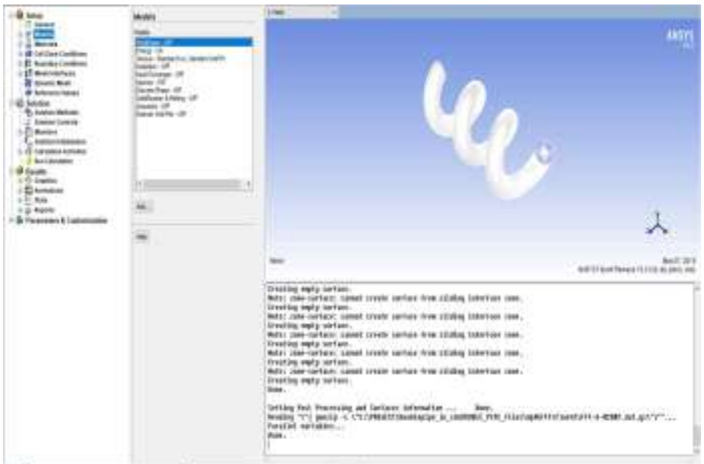

Fig -8.4: Models

8.6 MATERIALS

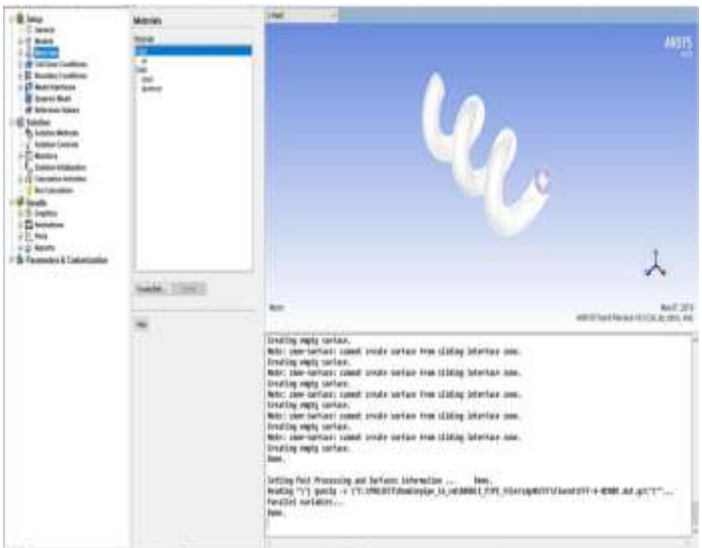

Fig -8.5: Materials

8.7 CELL ZONE CONDITIONS

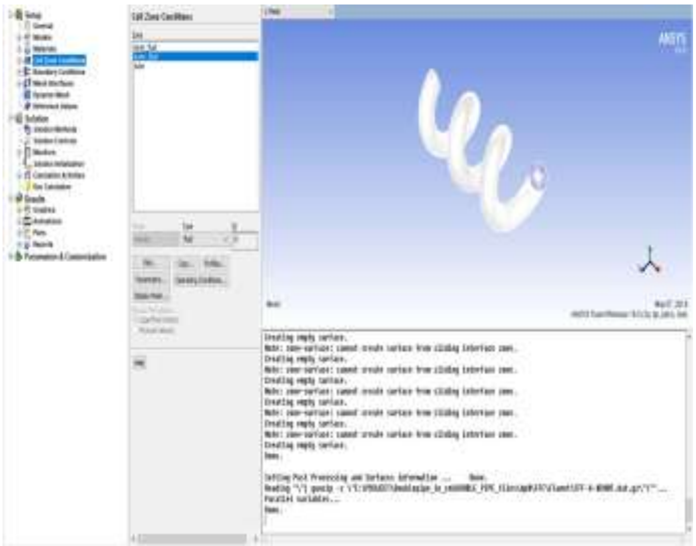

Fig -8.6: Cell Zone Conditions

BOUNDARY CONDITIONS

- Flow rate :0.0104 kg/sec
- Inner pipe inlet temperature = 278.15 K
- Outer pipe inner temperature = 318.15 K

8.8 SOLUTION INITIALIZATION

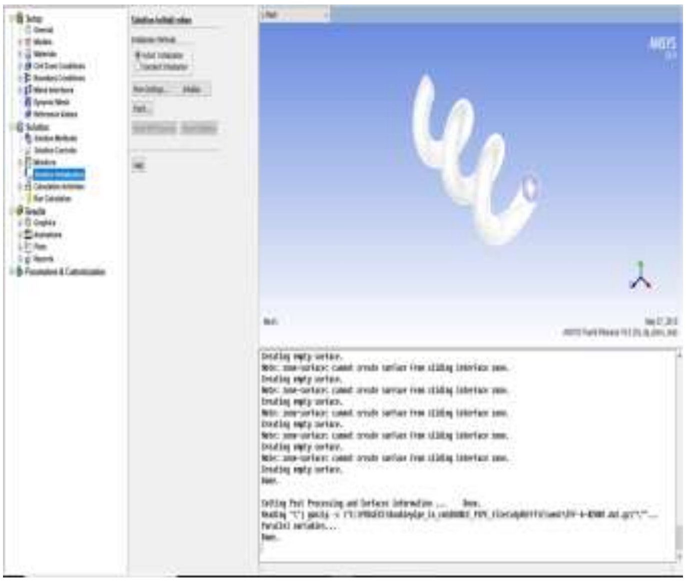

Fig -8.7: Hybrid Initialization

8.9 RUN CALCULATIONS

Iterations passed : 5252

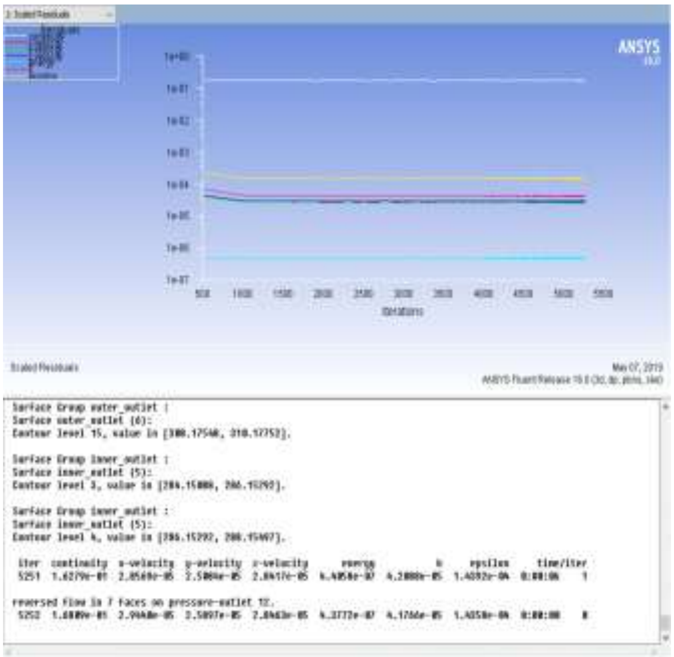

Fig -8.8: Iterations

8.10 RESULTS

Outer pipe outlet temperature : 308.17 K

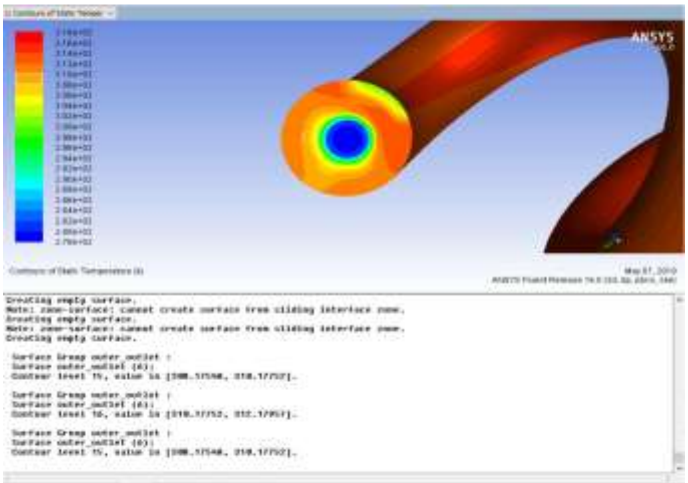

Fig -8.9: Outer Pipe Outlet Temperature

Inner pipe inlet temperature : 288.15 K

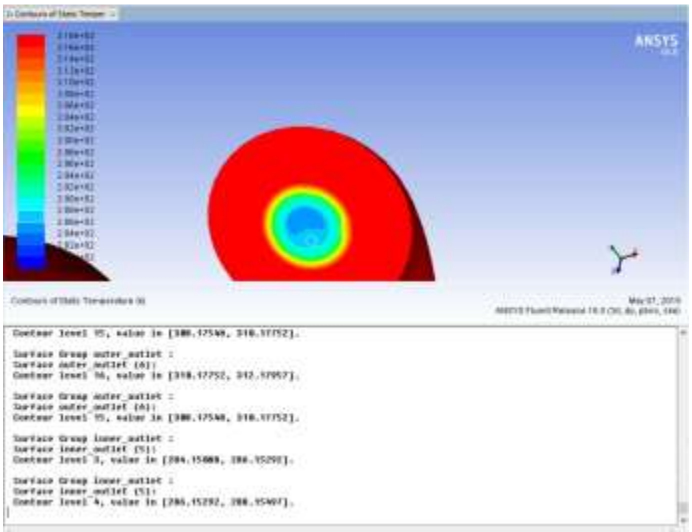

Fig -8.10: Inner Pipe Outlet Temperature

9. RESULT

9.1 COP CALCULATION

SHELL AND TUBE HEAT EXCHANGER

- Shell outlet temperature T_H = 288.18 K
- Tube inlet temperature T_C = 278.15 K
- COP = T_H / (T_H - T_C)
  = 288.18 / (288.15-278.15)
  = **28.7**

DOUBLE TUBE HEAT EXCHANGER

- Outer tube outlet temperature T_H = 308.17 K
- Inner tube inlet temperature T_C = 278.15 K

- COP $= T_H / (T_H - T_C)$
  $= 308.17 / (308.17-278.15)$
  $= \underline{\mathbf{10.26}}$

10. CONCLUSION

The temperature obtained at the outlet of outer tube in shell and tube heat exchanger is 308 K. Which on calculation give COP as 10.28.

But the outlet of shell in shell and tube heat exchanger is 288.15 K which in turn gives a COP of 28.

From the obtained data we could easily conclude that shell and tube heat exchanger is more efficient than double pipe heat exchanger as it is able to deliver more than twice COP.

So for dry air production shell and tube heat exchanger can be used as a replacement for double pipe heat exchanger for obtaining higher COP values.

REFERENCES

1. Prabal Roy, Surjeet Singh Rajpoot, Improvement of Efficiency of Air Refrigeration System by lowering the Inlet Temperature of Air, International Journal of Engineering Trends and Technology (IJETT) – Volume 55 Number 1 Januar2018.
2. Shambhu Kumar Rai and Parmeshwar Dubey "A REVIEW ON HEAT EXCHANGER" in Vol-3 Issue-1, 2017 IJARIIE-ISSN(O)-2395-4396 3678.
3. Mahmood Aslam Bhutta, Nasir Hayat, CFD applications in various heat exchangers design: A review , Applied Thermal Engineering 32 (2012) 1e12.
4. Ammar Ali Abd, Mohammed Qasim Kareem, Performance Analysis of Shell and Tube Heat Exchanger: Parametric Study, S2214-157X(17)30341-6.
5. Gurbir Singh, Hemant Kumar, Computational Fluid Dynamics Analysis of Shell and Tube Heat Exchanger, Journal of Civil Engineering and Environmental Technology Print ISSN: 2349-8404; Online ISSN: 2349-879X; Volume 1, Number 3; August, 2014 pp. 66-70 © Krishi.
6. Pranita Bichkar, Study of shell and tube heat exchanger with effect of types of baffles. Procedia manufacturing 20 (2018).
7. Ravi Kumar Banjare, 'A Paper on The Analysis of Effect of Material Used in Heat Exchanger and Its Performance', International Journal of Research in Advent Technology, Vol.3, No.10, October 2015 E-ISSN: 2321-9637 Available online at www.ijrat.org
8. Li He 'Numerical Investigation on Double Tube-Pass Shell-And-Tube Heat Exchangers with Different Baffle Configurations', S1359-4311(17)35396-6 DOI: https://doi.org/10.1016/j.applthermaleng.2018.07.098 Reference: ATE 12456.
9. Usman Salahuddin 'A Review of The Advancements Made in Helical Baffles Used in Shell and Tube Heat Exchangers International Communications in Heat and Mass Transfer 67 (2015) 104–108.
10. N. Piroozfam 'Numerical Investigation of Three Methods for Improving Heat Transfer in Counter Flow Heat Exchangers' International Journal of Thermal Sciences 133 (2018) 230–239
11. Dipankar Del [11] in his journal 'Helical Baffle Design in Shell and Tube Type Heat Exchanger with CFD Analysis'

BIOGRAPHIES

SACHU PRASAD
Student, Department of Mechanical Engineering, Musaliar College of Engineering and Technology, Pathanamthitta, Kerala, India.

MELVIN M THOMAS
Student, Department of Mechanical Engineering, Musaliar College of Engineering and Technology, Pathanamthitta, Kerala, India.

LINCE MATHEW THOMAS
Student, Department of Mechanical Engineering, Musaliar College of Engineering and Technology, Pathanamthitta, Kerala, India.

RAHUL REGHU
Student, Department of Mechanical Engineering, Musaliar College of Engineering and Technology, Pathanamthitta, Kerala, India.

AJAI M
Assistant Professor, Department of Mechanical Engineering, Musaliar College of Engineering and Technology, Pathanamthitta, Kerala, India.