

Parametric Study of a Shell and Tube Heat Exchanger Using CFD

A.Vennila

B.E. Mechanical Engineering
SNS College of Technology
Coimbatore, India

A.M. Fevija Dhas

B.E. Mechanical Engineering
SNS College of Technology
Coimbatore, India

Karthick Mahadev Shivpuje

B.E. Mechanical Engineering
SNS College Of Technology
Coimbatore, India

Gokulanaath S.

B.E. Mechanical Engineering
SNS College Of Technology
Coimbatore, India

Abstract- Shell and tube heat exchanger are one of the major component's in most of the process industries, these systems that are available in different sizes has various parts. The outer shell through which the hot fluid flows, the inner tube domain in which the cold fluid flows are the major assemblies of the system. Though so many literatures are available in the design of shell and tube heat exchangers still it is a wide-open opportunity for designers to keep heat transfer rate maximum with minimum pressure drop of these systems. The arrangement of tubes, number of baffles, inclination of baffles, baffle pitch, tubes pitch, and baffle cut are some of the variables based on which the hydraulic and thermal performance of these systems are rely upon. In this work, a novel approach is attempted to conduct a parametric study to understand and identify the optimal level of about pointed out variables. Computational fluid dynamics which is found to be a better alternate for cumbersome, cost and time consuming experimental methodology is utilized in this study.

Keywords - Heat exchanger, counter flow, parallel flow, computational fluid dynamics, ansys.

INTRODUCTION

Fluid dynamics is a field of science which studies the physical laws governing the flow of fluids under various conditions. Great effort has gone into understanding the governing loss and the nature of fluids themselves, resulting in a complex yet theoretically strong field of research. Computational Fluid Dynamics or CFD as it is popularly known is used to generate flow simulations with the help of computer. CFD involves the solution of the governing loss of fluid dynamics numerically. The complex sets of partial differential equation of solved on in geometrical domain divided into small volumes, commonly known as a mesh (or grid).

COMPUTATIONAL FLUIDDYNAMICS

Computational fluid dynamics(CFD)is the science of predicting fluid flow, heat transfer, mass transfer, chemical reactions, and related phenomena by solving the mathematical equations which govern these processes using a numerical process.

GOVERNING EQUATIONS OF CFD

Applying the fundamental laws of mechanics to a fluid gives the governing equations for a fluid. The conservation of mass equation and the conservation of momentum equation is these equations along with the conservation of energy equation form a set of coupled, nonlinear partial differential equations. It is not possible to solve these equations analytically for most engineering problems.

However, it is possible to obtain approximate computer-based solutions to the governing equations for a variety of engineering problems. This is the subject matter of Computational Fluid Dynamics (CFD).

STEPS FOLLOWED IN CFD

- Fluid domain extraction.
- Surface meshing.
- Volume mesh.
- Solving the CFD problem.
- Post processing.
- Report generating.

SOLVING THE CFD PROBLEM

- Reading the file. The reading the file should clear as case file or data file or case and data file. In this we have to read case and data file.
- Scaling the grid.
- Checking the grid.
- Defining the models- Model should define whether it is steady or unsteady.
- Steady and viscous.
- Defining the materials.
- Defining the boundary condition

- Controls
- Initialize
- Monitor
- Iterate

NUMERICAL ERRORS

- Solving equations on a computer invariably introduces numerical errors.
- Round-off error; due to finite word size available on the computer, Round-off errors will always exist (though they can be small in most cases).
- Truncation error: due to approximate in the numerical models. Truncation errors will go to zero as the grid is refined. Mesh refinement is one way to deal with truncation error.

BOUNDARY CONDITIONS

As with physical models, the accuracy of the CFD solution is only as good as the initial/boundary conditions provided to the numerical model.

Example: Flow in a duct with sudden expansion. If flow is supplied domain by a pipe, you should use a fully-developed profile for velocity rather than assume uniform conditions.

CFD software is used in this work to calculate the pressure drop and overall heat transfer coefficient for five models. The methodology and the results are discussed in the next chapters.

The governing equation of fluid motion may result in a solution when the boundary conditions and the initial conditions of specified. The form of boundary conditions that is required by any partial differential equation depends on the equation itself and the way that it has been discretized. Common boundary conditions are classified either in terms of the numerical value that have to be set or in terms of the physical type of boundary condition.

SOLID WALLS

Many boundaries within the fluid flow domain will be solid walls, and these can be either stationary or moving walls. If the flow is laminar then the velocity components can be set to be the velocity of walls. When the flow is turbulent, however, the situation is more complex.

INLETS

At an inlet, fluid enters the domain and therefore, its fluid velocity or pressure or the mass flow rate may be known. Also, the fluid may have certain characteristics, such as turbulence characterizes which need to specified.

SYMMETRY BOUNDARIES

When the flow is symmetrical about some plane there is no flow through the boundary and the derivatives of the variables normal to the boundary are zero.

CYCLIC OR PERIODIC BOUNDARIES

These boundaries come in pairs and are used to specify the flow has the same values of the variables at equivalent position and both of the boundaries.

PRESSURE BOUNDARY CONDITIONS

The ability to specify a pressure condition at one or more boundaries of a computational region is an important and useful computational tool. Pressure boundaries represent such things as confined reservoirs of fluid, ambient laboratory conditions and applied pressures arising from mechanical devices. Generally, pressure condition cannot be used at boundary where velocities are also specified, because velocities are influenced by pressure gradients.

The only exception is when pressures are necessary to specify the fluid properties. Density crossing a boundary conditions, referred to as static or stagnation pressure conditions. In a static condition the pressure is more or less continuous across the boundary and the velocity at the boundary is assigned value based on a zero normal-derivative condition across the boundary. In contrast, stagnation pressure condition assumes stagnation conditions outside the boundary so that the velocity at the boundary is zero. Since the static pressure condition says nothing about fluid outside the boundary (i.e., other than it is supposed to be the same as the velocity inside the boundary) it is less specific than the stagnation pressure condition. In this sense, the stagnation pressure condition is generally more physical is recommended for most applications.

OUTFLOW BOUNDARY CONDITIONS

In many simulations, there is need to have fluid flow out of one or more boundaries of the computational region. In compressible flow, when the flow speed at the outflow boundary is supersonic, it makes little difference how the boundary conditions are specified since flow disturbances cannot propagate upstream.

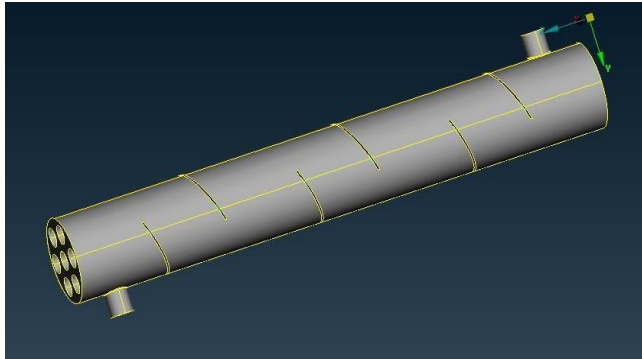
In low speed and incompressible flows, however, disturbances introduced at an outflow boundary can have an effect on the entire computational region. The simplest and most commonly used outflow condition is that of "continuative "boundary

Continuative boundary conditions consist of zero normal derivatives at the boundary for all quantities. The zero-derivative condition is intended to represent a smooth continuation of the flow through the boundary.

As a general rule, a physically meaningful boundary condition. Such as a specified pressure condition, should be used at out flow boundaries whenever possible. When a continuative condition is used it should be placed as far from the main flow region as is practical so that any influence on the main flow will be minimal.

OPENNING BOUNDARY CONDITION

If the fluid flow crosses the boundary surface in either direction an condition needs to be utilized. All of the



fluid might flow out of the domain, or into the domain, or a combination of the two might happen.

FREE SURFACES AND INTERFACES

If the fluid has a free surface, then the surface tension forces need to be considered. This requires utilization of the Laplace's equation which specifies the surface tension-induced jump in the normal stress p_s across the interface

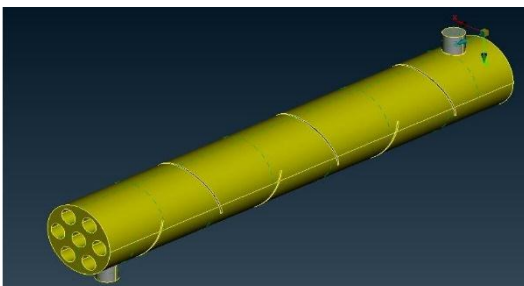
CFD PREPROCESSING

The pre-processing consists of the following:

BASE CASE

Geometric model is generated in 'SOLIDWORKS' which is very popular modelling software. The generated model is exported to the further process in the form of. IGES as it is a third- party format which can be taken in to any other tools. Extracting the fluid region is the next step in which all the surfaces which are in the contact of fluid are taken alone and all other surfaces are removed completely. To keep the domain air/water tight some extra surfaces are created. This clean -up is done in ANSA meshing tool which is very robust clean up tool. Extracted domain for vortex generation and finder assemblies are shown below.

MESHING



After cleaning up the geometry surface mesh is generated in ANSA tool itself. All the surfaces are discretized using tri surface element .As the geometry has some complicated and skewed surfaces tri surface elements are

used to capture the geometry. The following figure shows the surface meshes.

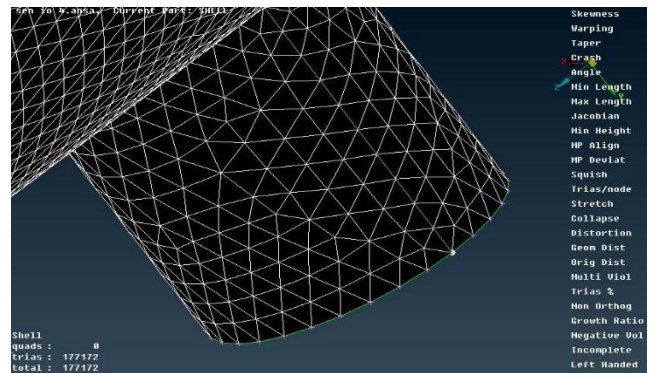
SURFACE MESH VOLUME MESHING

Volume mesh is generated in T-Grid which is a robust volume mesh generator. Volume is discretised using tetrahedron. Each and every cell centroids the co-ordinate at which the navies-stokes system of equations is solved.

MESH DETAILS:

Meshing Details

FLUID AND BOUNDARY



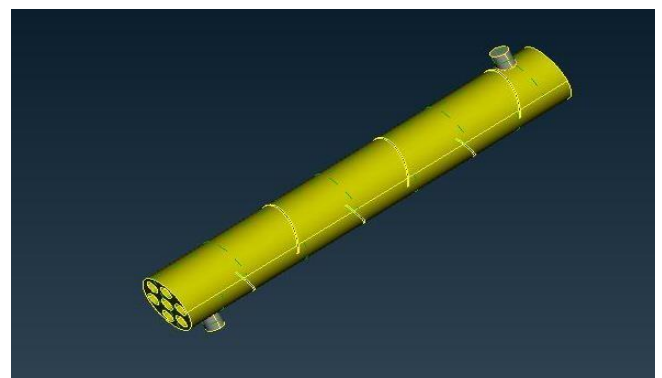
CONDITIONS

The following table shows the properties of steam, water and steel and copper considered for the analysis.

Fluid and Wall Boundary Conditions

$$\frac{\partial}{\partial t} (\alpha \rho_v) + \nabla \cdot (\alpha \rho_v \vec{V}_v) = m_{l \rightarrow v} - m_{v \rightarrow l}$$

SOLVER SETUP



ANSYS-FLUENT was used as the solver. The following assumptions and methodology was used to model this problem.

ASSUMPTIONS

- Flow was assumed to be three dimensional and turbulent.

METHODOLOGY
 INTRODUCTION

Turbulence was modelled by k-ε realizable model. Energy equation was used. Conductive and convective heat transfer modes were considered. Mixture multiphase model was applied. Mass transfer mechanism was considered to model Condensation. Segregated scheme was used with simple algorithms. First order discretization scheme

α

GOVERNING EQUATIONS OF FLUID DYNAMICS

The basic governing equations which describe the fluid dynamics, are used to solve the steam and water flow. The energy equation was used to define the conductive heat transfer across the fluid through the solid region.

Conservation of Mass

$$\frac{\partial \rho}{\partial t} + \text{div}(\rho \mathbf{u}) = 0$$

Conservation of X Momentum

$$\frac{\partial(\rho u)}{\partial t} + \text{div}(\rho u \mathbf{u}) = -\frac{\partial(p)}{\partial x} + \text{div}(\mu \text{grad } u) + \text{SM}_x$$

Conservation of Y Momentum

$$\frac{\partial(\rho v)}{\partial t} + \text{div}(\rho v \mathbf{u}) = -\frac{\partial(p)}{\partial y} + \text{div}(\mu \text{grad } v) + \text{SM}_y$$

Conservation of Z Momentum

$$\frac{\partial(\rho w)}{\partial t} + \text{div}(\rho w \mathbf{u}) = -\frac{\partial(p)}{\partial z} + \text{div}(\mu \text{grad } w) + \text{SM}_z$$

Conservation of Energy

Internal energy:

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

EVAPORATION-CONDENSATION MODEL

The evaporation-condensation model is a mechanistic model with a physical basis. It is available with the mixture and Euler-Ian multiphase models. The liquid-vapor mass transfer (evaporation and condensation) is governed by the vapor transport equation.

Where,

- v – Vapor phase,
- - Vapor volume fraction,
- v –vapor density, – vapor phase velocity
- are the rates of mass transfer due to evaporation and condensation, respectively. These rates use units of kg/s/m³.

As shown in the right side of Equation ANSYS FLUENT defines positive mass transfer as being from the liquid to the vapor for evaporation- condensation problems. Based on the following temperature regimes, the mass transfer can be described as follows,

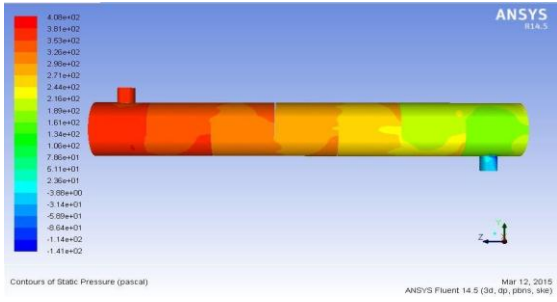
- If $T > T_{\text{sat}}$ Evaporation $m_{l \rightarrow v}$
 $= \text{coeff} * \alpha_l \rho_l (T - T_{\text{sat}}) / T_{\text{sat}}$
- If $T < T_{\text{sat}}$ Condensation $= \text{coeff} * \alpha_v \rho_v (T - T_{\text{sat}}) / T_{\text{sat}}$

‘Coeff*’ is a $m_{v \rightarrow l}$ coefficient(β) that needs to be fine-tuned and interpreted as a relaxation time α and ρ are the phase volume fraction and density respectively. The source term for the energy equation can be obtained by multiplying the rate of mass transfer by the latent heat. Consider the Hertz Knudsen formula, which gives the evaporation-condensation flux based on the kinetic theory for a flat interface.

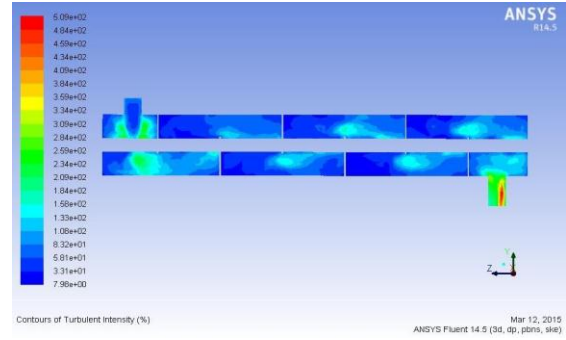
$$DP / DT = L/T (v_g - v_l)$$

The heat flux has units of kg/s/m³. P represents the partial pressure of the vapor at the interface on the gas side in KN/m² and T is the temperature in K. The coefficient β is the accommodation coefficient that shows the portion of vapor molecules going into the liquid surface and absorbed by this surface. Vg and vl are specific volume for the gas and liquid respectively. L is the latent heat in J/kg. Based on this differential expression, variations in temperature can be obtained from variations of pressure close to the saturation conditions.

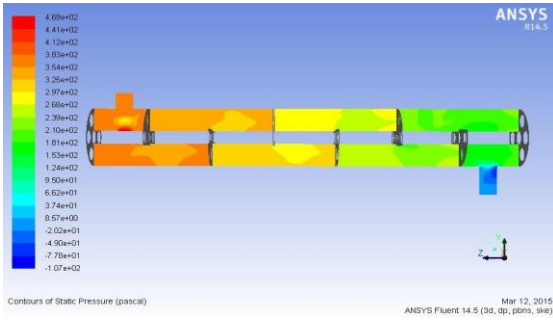
ANALYSIS
 BASE CASE-1



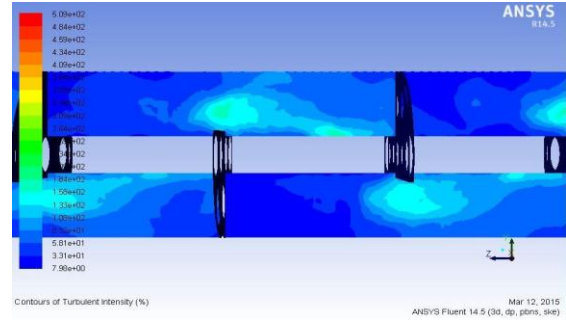
Static Pressure



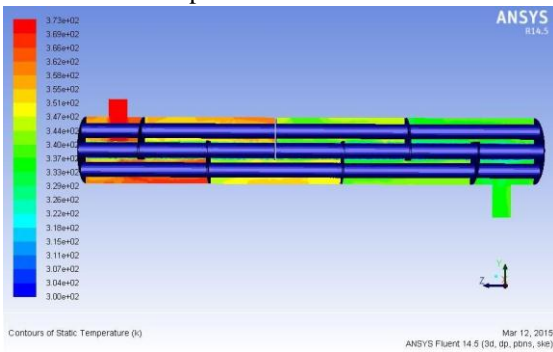
Turbulent Intensity



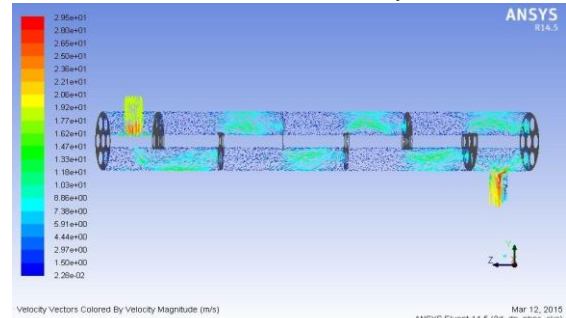
Cutplane View of Static Pressure



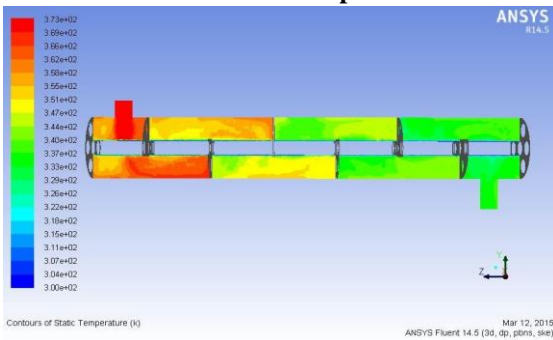
Zoom in View of Turbulent Intensity



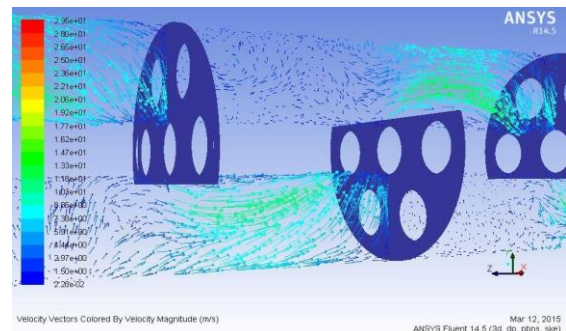
Static temperature



Cutplane View of Velocity Magnitude

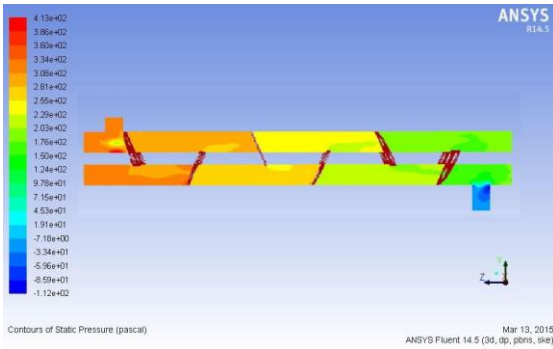


Cutplane View of Static Temperature

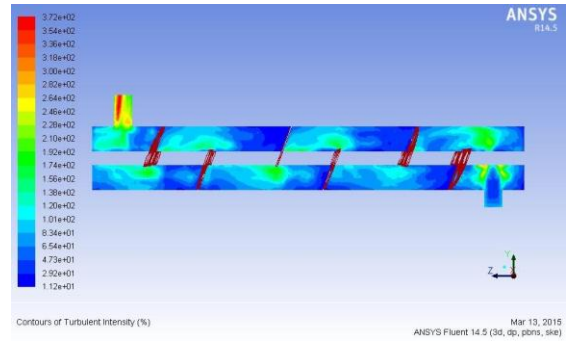


Zoomed View Velocity Vector

CASE-2

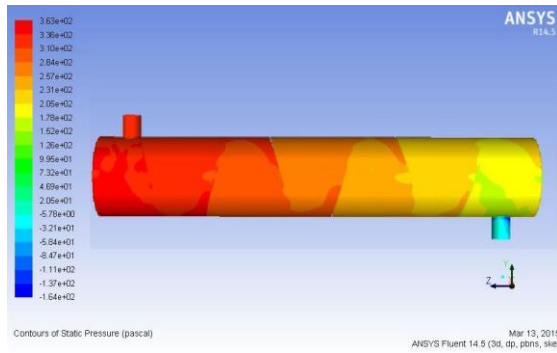
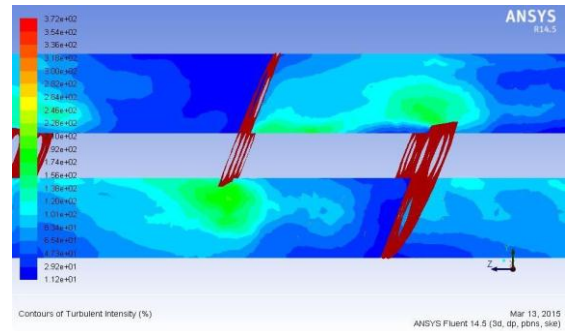


Cut Plane View of Static Pressure

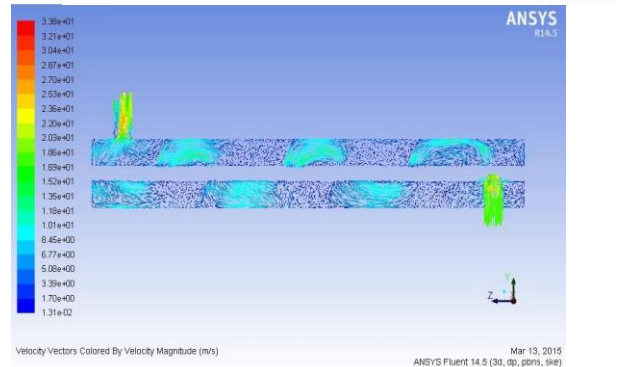


Cut Plane View of Turbulent Intensity

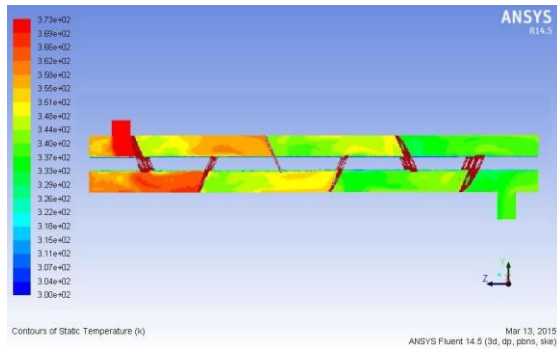
Zoom in View of Turbulent Intensity



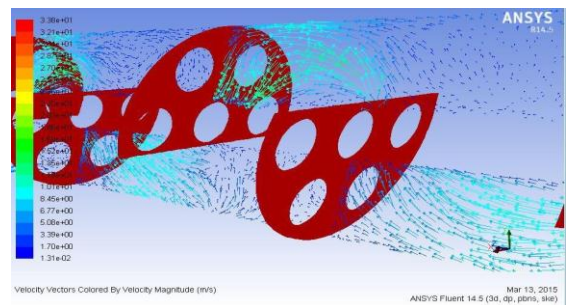
Static Pressure



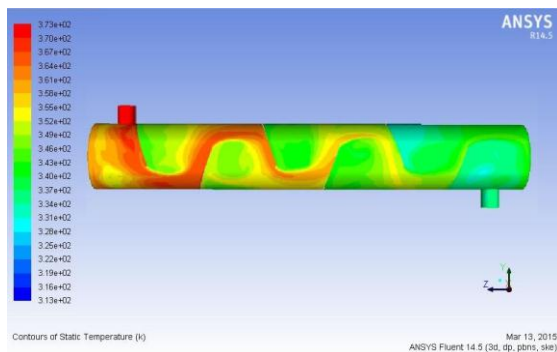
Velocity Vector



Cut Plane View of Static Temperature

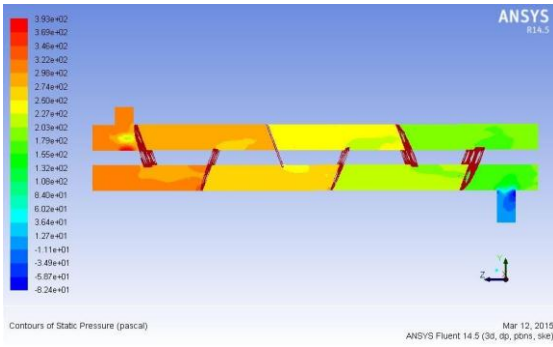


Zoom In View of Velocity Vector

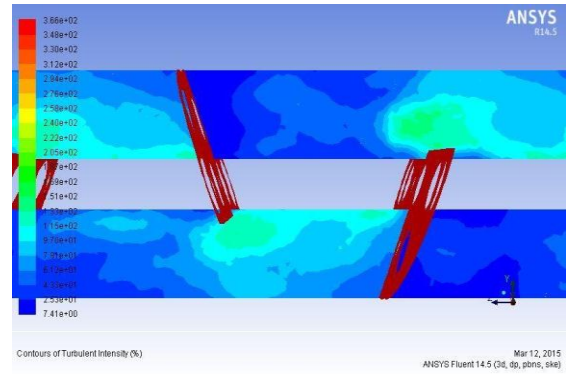


Static Temperature

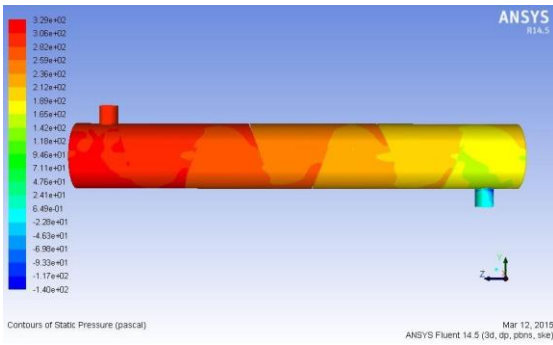
CASE 3



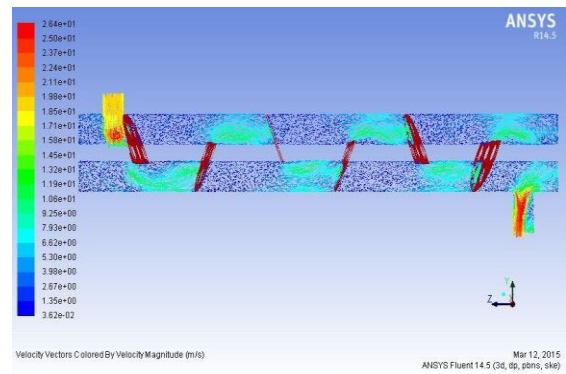
Cut Plane View of Static Pressure



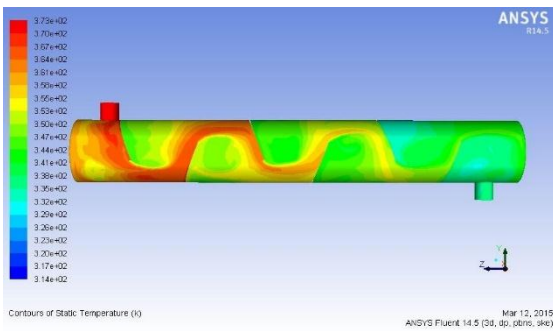
Cut Plane View of Turbulent Intensity



Static Pressure on Shell



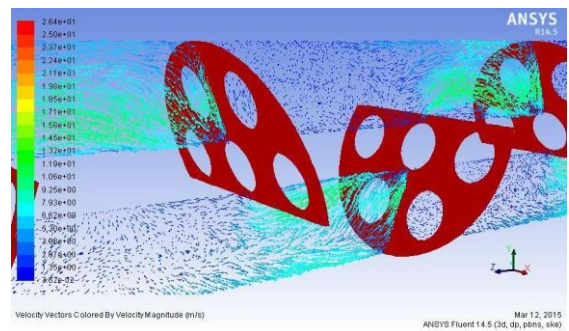
Velocity Vector



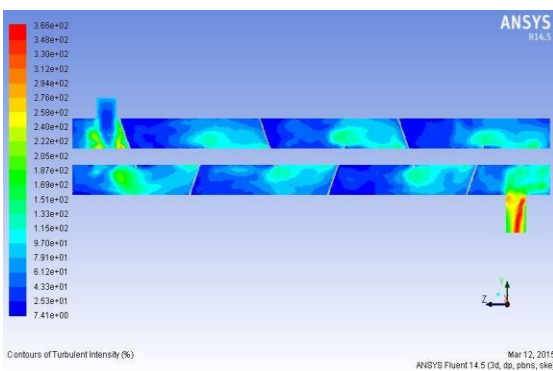
Static Temperature

CASE 4

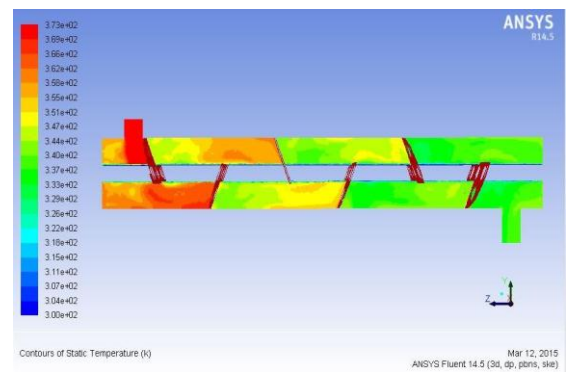
Cut Plane View of Velocity Vector Magnitude

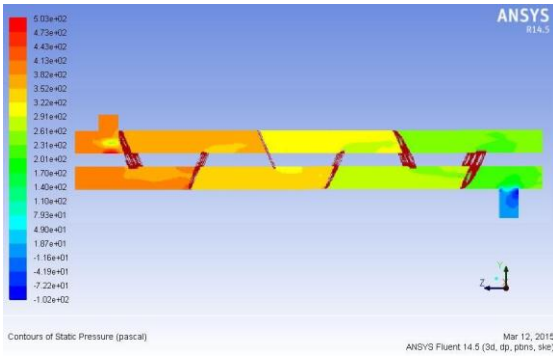


Cut Plane View of Static Temperature

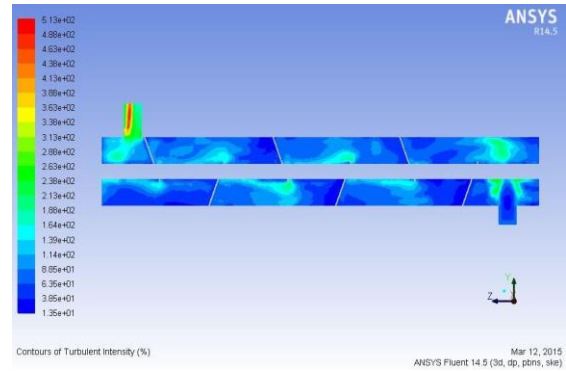


Turbulent Intensity

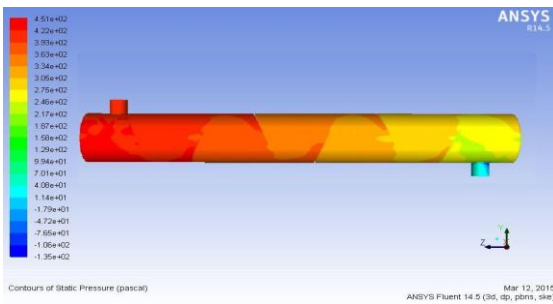




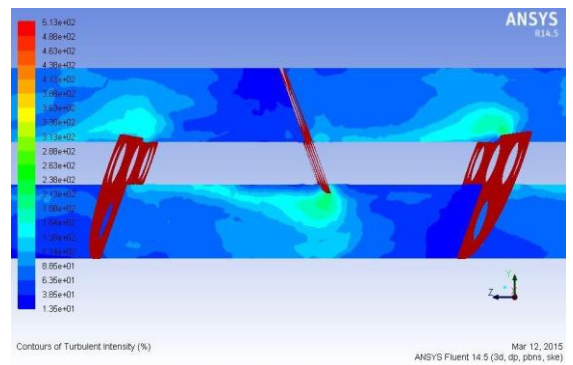
Cut View of Static Pressure



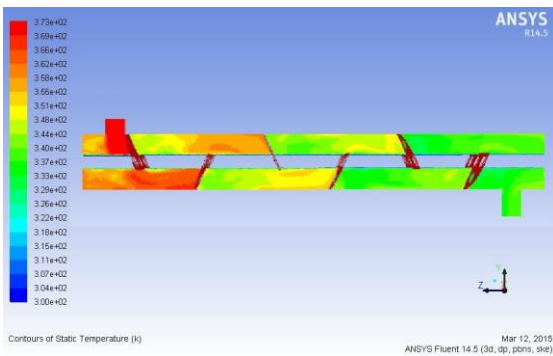
Turbulent Intensity



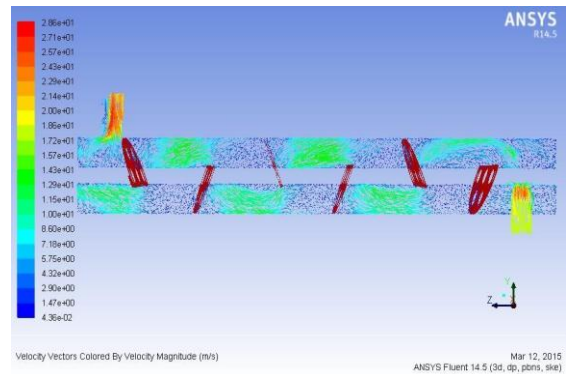
Static Pressure on Shell



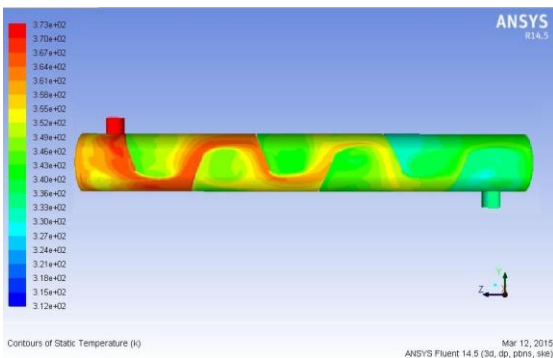
Zoom in View of Turbulent Intensity



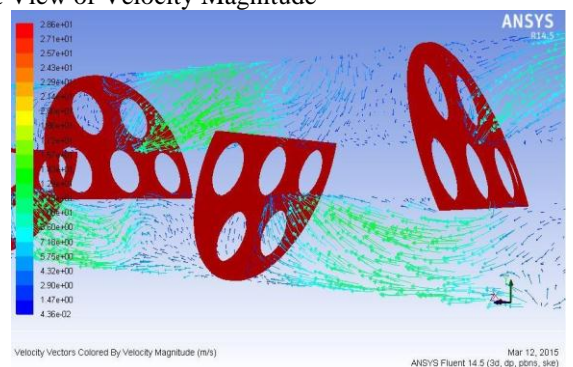
Cut View of Static Temperature



Cut View of Velocity Magnitude



Static Temperature on Shell



Zoom in View of Velocity Magnitude

In this project the above figure represents the static pressure, temperature and turbulent intensity and velocity vector of heat exchanger. It shows that pressure entries at high rate in inlet and decrease at outlet and same process are in

temperature also turbulent intensity show the same variation and velocity vector also decrease. All case have same process but we finally found out which case is better in maximum heat transfer with minimum pressure drop.

We used to analysis all case with pressure, temperature, turbulent intensity, velocity vector at which it used transfer the heat from one substance to another substance within a shell. In heat exchanger the baffles are used to withstand the tube at straight position and used enhance heat transfer rate and minimum pressure drop. We have mentioned the pressure and temperature different for all the case in result.

RESULT AND DISCUSSION

In this project we discuss about analysis between heat transfer rate temperature and pressure drop.

Temperature Measurement

Case 1 – 334.429

Case 2 – 333.691

Case 3 – 336.885

Case 4 – 332.140

Pressure Measurement

Case1 – 366.913

Case 2 – 320.383

Case 3 –296.438

Case 4 – 404.775

- Inlet Temperature=373 K

From the novel approach of CFD with the given data that agreed with the experimental method.

CONCLUSION

In this work a novel approach is attempted to conduct a parametric study to understand and identify the optimal level of about pointed out variables. Computational fluid dynamics which is found to be a better alternate for cumbersome, cost and time consuming experimental methodology is utilized in this study.

The shell side of a small shell-and-tube heat exchanger is modelled with sufficient detail to resolve the flow and temperature fields. From the CFD simulation results, for fixed tube wall and shell inlet temperatures, shell side heat transfer coefficient, pressure drop and heat transfer rate values are obtained. The sensitivity of the shell side flow and temperature distributions to the mesh density, the order of discretization and the turbulence modelling is observed. By varying baffle angles values of 25 degree and 30degree, for shell side flow rates, the simulation results are compared with the results from the Kern and Bell–Delaware methods.

Using CFD, together with supporting experiments, may speed up the shell-and-tube heat exchanger design process and may improve the quality of the final design. In the near future, improvements in the computer technology will make full CFD simulations of much larger shell-and-tube heat exchangers possible.

REFERENCES

- Gaddis D, editor. Standards of the Tubular Exchanger Manufacturers Association. Tarrytown (NY): TEMA Inc.; 2007.
- Kern DQ. Process heat transfer. New York (NY): McGraw-Hill; 1950.
- Bell KJ. Delaware method for shell side design. In: Kakaç S, Bergles AE, Mayinger F, editors. Heat exchangers: thermal–hydraulic fundamentals and design. New York: Hemisphere; 1981. p. 581–618.
- Gay B, Mackley NV, Jenkins JD. Shell-side heat transfer in baffled cylindrical shell-and-tube exchangers – an electrochemical mass transfer modeling technique. *Int J Heat Mass Transfer* 1976; 19:995–1002.
- Halle H, Chenoweth JM, Wabsganss MW. Shell side water flow pressure drop distribution measurements in an industrial-sized test heat exchanger. *J Heat Transfer* 1988; 110:60–7.
- Pekdemir T, Davies TW, Haseler LE, Diaper AD. Pressure drop measurements on the shell side of a cylindrical shell-and-tube heat exchanger. *Heat Transfer Eng* 1994; 15:42–56.