

CFD ANALYSIS OF SHELL TUBE HEAT EXCHANGER WITH AND WITHOUT BAFFLER

P.S. Aswin¹, A. Mohan²

¹U.G student, Dept. of Mechanical Engineering, MGR Educational and research Institute, Chennai, Tamilnadu, India

²Assistant Professor, Dept. of Mechanical Engineering, Master of Simulation Technology, Thuraiyur, Tamilnadu, India,

Abstract – In this paper the shell tube heat exchanger is analyzed with ANSYS, where the shell contains of tubes inside it and hot water flows through it and cold water flows through the outer path way of the tube on the shell. The heat transfer, velocity and stream line flow of the fluid is analyzed in this paper. The analysis is under taken for two various shell tube heat exchangers, the various shell tube heat exchangers are (a) Shell tube heat exchanger without baffler (b) Shell tube heat exchangers with baffler. Shell tube heat exchanger without baffler is the heat exchanger where a normal shell will be proceeded to the analysis and the result is extracted. Shell tube heat exchanger with baffler is the heat exchanger where a baffler will be presented on the shell where it provided a lot of time for the cold fluid to be in contact with hot tube so that the heat transfer will get more efficient.

Key Words: CFD, Ansys fluent, Shell tube heat exchanger, Baffler, Heat transfer

1. INTRODUCTION

1.1 General

A Heat exchanger is a system used to transfer heat between two or more fluids. Heat exchangers are used in both cooling and heating processes. They are widely used in space heating, refrigeration, air conditioning, power stations, chemical plants, petrochemical plants, petroleum refineries, natural-gas processing, and sewage treatment. Exchangers work is essential in various parts where they give the stability of working according to their need. In order of heat exchangers there are different types of heat exchanger where the flow exists the different types of heat exchangers are (a) counter flow heat exchanger (b) co-current flow heat exchanger (c) cross flow heat exchanger (d) hybrid flow heat exchanger.

Counter flow heat exchanger is the flow where three tubes exists on the system one of the tube in the middle consists of hot fluid and other two consists of cold fluid where flow of the cold fluid will be completely opposite to the flow of the hot fluid

Co-current flow heat exchanger is the flow where three tubes exists in the system one of the tube in the middle consists of hot fluid and others consists of cold fluid where flow of the cold fluid will be in the same path

Cross flow heat exchanger are intermediate in efficiency between countercurrent flow and parallel flow exchangers. In these units, the streams flow at right angles to each other

In industrial heat exchangers, hybrids of the above flow types are often found. Examples of these are combined cross flow/counter flow heat exchangers and multi pass flow heat exchangers

In a regenerative heat exchanger, the flow path normally consists of a matrix, which is heated when the hot fluid passes through it (this is known as the "hot blow"). This heat is then released to the cold fluid when this flows through the matrix (the "cold blow"). Regenerative Heat Exchangers are sometimes known as Capacitive Heat Exchangers. It consists of two types where (a) static (d) dynamic

Indirect heat exchanger in this type, the steams are separated by a wall, usually metal. Examples of these are tubular exchangers. Shell tube heat exchanger is the indirect heat exchanger type

1.2 Objective

Our main focus of our project is to get difference in the result of shell tube heat exchangers with and without bafflers using Ansys fluent.

2. MODELLING OF HEAT EXCHANGER

Software : AutoCAD

CAD modelling software is dedicated for the specialized job of 3D-modeling. The model of the steam boiler structures also includes many complicated parts, which are difficult to make by any of other CAD modelling as well as Finite Element software. CAD modelling of the complete steam boiler structure is performed by using AUTO CAD software. AUTO CAD is having special tools in generating surface design to construct typical surfaces, which are later converted into solid models.

Solid model of all parts of the structures are then assembled to make a complete structure. The process of assembly is very much analogous to general process of fabricating structures while real production. The CAD model of single and boiler used for FE Analysis during assembly is shown in figure. The assembled CAD model has been prepared from various part modelling drawings. After geometry creation these models were imported in ANSYS for thermal analysis.

Table 1: Steam Boiler Parameter

Sl. No.	Parameter		Forms
1	Shell	Inner diameter	188 mm
		Outer diameter	196 mm
2	Tube	Inner diameter	16 mm
		Outer diameter	20 mm
		Length	700 mm
3	Number of tubes		6

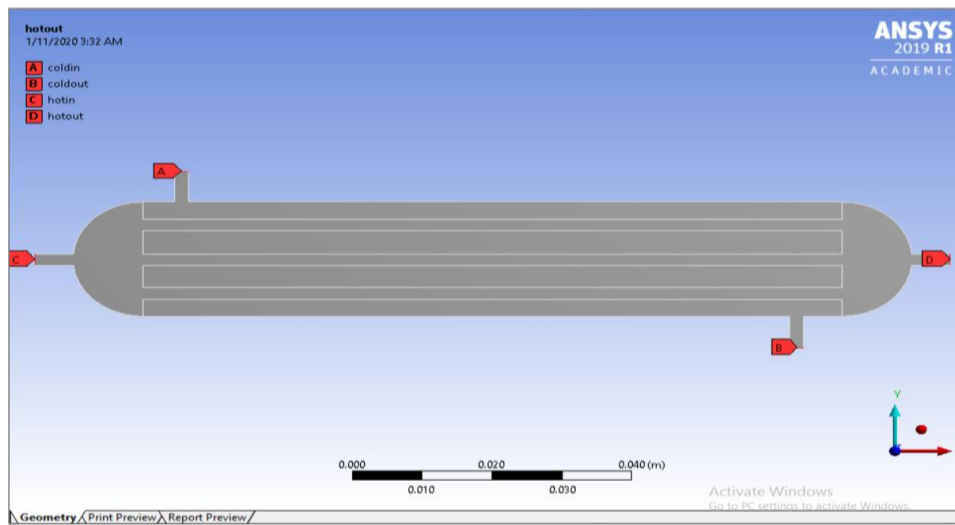


Fig -1: Named section View of heat exchanger without baffle

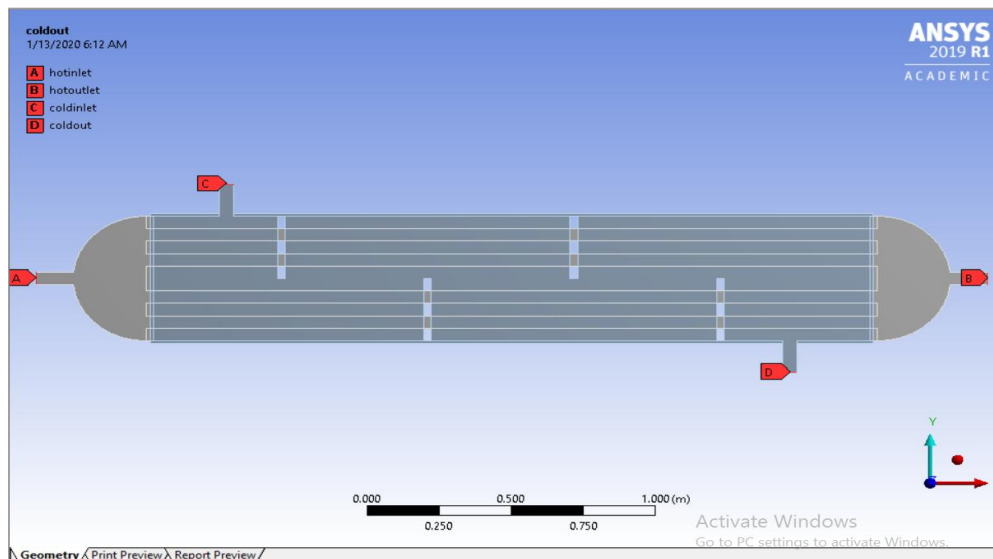


Fig -2: Named section View of heat exchanger with baffler

2.1 COMPUTATIONAL FLUID DYNAMICS ANALYSIS

Software: ANSYS Fluent

Computational fluid dynamics (CFD) study of the system starts with the construction of desired geometry and mesh for modelling the dominion. Generally, geometry is simplified for the CFD studies. Meshing is the discretization of the domain into small volumes where the equations are solved by the help of iterative methods. Modelling starts with the describing of the boundary and initial conditions for the dominion and leads to modelling of the entire system. Finally, it is followed by the analysis of the results, discussions and conclusions.

2.1.1 GEOMETRY

Geometry of the shell tube heat exchanger is created in the AutoCAD software and exported as IGES file previously, Now the IGES file is imported in the ANSYS software.

2.1.2 MESH

Initially a relatively coarser mesh is generated. This mesh contains mixed cells (Tetra and Hexahedral cells) having both triangular and quadrilateral faces at the boundaries. Care is taken to use structured hexahedral cells as much as possible. 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, a fine mesh is generated. For this fine mesh, the edges and regions of high temperature and pressure gradients are finely meshed.

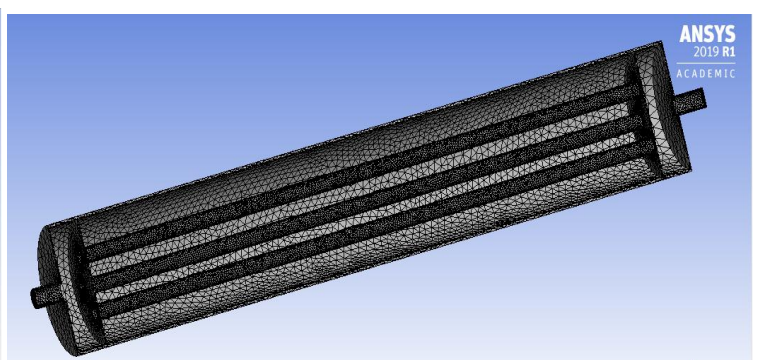
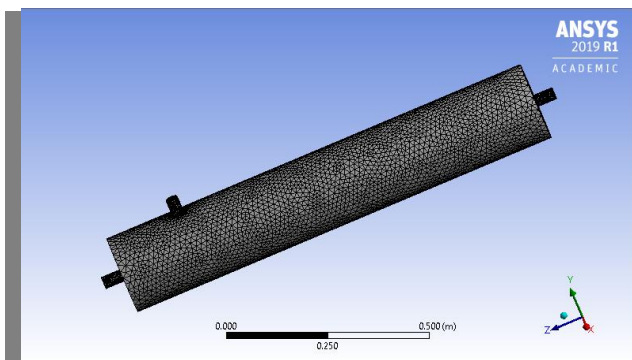


Fig -3: Mesh view of heat exchanger **Fig -4:** Sectional Mesh view of heat exchanger

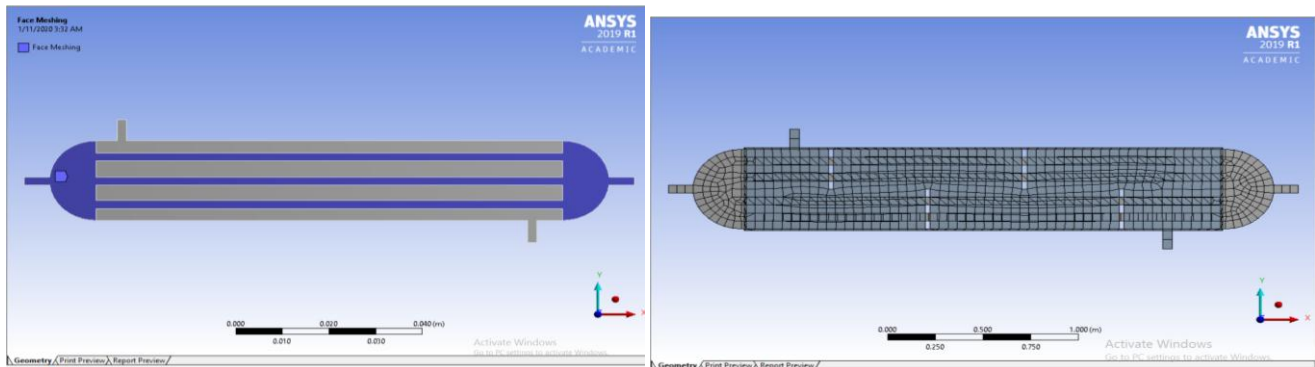


Fig -5: Sectional view of face mesh model **Fig -6:** Sectional view of mesh with baffle

3. SOLUTION

3.1 PROBLEM SETUP

The setup is initialized. The mesh is checked and the quality is obtained. The analysis type is changed to Pressure Based type. The velocity formulation is changed to absolute and time to steady state. Gravity is defined as $y = -9.81 \text{ m/s}^2$

3.2 MODELS

Energy is set to ON position. Viscous model is selected as “k-ε model (2 equations). Radiation model is changed to Discrete Ordinates.

3.3 MATERIAL

The create/edit option is clicked to add water-liquid and copper, steel to the list of fluid and solid respectively from the fluent database.

3.4 CELL ZONE

The parts are assigned as water and copper, steel as per fluid/solid parts.

3.5 BOUNDARY CONDITION

Boundary conditions are used according to the need of the model. The inlet and outlet conditions are defined as mass flow rate inlet and pressure outlet. As this is parallel -flow with two tubes so there are two inlets and two outlets. The walls are separately specified with respective boundary conditions. No slip condition is considered for each wall.

Table 2: Inputs parameters

Quantities	Boundary condition
Working fluid	Water, Steam
Water	Mass flow rate = 0.025 kg/s Temperature = 300 k
Steam	Mass flow rate = 0.125 kg/s Temperature = 900 k

3.6 RUN CALCULATION

The number of iteration is set and the solution is calculated and various contours, vectors and plots are obtained.

4. RESULT

4.1 Contours of Steam Boiler

The temperature, pressure and velocity distribution along the heat exchanger can be seen through the contours.

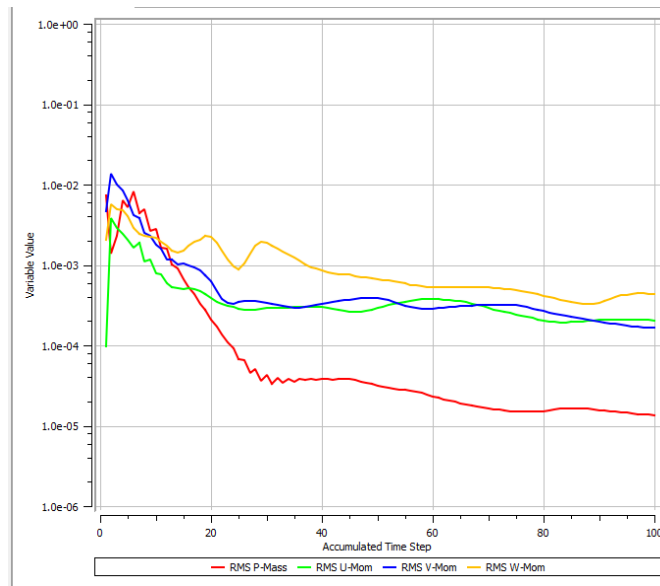


Chart -1: Iterations of heat exchanger

4.2 PRESSURE DISTRIBUTION

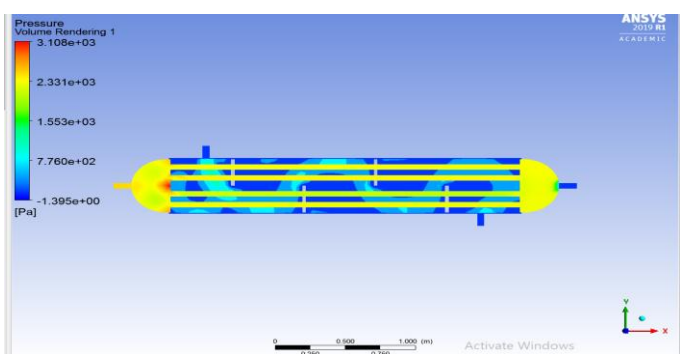
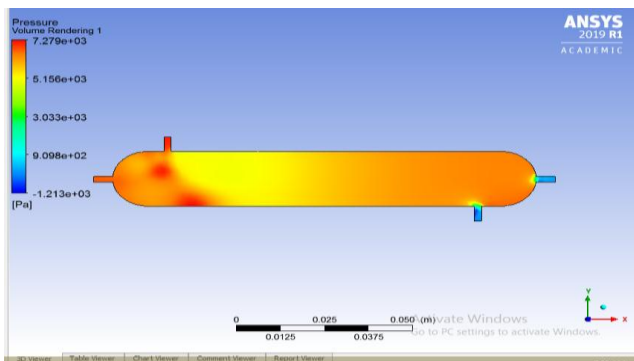


Fig -7: Pressure Distribution without baffle Fig -8: Pressure Distribution with baffle

4.3 TEMPERATURE DISTRIBUTION

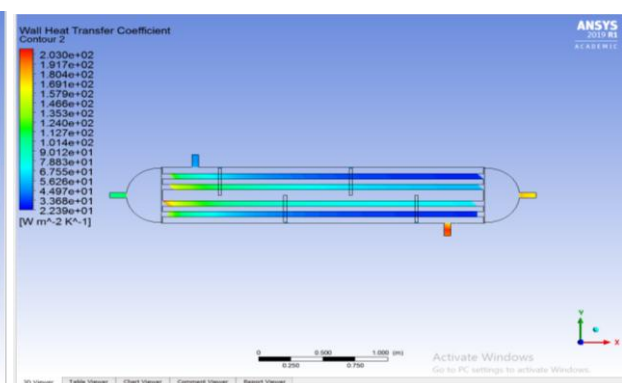
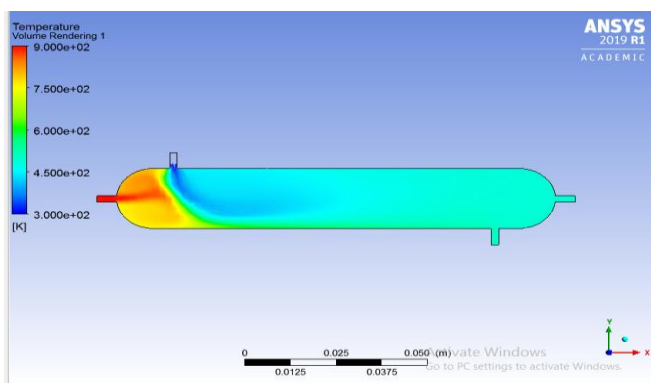


Fig -9: Without baffle Fig -10: With baffle

4.4 VELOCITY DISTRIBUTION

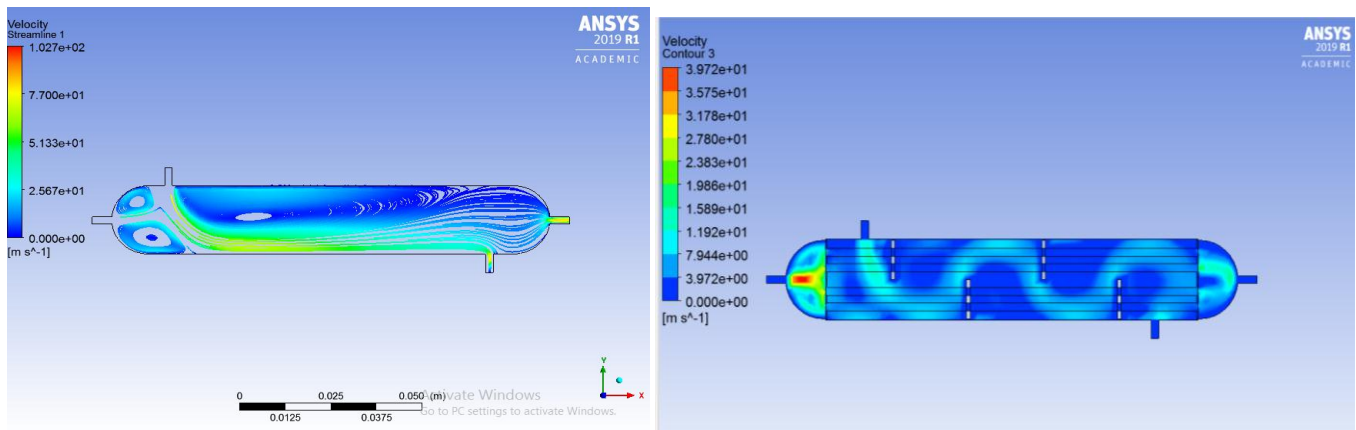


Fig -11: Velocity Distribution without baffle Fig -12: Velocity Distribution with baffle

5. CONCLUSION

In this project the steam flow in steam boiler tubes is modeled using AUTO CAD design software. The thesis will focus on CFD analysis of steam boiler without baffles. CFD analysis to determine the temperature, velocity, pressure drop. From the above analysis the temperature is increases from 300k to 900k. CFD analysis has been done on both the models such that heat exchanger with baffles and heat exchanger without baffles. On comparing the both result, heat exchanger with baffles performance better than heat exchanger without baffles.

REFERENCES

- [1] Design and Thermal Analysis of Steam Boiler used in Power Plants V. Ashok Kumar, G. Kumar, K.Pavan Kumar IARJSET ISSN (Online) 2393-8021 ISSN (Print) 2394-Vol. 3, Issue 11, November 2016
- [2] International Journal of Emerging Technology and Advanced Engineering (ISSN 2250-2459, ISO 9001:2008 Certified Journal, Volume 2, Issue 12, December 2012) 301 Evaluation of Heat Losses in Fire Tube Boiler
- [3] S.Krishnanunni¹, Josephkunju Paul C, Mathu Potti, Ernest Markose Mathew Referred Finite Element Method and Applications in Engineering Using Ansys second edition by Springer.
- [4] Referred High pressure boilers fifth edition by Frederick M Steingress, Harold j Frost, Daryl R Walker.

BIOGRAPHIES



P.S.Aswin, U.G student, Dept. of Mechanical Engineering, MGR Educational and research Institute, chennai, Tamilnadu, India



A.Mohan, Assistant Professor, Dept. of Mechanical Engineering, Master of Simulation Technology, Thuraiyur, Tamilnadu, India,