ANALYSIS OF EFFECT OF BAFFLE INCLINATION ANGLE ON SHELL AND TUBE HEAT EXCHANGER USING COMPUTATIONAL FLUID DYNAMICS

P.Abbas Kiyasudeen
Assistant Professor, Department of Mechanical Engineering, Mohamed Sathak Engineering College, Kilakarai, Ramanathapuram District, Tamilnadu, India

T.Dharmaprabhakaran
P.G Student, Department of Mechanical Engineering, Mohamed Sathak Engineering College, Kilakarai, Ramanathapuram District, Tamilnadu, India

Abstract

In the shell side design of a shell and tube heat exchanger, the baffles inclination depends on heat transfer coefficient and the pressure drop. Those are investigated by numerically modeling a smallheat exchanger. The flow and temperature fields inside the shell are resolved using a commercial CFD software tool FLUENT. In this present work, attempts were made to investigate the impacts of various baffle inclination angles on fluid flow and the heat transfer characteristics of a shell and tube heat exchanger for different baffle inclination angles namely -20°, 0° and +20°. The simulation results for various shell and tube heat exchangers, one with segmental baffles perpendicular to fluid flow and two with segmental baffles inclined to the direction of fluid flow are compared for their performance. The results are observed to be sensitive to the turbulence model selection. For a given baffle cut of 36%, the heat exchanger performance is investigated by varying baffle inclination angle. Finally from the CFD simulation results, the best design is concluded.
Keywords: *Shell and tube heat exchanger, CFD, Heat transfer &Pressure drop, Baffle inclination angle*

1. Introduction

1. 1 Computational Fluid Dynamics

Computational Fluid Dynamics abbreviated as CFD is the analysis of systems involving fluid flow, heat transfer and associated phenomena such as chemical reactions by means of computer based simulation. The technique is very powerful to perform the millions of calculations required to simulate the interaction of fluids and gases with complex surfaces used in engineering. The calculations required simulating the interaction of fluids with surfaces defined by boundary conditions, and initial conditions are done by the commercial CFD packages. CFD is useful for studying fluid flow, heat transfer, chemical reactions, etc. by solving mathematical equations with the help of numerical analysis. CFD resolve the entire system in small cells and apply governing equations on these discrete elements to find numerical solutions regarding pressure distribution, temperature gradients. CFD is a sophisticated computationally-based design and analysis technique. CFD Software gives you the power to simulate flows of gases and liquids, heat and mass transfer, moving bodies, multiphase physics, chemical reaction, fluid-structure interaction and acoustics through computer modeling. This software can also build a virtual prototype of the system or device before can be apply to real-world physics and chemistry to the model, and the software will provide with images and data, which predict the performance of that design. CFD is useful in a wide variety of applications and use in industry. CFD is one of the branches of fluid mechanics that uses numerical methods and algorithm can be used to solve and analyze problems that involve fluid flows and also simulate the flow over a piping, vehicle or machinery. More recently the methods have been applied to the design of internal combustion engine, combustion chambers of gas turbine and furnaces, also fluid flows and heat transfer in heat exchanger. The development in the CFD field provides a capability comparable to other Computer Aided Engineering (CAE) tools such as stress analysis codes.

1.1.1 Basic Approach to CFD

The basic approach steps in CFD are discussed below and the diagram is shown in figure 1.1.

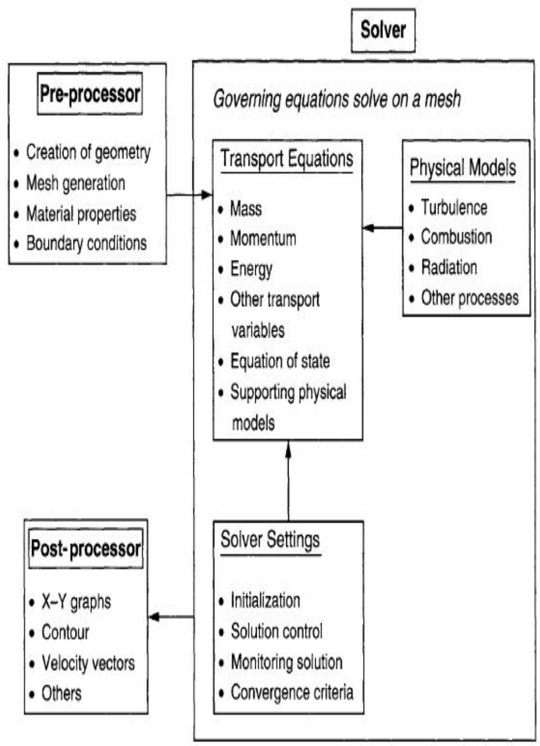

Figure 1.1 Basic Approach in CFD

a) Pre-processor: Establishing the model

- Identify the process or equipment to be evaluated.
- Represent the geometry of interest using CAD tools.
- Use the CAD representation to create a volume flow domain around the equipment containing the critical flow phenomena.
- Create a computational mesh in the flow domain.

b) Solver:

- Identify and apply conditions at the domain boundary.
- Solve the governing equations on the computational mesh using analysis software.

c) Post processor: Interpreting the results

- Post-process the completed solutions to highlight findings.
- Interpret the prediction to determine design iterations or possible solutions, if needed.

1.1.2 Applications of CFD

CFD not just spans on chemical industry, but a wide range of industrial and nonindustrial application areas which is in below:

- Aerodynamics of aircraft
- Combustion in IC engines
- Gas turbine in power plant
- Aerodynamics of vehicle
- Loads on offshore structure in marine engineering
- Blood flows through arteries and vein in biomedical engineering
- Weather prediction in meteorology
- Flow inside rotating passages and diffusers in turbo-machinery
- External and internal environment of buildings like wind loading and heating or Ventilation system
- Mixing and separation or polymer moldings in chemical process engineering
- Distribution of pollutants and effluent in environmental engineering

1.2. Heat Exchangers

Heat exchangers are one of the mostly used equipment in the process industries. Heat Exchangers are used to transfer heat between two process streams. One can realize their usage that any process which involve cooling, heating, condensation, boiling or evaporation will require a heat exchanger for these purpose. Process fluids, usually are heated or cooled before the process or undergo a phase change.

Different heat exchangers are named according to their application. For example, heat exchangers being used to condense is known as condensers, similarly heat exchanger for boiling purposes are called boilers. Performance and efficiency of heat exchangers are measured through the amount of heat transfer using least area of heat transfer and pressure drop. A better presentation of its efficiency is done by calculating over all heat transfer coefficient. Pressure drop and area required for a certain amount of heat transfer, provides an insight about the capital cost and power requirements (Running cost) of a heat exchanger. Usually, there is lots of literature and theories to design a heat exchanger according to the requirements.

The purpose of constructing a heat exchanger is to get an efficient method of heat transfer from one fluid to another, by direct contact or by indirect contact. The heat transfer occurs by three principles: conduction, convection and radiation.

In a heat exchanger the heat transfer through radiation is not taken into account as it is negligible in comparison to conduction and convection. Conduction takes place when the heat from the high temperature fluid flows through the surrounding solid wall. The conductive heat transfer can be maximized by selecting a minimum thickness of wall of a highly conductive material. But convection is plays the major role in the performance of a heat exchanger.

Forced convection in a heat exchanger transfers the heat from one moving stream to another stream through the wall of the pipe. The cooler fluid removes heat from the hotter fluid as it flows along or across it.

In commercial aircraft heat exchangers are used to take heat from the engine's oil system to heat cold fuel. This improves fuel efficiency, as well as reduces the possibility of water entrapped in the fuel freezing in components.

One common example of a heat exchanger is the radiator in a car, in which the hot radiator fluid is cooled by the flow of air over the radiator surface.

1.2.1 Types of Heat Exchangers

There are many different types of Heat Exchangers, designed for different purposes as in figure1.2.

Heat exchangers are of two types:-

- Where both media between which heat is exchanged are in direct contact with each other is direct contact heat exchanger.
- Where both media are separated by a wall through which heat is transferred so that they never mix, indirect contact heat exchanger.

2. Methodology

2.1 Description of Problem

The flow chart of the process is shown in the figure 4.1. The normal shell and tube heat exchangers have a moderate rate of heat transfer. In order to enhance the heat transfer rate the baffles are introduced in the shell part. The baffles can be placed in various angles to vary the heat transfer rate. But the rate of heat transfer will be high in only certain inclination. The normal HE doesn’t have any obstruction in the fluid path. Thus the heat transfer rate is considerable low. In order to overcome this issue we redesign the HE by introducing the baffles.

2.2 Design of the shell and tube Heat Exchanger

There are several designs in shell and tube heat exchanger. Even though, the basic principle is still the same. The tubes may be straight or bent in the shape of a U, called U-tubes. This U-tubes type typically use in nuclear power plants. The heat exchanger is used to boil water recycled from a surface condenser into steam to drive a turbine to produce power.

2.2.1 Specifications of the design

The design specifications and parameters of shell and tube heat exchanger are listed in the table 4.1.

Table 2.1 Geometric Dimensions of Shell and Tube Heat Exchanger

Parameters	Dimensions
Length of the shell	700 mm
Diameter of the shell	150 mm
Length of the tube	500 mm
Diameter of the tube	12 mm
Number of tubes	16
Number of baffles	3
Profile of the tube	Straight
Baffle inclination angle	-20°,0° and 20°
Inlet/outlet diameter	50.8mm

2.2.2 Basic Parts

2.2.2.1 Baffle Structure

Baffles serve two functions: They support the tubes in the proper position during assembly and operation and prevent vibration of the tubes caused by flow-induced eddies, and secondly, they guide the shell-side flow back and forth across the tube field, increasing the velocity and the heat transfer coefficient. The tubes are placed inside the baffle through the holes as in figure. The baffle cut involved in our design is about 30% and the baffle is 70% of the inner diameter of the shell as in figure 2.1

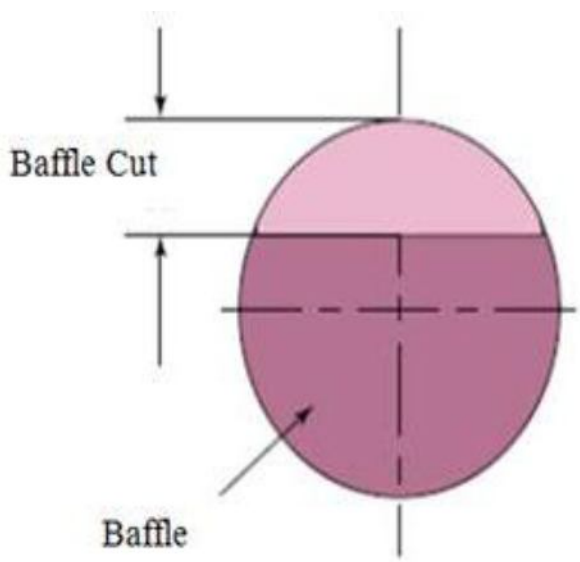

Figure 2.1 Baffle cut Indication Diagram

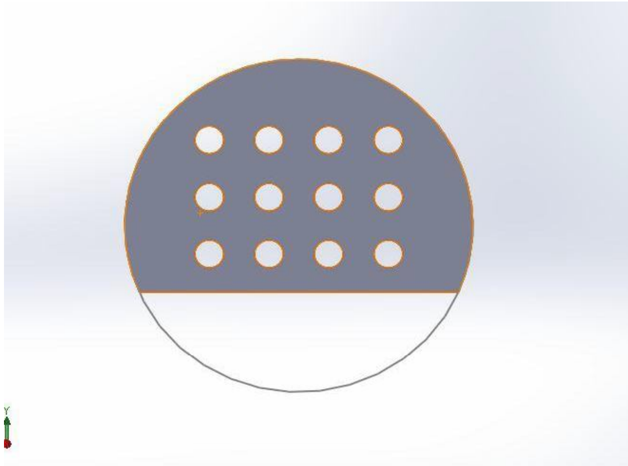

Figure 2.2 Baffle with Holes

2.2.2.2 Shell

The shell is simply the container for the shell-side fluid, and the nozzles, or the inlet and exit ports. The shell normally has a circular cross section and is commonly made by rolling a metal plate of the appropriate dimensions into a cylinder and welding the longitudinal joint (rolled shells). Small diameter shells can be made by cutting pipe of the desired diameter to the correct length (pipe shells). The roundness of the shell is important in fixing the maximum diameter of the baffles that can be inserted and therefore the effect of shell-to-baffle leakage. Pipe shells are more nearly round than rolled shells unless particular care is taken in rolling, In order to minimize out-of-roundness, small shells are occasionally expanded over a mandrel; in extreme cases, the shell is cast and then bored out on a boring mill. In large exchangers, the shell is made out of low carbon steel wherever possible for reasons of economy, though other alloys can be and are used when corrosion or high temperature strength demands must be met. The geometrical dimensions of the existing heat exchanger are mentioned in the figure and the cross sectional view of the shell is shown in figure.

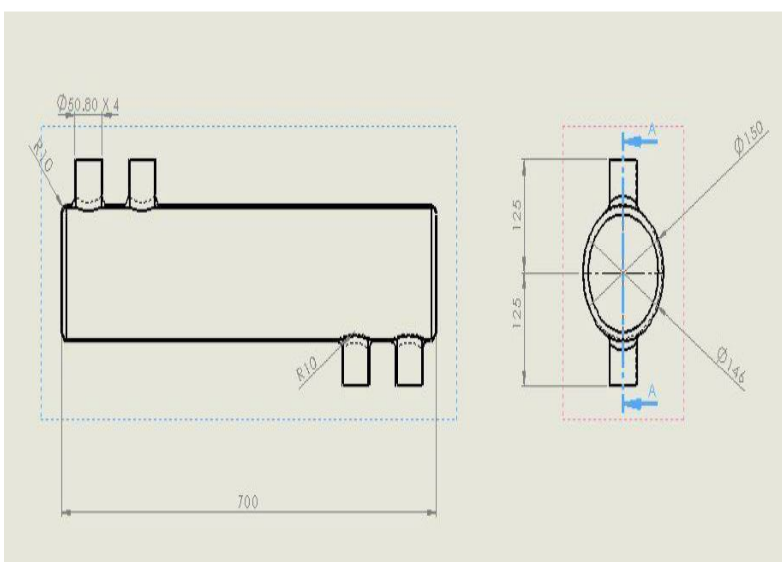

Figure 2.3 Geometric Dimensions of STHE

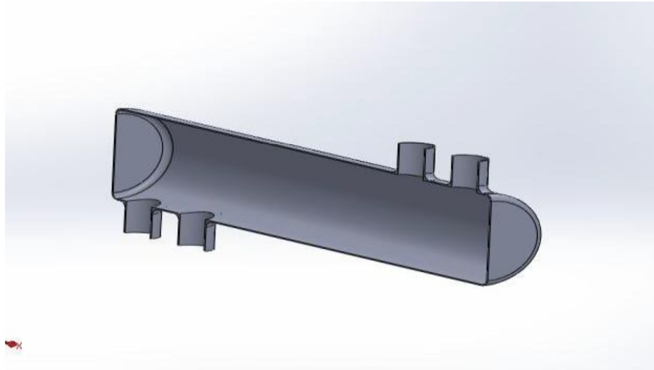

Figure 2.4Design Cross Sectional View of Shell

2.2.2.3 Tube

The tubes are the basic component of the shell and tube heat exchanger, providing the heat transfer surface between one fluid flowing inside the tube and the other fluid flowing across the outside of the tube. The tubes may be seamless or welded and most commonly made of copper or steel alloys. Other alloys of nickel, titanium, or aluminum may also be required for specific applications. The model of a tube design is shown in the figure.

Figure 2.5 Design view of Tube

2.2.2.4 Assembled View Inside the Shell (0 Deg Inclination)

After the completion of the design of assembled view, it needs to assemble inside the shell. It is clearly shown in the figure.

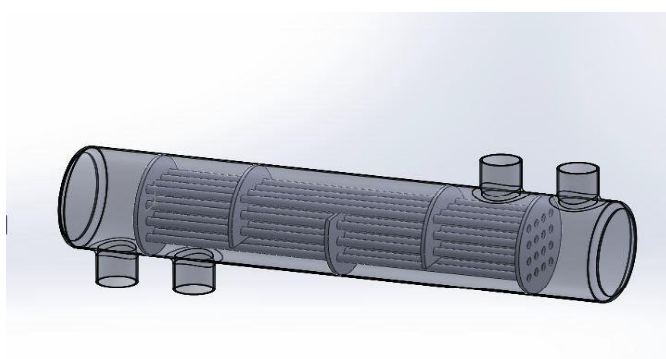

Figure 2.6 Assembled View Inside the Shell (0 Deg)

2.2.2.5 Heat Exchanger with +20° Baffle Inclination

In this project, it is supposed to change the baffle angles to increase the heat transfer rate. First the positive angle to be designed as in figure 4.8.

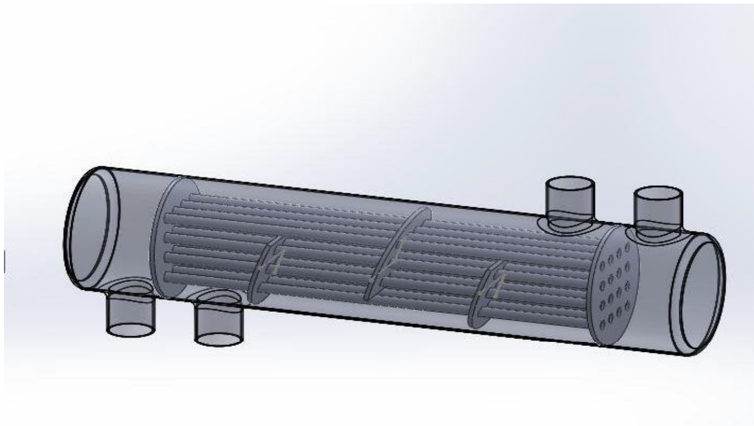

Figure 2.7 Heat Exchanger with +20° Baffle Inclination

2.2.2.6 Heat Exchanger with -20° Baffle Inclination

Similarly it is need to be checking the heat transfer rate and the other parameters in the negative baffle inclination angle as in figure.

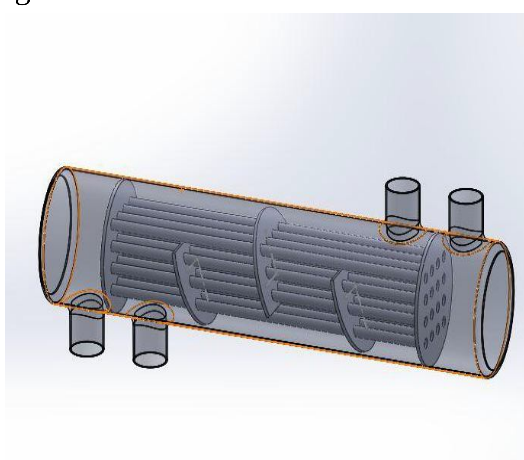

Figure 2.8 Heat Exchanger with -20° Baffle Inclination

2.3 Final Model of Shell and Tube Heat Exchanger

The components required for building the shell and tube heat exchanger have been selected according to the specification and assembled with proper instructions. The final model is shown in the above figure.

2.4 Surface Meshing

The surface meshing is done with the help of ANSA software. Before analyzing a model it should be discredited in to number of small triangular elements then it will be meshed. The surface mesh models for the different baffle inclination are shown in the figures.

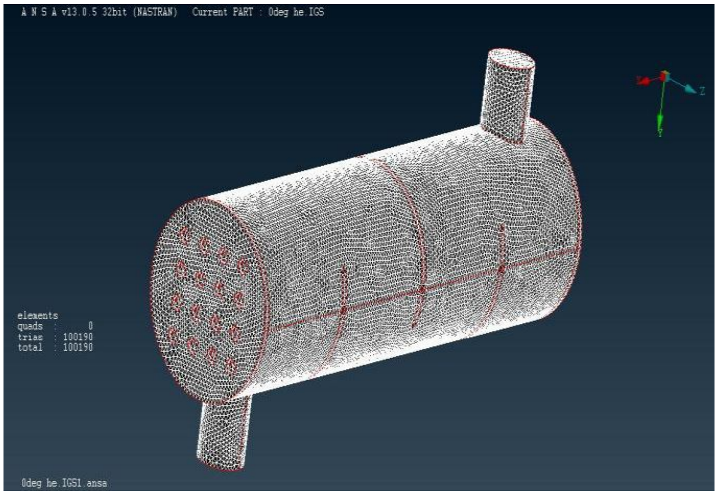

Figure 2.9 Surface Meshing with 0 Degree Inclination

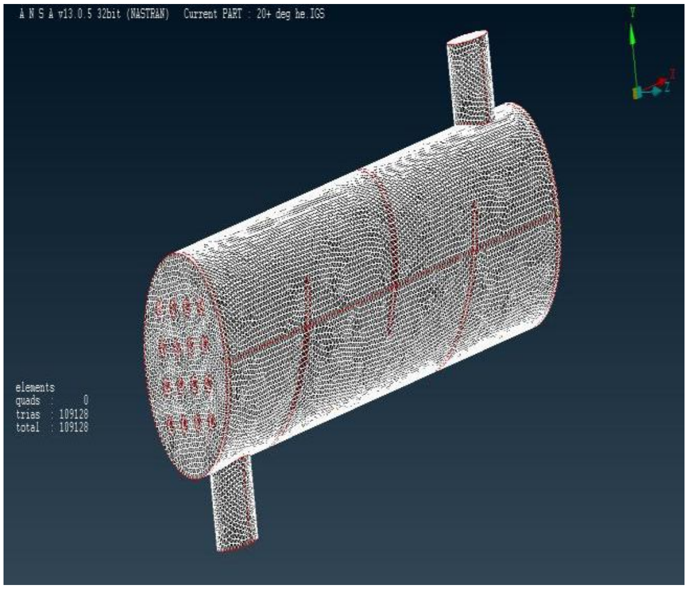

Figure 2.10 Surface Meshing with +20 Degree Inclination

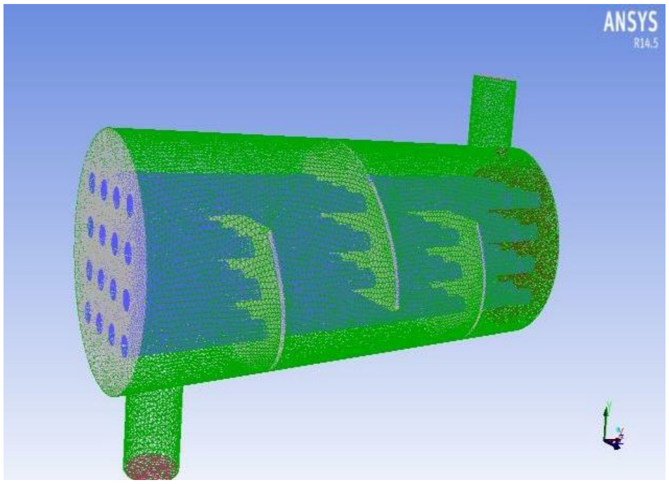

Figure 2.11 Surface Meshing with -20 Degree Inclination

Result and Discussion

3.1 Analysis Results

The model is analyzed with three different baffle inclination angles by using ANSYS FLUENT software. The final results are shown in the figures 3.1, 3.2 & 3.3. The results are compared and conclude the best design. The Temperature and Pressure values are compared and tabulated in the table 3.1.

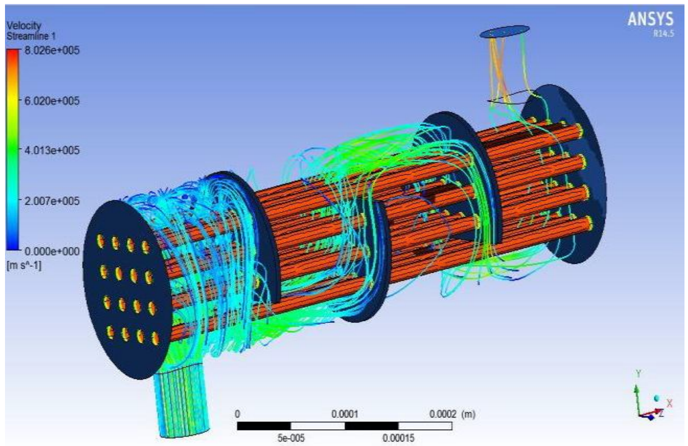

Figure 3.1 Variation of Streamlines along the Shell for 0° Baffle Inclination Angle

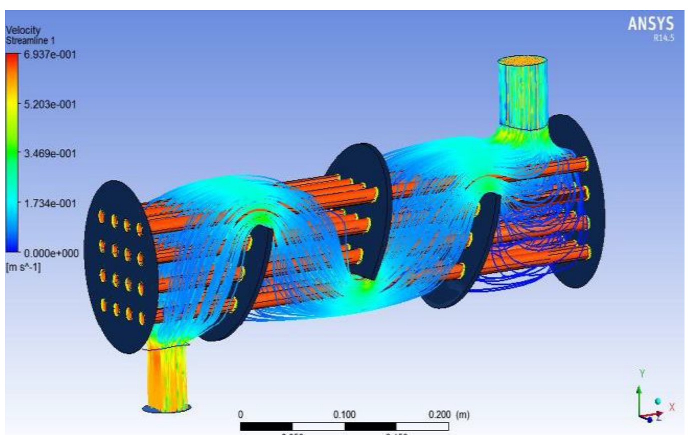

Figure 3.2 Variation of Streamlines along the Shell for +20° Baffle Inclination Angle

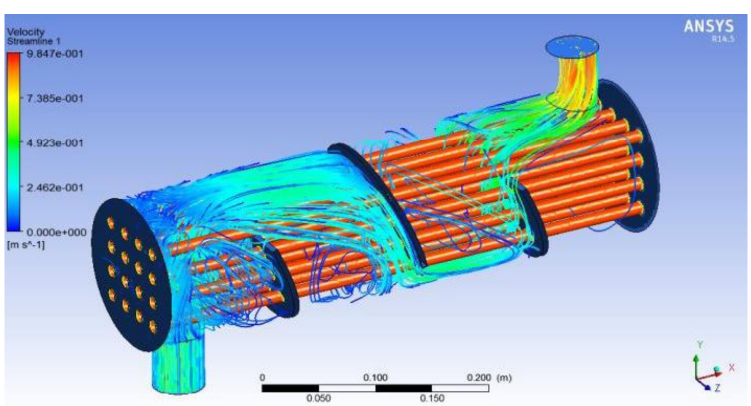

Figure 3.3 Variation of streamlines along the shell for -20° Baffle inclination angle

Baffle Inclination Angle (degrees)	Total Temperature (K)		Total Pressure (Pascal)	
	Min.	Max.	Min.	Max.
-20°	267	347	-168	897
0°	300	338	-193	790
+20°	260	377	-147	544

Conclusion

From this project work, the computational model of a baffled shell and tube heat exchanger is designed using the 3D modeling software, Solid works. Modeled heat exchanger is analyzed by ANSYS FLUENT tool. The dimensions are derived from an existing heat exchanger. And the baffles are employed in three different degrees to find the best design.

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

- The Heat transfer rate is increased when we increase the baffle inclination angle (from 0 to 20°)
- The shell-side pressure is decreased with increase in baffle inclination angle
- The maximum baffle inclination angle can be 20°.If the angle is beyond/above 20°, the center row of tubes are not supported. Hence the baffle cannot be used effectively
- Hence it can be concluded that STHE with +20° angle results is better performance compared to -20° and 0° baffle inclination angles.

References

1. Emerson, WH., 1963, ‘Shell-side pressure drop and heat transfer with turbulent flow in segmental baffled shell-tube heat exchangers’, International Journal of Heat and Mass Transfer, Vol.6, pp.649-668.
2. Ender Ozden, 2010, ‘Shell side CFD analysis of a small shell-and-tube heat exchanger’, Energy Conversion and Management, Vol.51, pp.1004 -1014.
3. Gaddis, D, editor, 2007, ‘Standards of the Tubular Exchanger Manufacturers Association’, ninth ed,.Tarrytown (NY): TEMA Inc,Vol.12, pp.124-184.
4. Gay, B,Mackley & NV., Jenkins, JD., 1976, ‘Shell-side heat transfer in baffled cylindrical shell and tube exchangers’, International Journal of Heat and Mass Transfer,Vol.19, pp.995-1002.
5. Halle H & Chenoweth JM, 1988, ‘Shell side water flow pressure drop distribution measurements in an industrial-sized test heat exchanger’, Journal of Heat Transfer, Vol.110, pp.60-67.
6. Kakac, S., &Liu, H., 2002, ‘Heat Exchangers Selection, Rating and Thermal Design’, CRC press, second ed, Washington D.C., Vol.24, pp.318-335.

7. Karno.A& Ajib.S, 2006, 'Effects of baffle cut and number of baffles on pressure drop and heat transfer in shell-and-tube heat exchangers – numerical simulation', International Journal of Heat Exchangers, Vol.7, pp.299-322.
8. Li, H.,& Kottke, V.,1998, 'Effect of baffle spacing on pressure drop and local heat transfer in shell and- tube heat exchangers for staggered tube arrangement', International Journal of Heat and Mass Transfer, Vol.41, pp.1303-1311.
9. Mukherjee, R., 2004, 'Practical Thermal Design of Shell-and-Tube Heat Exchangers', BegellHouse.Inc, New York, Vol.36, pp.528-564.
10. Sparrows, EM.& Reifschneider, LG., 1986, 'Effect of inter baffle spacing on heat transfer and pressure drop in a shell-and-tube heat exchanger', International Journal of Heat and Mass Transfer, Vol.29, pp.1617-1628.
11. Thirumarimurugan, M., Kannadasan, T, & Ramasamy, E, 2008, 'Performance Analysis of Shell and Tube Heat Exchanger Using Miscible System', American Journal of Applied Sciences, Vol.5, pp.548-552.
12. UdayKapale, C.,& Satish Chand, 2006, 'Modeling for shell-side pressure drop for liquid flow in shell-and-tube heat exchanger', International Journal of Heat and Mass Transfer, Vol.49, pp.601-610.
13. Versteeg, H.K., & Malalasekera, W., 1995, 'An introduction to computational fluid dynamics: the finite volume method', first ed, Essex (England): Pearson. Vol.67, pp.412-456.