CFD ANALYSIS OF SHELL AND TUBE HEAT EXCHANGERS WITH DIFFERENT BAFFLES

Shubham Kumar Jain[1], Rohit Soni[2]

M.Tech Scholar[1], Professor[2]

Department of Mechanical Engineering, Trinity Institute of Technology and Research, Bhopal.

ABSTRACT

Shell and tube heat exchangers are the most common type of heat exchangers used in present scenario. Heat exchangers are widely used equipment in various industries such as power generation and transportation, refrigeration industry and chemical process industries because it suits high pressure application. Presented in this project is comparison for several shell- and- tube heat exchangers with segmental baffles. The objective of this project is to design a shell and tube heat exchanger with segmental baffles and to study the flow and temperatures inside the shell and tubes using ANSYS software tool for the different baffles assemblies and orientation also overall heat transfer is calculated for each design.

Keywords: Shell and tube heat exchanger, Solid works, Ansys CFD Fluent, Segmental baffles, outlet pressure difference.

1. INTRODUCTION

A ‘heat exchanger’ may be defined as an equipment which transfers the energy from a hot fluid to a cold fluid, with maximum rate and minimum investment and running costs. In heat exchanger the temperature of each fluid changes as it passes through the exchangers, and hence the temperature of the dividing wall between the fluids also changes along the length of the exchanger. One of the important processes in engineering is the heat exchanger between flowing fluids, and many types of heat exchangers are employed in various types of installations, as petro-chemical plants, process industries, pressurised water reactor power plants, nuclear power stations and refrigeration systems. On the basis of design tubular or shell and tube type of heat exchangers are widely in use. The shell and tube heat exchangers are the one in which one of the fluids flows through a bundle of tubes enclosed by a shell. The other fluid is forced through the shell and it flows over the outside surface of the tubes. Such an arrangement is employed where *reliability* and *heat transfer effectiveness* are important. With the use of multiple tubes heat transfer rate is amply improved due to increased surface area. These heat exchanger’s larger heat transfer surface area-to-volume ratios than the most of common types of heat exchangers, and they are manufactured easily for a large variety of sizes and flow configurations. They can operate at high pressures, and their construction facilities disassembly for periodic maintenance and cleaning.

2. METHODOLOGY

In this work, performance for shell and tube heat exchanger is developed using Ansys CFD. Simulations are conducted to develop a model using the parameters such as temperatures and flow rates.

This work will be complete with following steps:-

1. First, we will prepare shell and tube heat exchanger model of different baffles arrangement in CATIA V5 and save as this part in IGES format and after that in ANSYS Workbench 14.5 Environment.
2. Then we will apply material for shell and tube heat exchanger model.
3. After that we will mesh the model.
4. Define type of analysis: fluent analysis
5. Define boundary condition for analysis boundary conditions play an important role in finite element calculation.
6. Run the analysis using design of experiment.
7. Get the results
8. Validate our results with various ANN models.
9. Compare all the results obtained

2.1 GEOMETRY MODELING

First the geometry of the model is created in CATIA V5R21. The model is saved in IGS. format. The external geometry file is imported in the design modeller of the ANSYS fluent. The geometry has totally 22 parts. One shell and 21 tubes bundle.

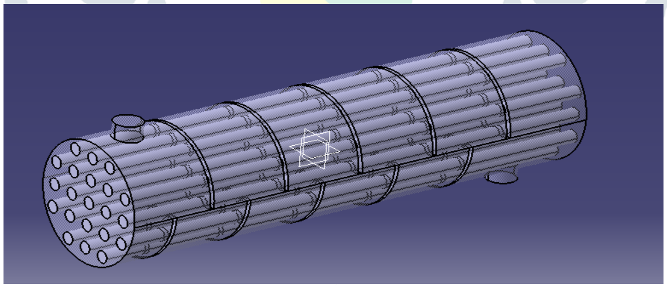

Fig. 1: Geometry of the model having single segmental baffle

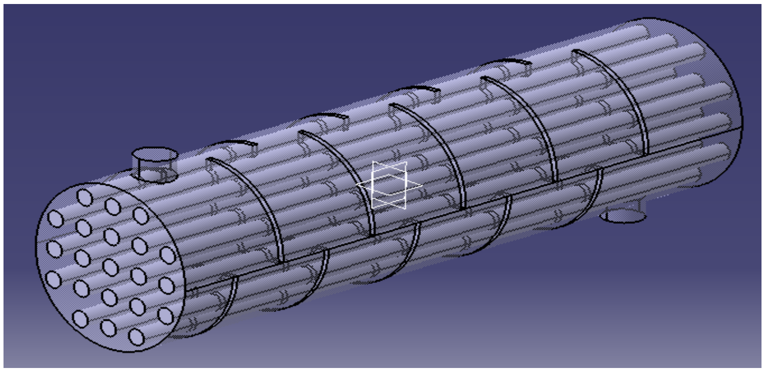

Fig. 2: Geometry of the model having flower ‘A’ baffle

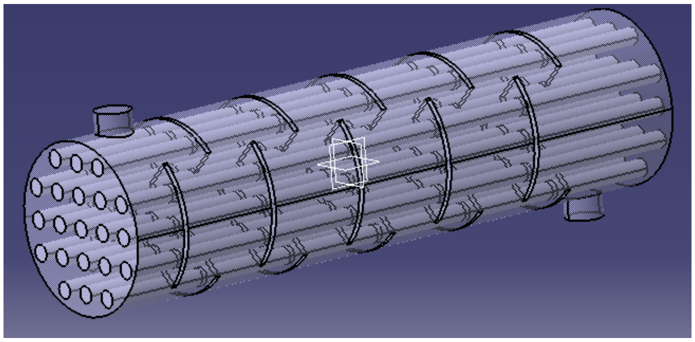

Fig. 3: Geometry of the model having flower 'B' baffle

2.2 MESHING

In free meshing a comparatively coarse mesh is generated. It contains each tetrahedral and hexahedral cell having triangular and quadrilateral faces at the boundaries. Later, a fine mesh is generated using edge filler. In this, the perimeters and regions of high and temperature gradients area unit finely meshed.

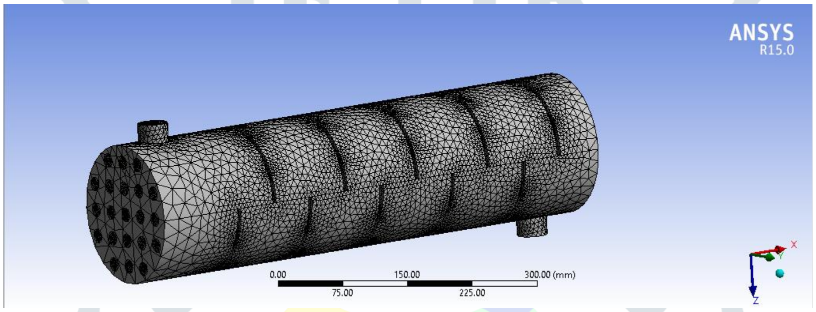

Fig. 4: Mesh model having single segmental baffle

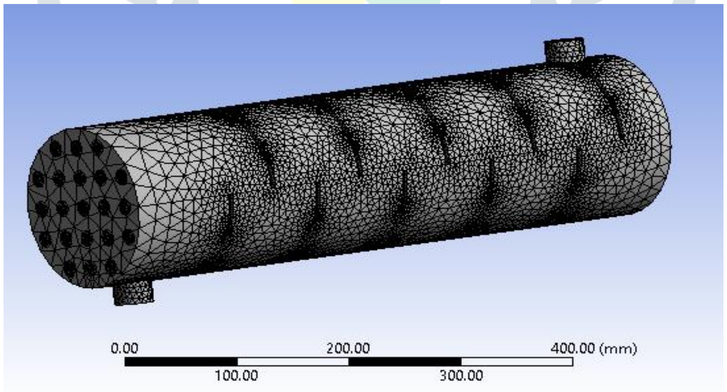

Fig. 5: Mesh model having flower 'A' baffle

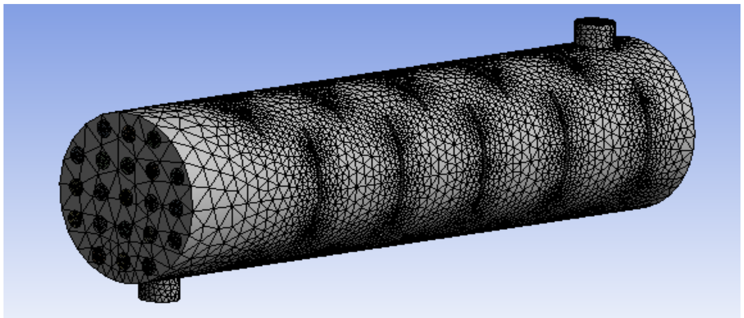

Fig. 6: Mesh model having flower 'B' baffle

2.3 BOUNDARY CONDITIONS

Different boundary conditions were applied for various zones. Since it's a shell-and-tube device, there are 2 inlets and 2 retailers. The inlets were outlined as rate inlets and retailers were outlined as pressure retailers. The water recess boundary conditions are set as Flow gap inlets and outlet boundary conditions are set as Pressure gap outlets. The outside wall is modelled as adiabatic. The simulation is solved to predict the heat transfer and fluid flow characteristics by exploitation k-ε turbulence model.

Shell aspect recess is about as flow gap the mass rate of flow varied from 0.7533 kg/s for various simulations and temperature is about to 303 K.

Tube aspect recess is about to flow opening the mass rate of flow is varied from 0.7 kg/s to 1.2 kg/s and also the temperature is about to 363 K.

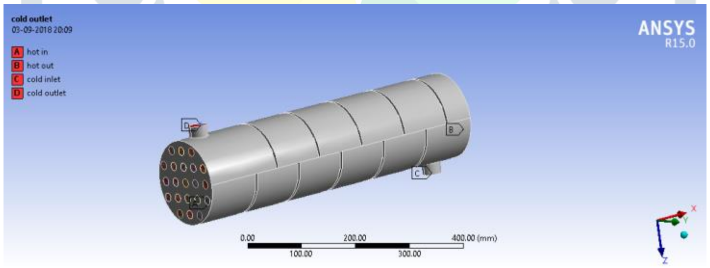

Fig. 7: Boundary condition in the model having single segmental baffle

3. RESULT

3.1 PRESSURE VARIATIONS WITH SINGLE SEGMENTAL BAFFLE CONFIGURATIONS IN STHX

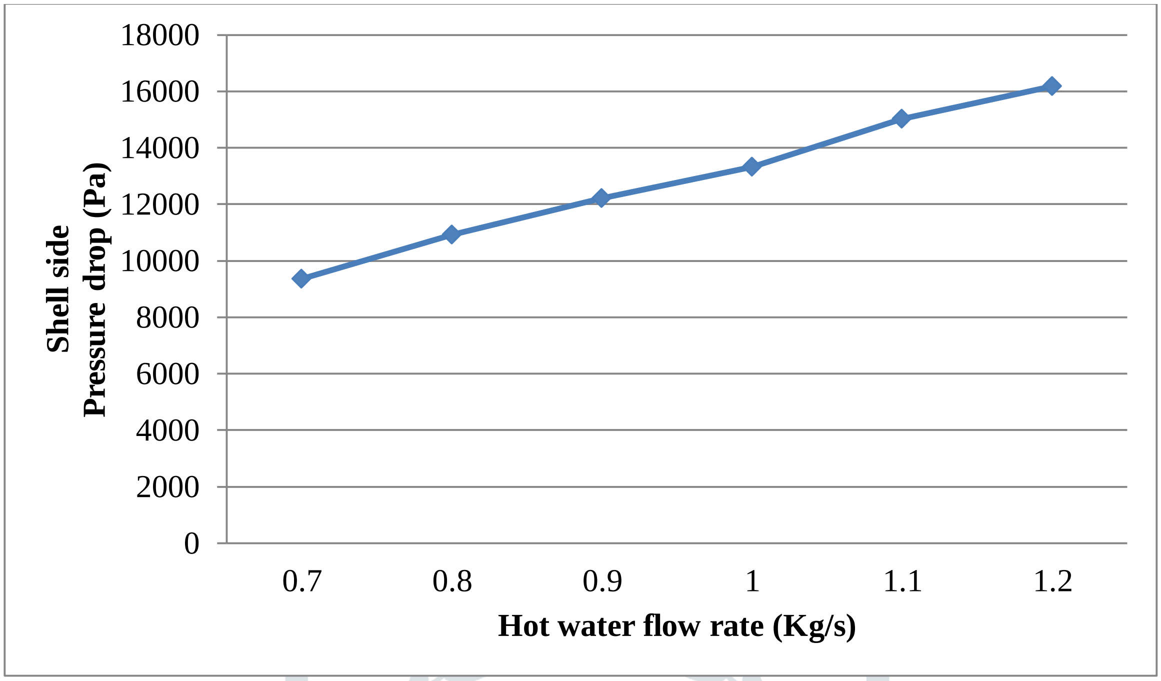

Fig. 8: Variation of shell side pressure drop in STHX with single segmental baffles

3.2 VARIATIONS IN HEAT TRANSFER COEFFICIENT FOR SINGLE SEGMENTAL BAFFLE IN STHX

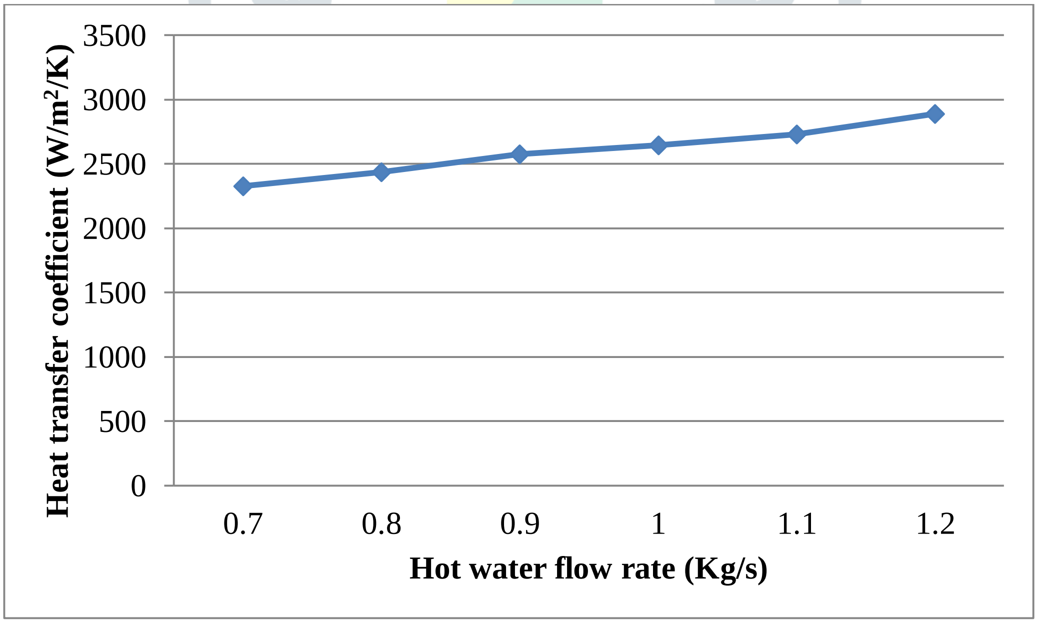

Fig. 9: Variation of heat transfer coefficient in STHX with single segmental baffle

3.3 PRESSURE VARIATIONS WITH FLOWER 'A' BAFFLE CONFIGURATIONS IN STHX

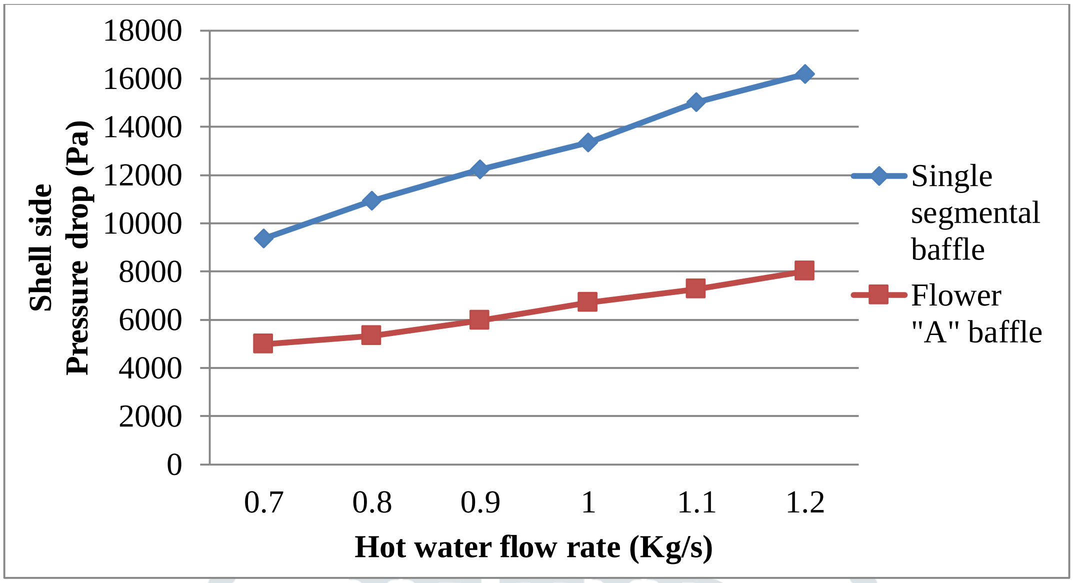

Fig. 10: Variation of pressure drop with parameters in STHX with Flower 'A' baffle

3.4 VARIATIONS IN HEAT TRANSFER COEFFICIENT FOR FLOWER 'A' BAFFLE IN STHX

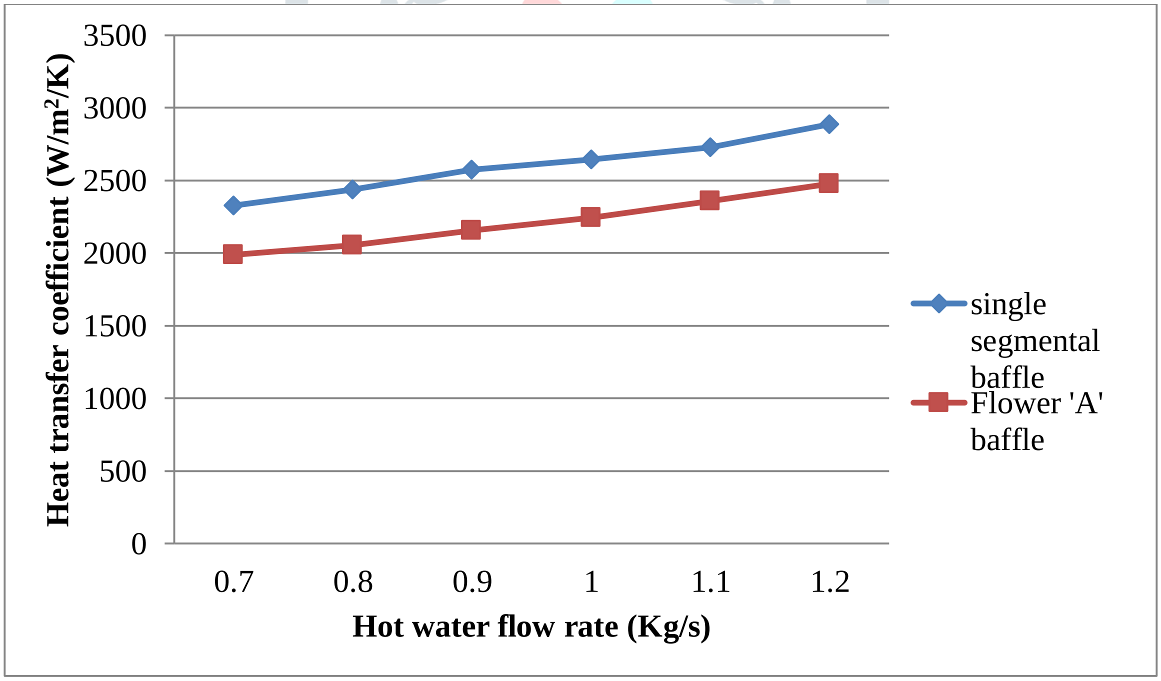

Fig. 11: Variation of heat transfer coefficient in STHX with flower 'A' baffle

3.5 PRESSURE VARIATIONS WITH FLOWER 'B' BAFFLE CONFIGURATIONS IN STHX

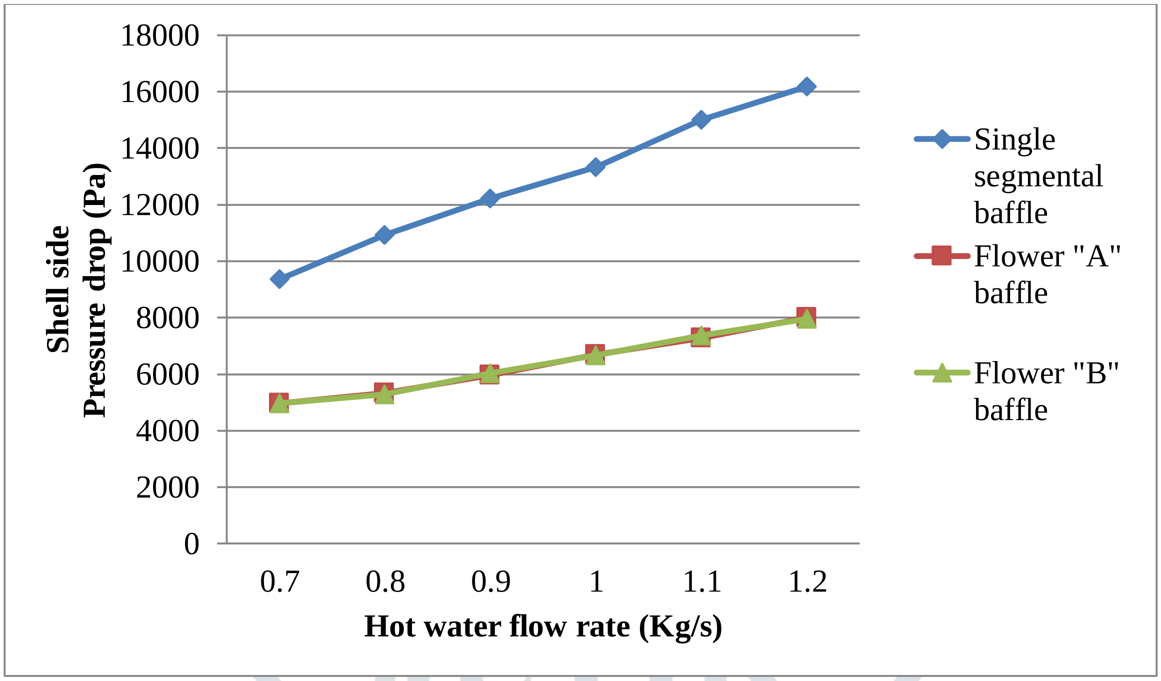

Fig. 12: Variation of shell side pressure drop in STHX with flower 'B' baffles

3.6 VARIATIONS IN HEAT TRANSFER COEFFICIENT FOR FLOWER 'B' BAFFLE IN STHX

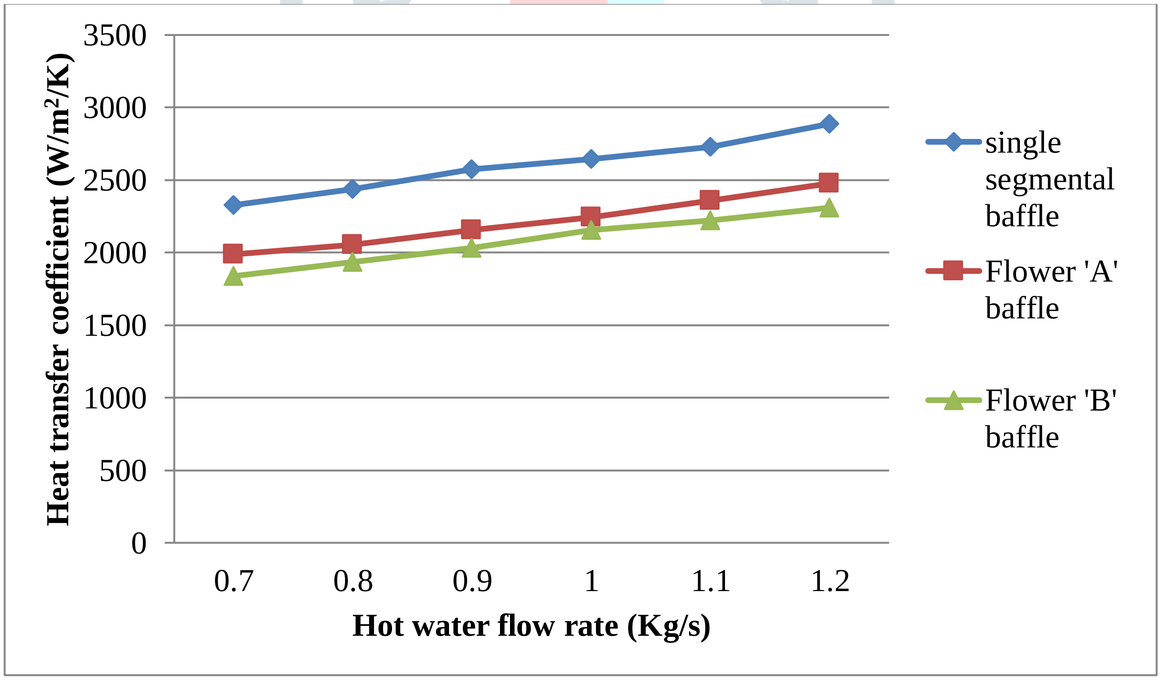

Fig. 13: Variation of heat transfer coefficient in STHX with flower 'A' baffle

4. CONCLUSION

1. From simulation, it's inferred that the flow pattern created is comparable to it created by flower 'A' baffle, except that a lot of stream lines are discovered in flower 'A' baffles in comparison with flower 'B' baffles and thus lesser is that the stagnation zone in flower 'A' in comparison with flower 'B' baffle.

Whereas in flower 'A' fluid rate magnitude on the shell facet changes sporadically within the central a part of the flower 'A' baffled device. Once the fluid passes a baffle, it's first of all accelerated speedily so flows across the breaches with giant rate. When speeding out of the breaches, the fluid is expanded suddenly and also the rate is reduced gradually. This periodic flow pattern is caused by the periodic changes of flow space that is induced by arrangement of flower baffles. Moreover, it's additionally detected that within the downstream simply behind a baffle, 2 recirculation flow regions are generated, wherever the speed magnitude is very tiny.

2. Single segmental baffles shows most pressure drop as a result of pin like turns the fluid takes at the edge of the baffles. In flower 'A' or flower 'B' baffles the angle of flip of the stream lines are reduced and thus the pressure drop is minimum.
3. Pressure variations at intervals the STHX with single, flower 'A' and flower 'B' sort baffles severally for a rate of flow of 0.3 kg/s on the shell facet and 0.7533 kg/s on the tube facet. it's discovered that the pressure drop of increasing order is as follows: one. Flower 'A', 2. Flower 'B', and 3. Single segmental baffle.
4. The slope of the pressure drop is found to extend with increase in mass flow. Additional the mass flow, steeper is that the curve profile. From fluid dynamics, a boundary layer develops at the channel wall that insulates the centre of the fluid from the warmth transfer surface. This decreases the speed of warmth transfer. Conversely, if the flow endlessly meets resistance, it's effectively stirred. This allows a bigger quantity of the flow to directly contact the warmth transfer surface earlier within the flow path.
5. Because the volumetric flow of the tube aspect fluid is multiplied from 0.7533 to 0.9533 Kg/s, the heat transfer constant multiplied from 2211 to 2470 W/(m2 /K). this is often as a result of increase within the volumetric flow will increase the mass flow in a very much faster rate than over all heat transfer constant or the heat energy transferred. Since the particular heat remains nearly constant, tube outlet temperature ought to decrease to suits law of conservation of energy. Because the flow of tube aspect fluid is multiplied, the tube aspect heat transfer constant will increase, that in turn decreases fin effectiveness and surface effectiveness.
6. Flower 'B' Baffles are simpler than Flower 'A' Baffles as they reduce the Pressure Drop to constant extent as that of Flower 'A' baffles however with a far better thermal performance associated.

5. REFERENCE

[1.] Saurabh Sharma, Ritesh Kumar Dewangan (2017), "Review on Shell and Tube Heat Exchanger (STHX) using Various Orientation Angle of Baffle", International Journal of Engineering Sciences & Research Technology, Vol. 6, Issue 10, PP: 366-369.

[2.] Mohammed Irshad, Mohammed Kaushar, G. Rajmohan (2017), "Design and CFD Analysis of Shell and Tube Heat Exchanger'', International Journal of Engineering Science and Computing, Volume 7 Issue No.4, PP: 185-190.

[3.] Bansi D. Raja, R.L. Jhala, Vivek Patel (2017), "Many-objective optimization of shell and tube heat exchanger", Thermal Science and Engineering Progress, Vol. 2, PP: 87-101.

[4.] D. Gireesh and J. Bala Bhaskara Rao (2017), "Design and Analysis of Heat Exchanger with Different Baffles", International Journal & Magazine of Engineering, Technology, Management and Research, Vol. 4, Issue 5, PP: 625-636.

[5.] Mohammed Irshad, Mohammed Kaushar, G. Rajmohan (2017), "Design and CFD Analysis of Shell and Tube Heat Exchanger", International Journal of Engineering Science and Computing, Vol. 3, Issue 4, PP: 6453- 6457.

[6.] Piyush Gupta, Avdhesh Kr. Sharma, Raj Kumar (2017), "Thermal Design of Shell and Tube Heat Exchanger Using Elliptical Tube", International Journal of Engineering and Manufacturing Science, Vol. 7, Issue 1, PP: 47-50.

[7.] Digvendra singh, Narayan Das Pal (2016), "Designing and Performance Evaluation of a Shell and Tube Heat Exchanger using Ansys", International Journal of Scientific Engineering and Applied Science, Volume-2, Issue-3, PP: 427-446.

[8.] Abazar Vahdat, Azada Nader Vahdat (2016), "Application of nanofluids for the optimal design of shell and tube heat exchangers using genetic algorithm", Case Studies in Thermal Engineering, Vol. 8, PP: 198-206.

[9.] Rahul Singh, Divyank Dubey, Harish chandra Thakur (2016), "CFD Analysis of Shell and Tube Heat Exchanger", International Journal for Scientific Research & Development, Vol. 3, Issue 12, PP: 791-796.

[10.] K. Maheswari Devi and G.V. Nagamani (2015), "Design and Thermal Analysis of Shell and Tube Heat Exchanger by Using Fluent Tool", International Journal & Magazine of Engineering, Technology, Management and Research, Vol. 2, Issue 9, PP: 359-366.