CFD ANALYSIS OF SHELL AND TUBE HEAT EXCHANGER DESIGN USING ANSYS FLUENT

Kavita Singh[1], P. S. Dhakar[2]
[1]PG Scholar NITM Gwalior, Madhya Pradesh, India
[2] Asst. Professor NITM Gwalior, Madhya Pradesh, India

Abstract: - The aim of the task is plan of shell and cylinder heat exchanger with helical tube and examine the stream and temperature field inside the shell using ANSYS programming tool. Three types of heat exchangers are planned in this examination with various structures of cylinders contains of 20 mm breadth and 1000 mm length shell measurement 210 mm. To expand the warmth trade limit of heat exchanger advancement is done which tries to distinguish the best parameter combination of heat exchangers. The prefix parameter (tube width) is utilized as an info variable and the yield parameter is the most extreme temperature distinction of shell and tube heat exchanger. Three types models are design on the basis tubes varieties of heat exchanger and CFX examination is completed in ANSYS 14.0.

Keywords: Shell and tube heat exchanger, Ansys, Temperature, Heat transfer coefficient, thermal analysis.

I. Introduction

The procedure of heat transfer between two liquids at various temperatures and parted by a solid partition is found in many building applications. The equipment used to actualize such heat exchange process is named as a heat transfer device. A heat exchanger is a device in which two liquid streams, one hot and one cold, are carried into thermal contact with each other in order to exchange transfer heat from the hot liquid stream to the cool one. It gives a moderately large surface area of heat exchange for given volume of the equipment. heat exchangers might be characterized based on contacting methods, development, flow arrangement plan or surface compactness. A shell and tube heat transfer devices are largely demanded in process plants. heat exchangers are one of the generally utilized in the process plant. heat transfer devices are utilized to exchange heat between two types of fluid. One can understand their use that any procedure which include chilling, heating, condensation, boiling or evaporation will need a heat transfer device for that type of work. Process liquids, typically are warmed or cooled before the procedure or experience a stage change. several heat exchangers are named by their application. For example, heat transfer devices being utilized to condense are called as condensers, also heat exchanger for boiling purpose are called boilers. performance and strength of heat exchangers are studied through the measure of heat transfer utilizing minimum territory of heat exchange and weight drop. A superior introduction of its proficiency is finished by computing over all heat exchange coefficient. Weight drop and region required for a specific measure of heat exchange, gives a knowledge about the capital expense and power necessities of a heat exchanger.

II. Flow Arrangement

There are two essential groupings of heat exchangers as per their fluid flow system. In parallel tubes heat exchangers, the two liquids pass in the exchanger at a similar end, and move in parallel to each other to the opposite side. In counter-stream heat transfer device, the fluids enter the shell from other ends. The countercurrent structure is most effective, in that it can exchange the most heat from the heat (exchange) medium. See countercurrent transfer. In a cross-flow heat transfer, the liquids make a trip generally perpendicular to each other through the exchanger.

For effectiveness, heat exchangers are intended to enlarge the surface territory of the wall between the two liquids, while decreased resistance from liquid move through the exchanger. The exchanger's execution can also be influenced by the expansion of fins or layering’s in one or the two directions, which increment surface region and may counter liquid stream or induce turbulence.

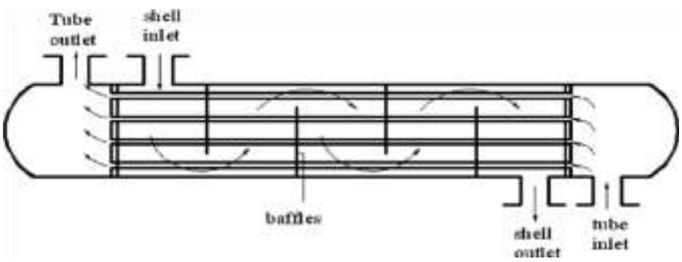

Figure 1: Shell and Tube Heat Exchanger with one shell pass and one tube pass

The main temperature over the heat exchange surface shifts with position, however a proper means temperature can be characterized. In most basic frameworks this is the "log mean temperature difference" (LMTD). In some cases, coordinate information of the LMTD isn't accessible and the NTU strategy is utilized.

III. Methodology

As per study it is found that CFD examination includes mainly three types of steps are described:

- **Pre-Processing:** This is starter stage of CFD simulation process which assists with explaining geometry in great suitable way. The stream domain of choice is partitioned into equivalent number of smaller parts known as components. There is distinctive Pre-Processing programming accessible are CFD-GEOM, ANSYS, Meshing, ANSYS, ICEM CFD, T Grid and so on. Pre-preparing this incorporates characterizing the problem, making the 3D display, getting the model to Ansys workbench, meshing, and applying physical working condition called boundary conditions.
- **Solving or Processing:** When the issue material science has been studied, fluid properties, stream physical science demonstrate, confine conditions are situated to handle using PC. There is eminent business programming available for this including: CFD++, Open FOAM, ANSYS CFX, Star CCM, ANSYS FLUENT etc. Using this item, it is possible to comprehend the managing conditions related to stream material science issue Handling incorporates unwinding of numerical or logical states of fluid stream until the point when the moment that participating in result is expert. Normally, it requires the PC to comprehend an enormous number of conditions and may take couple of hours to few days.
- **Post processing:** The last walk in the wake of getting the results from the solver is to inspect the results with different procedures like weight and speed shape plots, vector plot, streamlines, temperature form so forth right when the model has been grasped, the results can be separating both numerically and graphically. Post getting ready is about observation either in fundamental 2-D to 3-D depictions.

IV. Modelling and Analysis of Heat Exchanger

A software model is proficient by utilizing amounts of shell, tubes and baffles in ANSYS 14.0, Workbench structure modeler is utilized to develop Heat exchanger geometry and for further investigation. Geometry is made simple by offering a leeway for the plane symmetry. This heat exchanger is counter flow type and the tube side consists of one inlet and one outlet representing three types of design of tubes in heat exchanger. After model is designed, run thermal simulation of heat exchanger. The consequences overcome were discreetly studied with general condition.
Figures shows the three type of variations of tubes design in heat exchanger model in Ansys CFD.

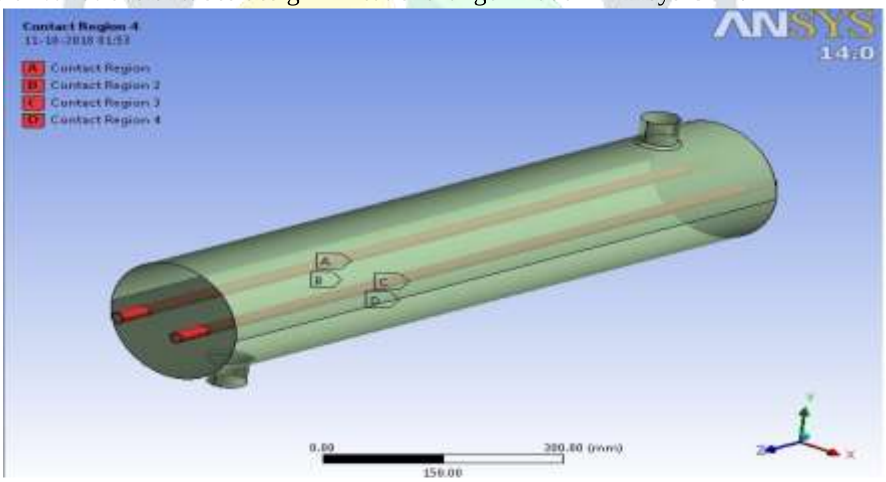

Figure 2: Parallel pipe flow tube heat exchanger

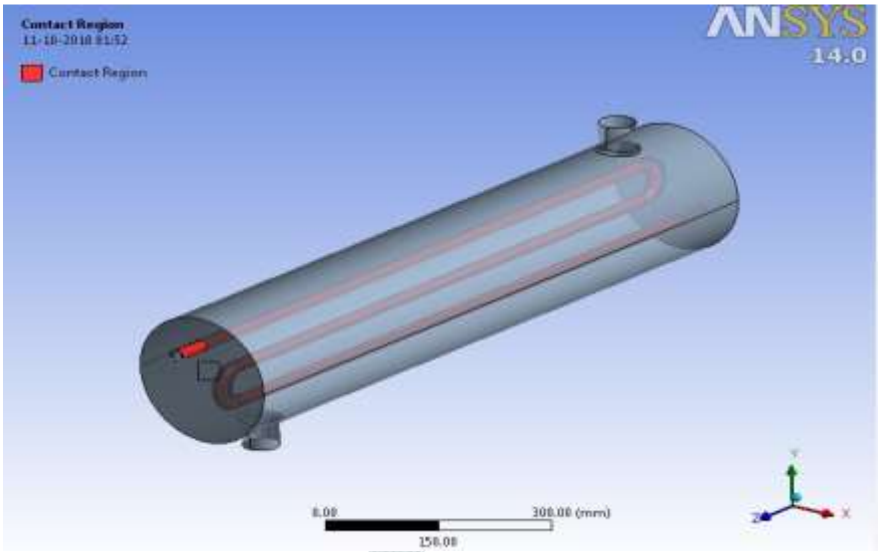

Figure 3: S Pattern pipe flow tube heat exchanger

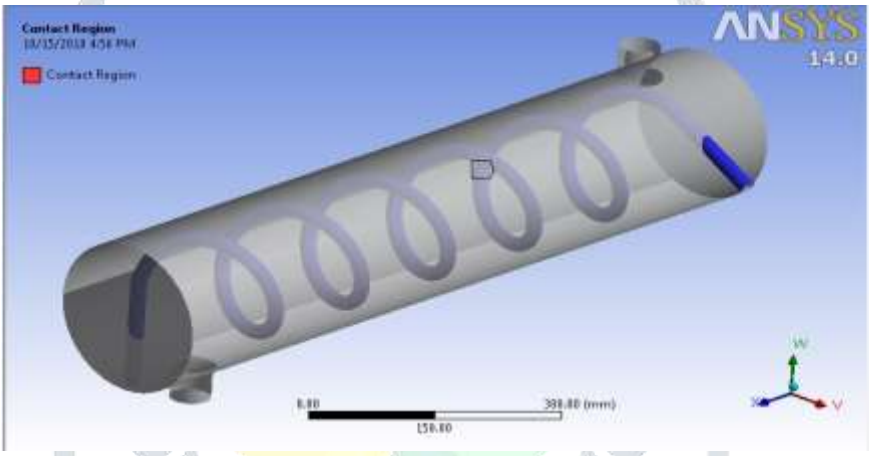

Figure 4: helical pipe flow tube heat exchanger

V. Boundary conditions:

Limit conditions are characterized: Inlet – Velocity delta
Inlet Outlet – weight outlet (zero guage Pressure)
Dividers – convective heat exchange, no slip criteria.
Liquid Material – Water and Water vapor
metal Material – copper and Steel
Energy condition – ON
Turbulence Model - K-epsilon, feasible, Standard wall treatment demonstrate
Solution Method – Second Order

VI. Results and Discussions

As per CFD analysis of All three models of heat exchangers with the variations in tubes we found the inlet outlet temperatures.

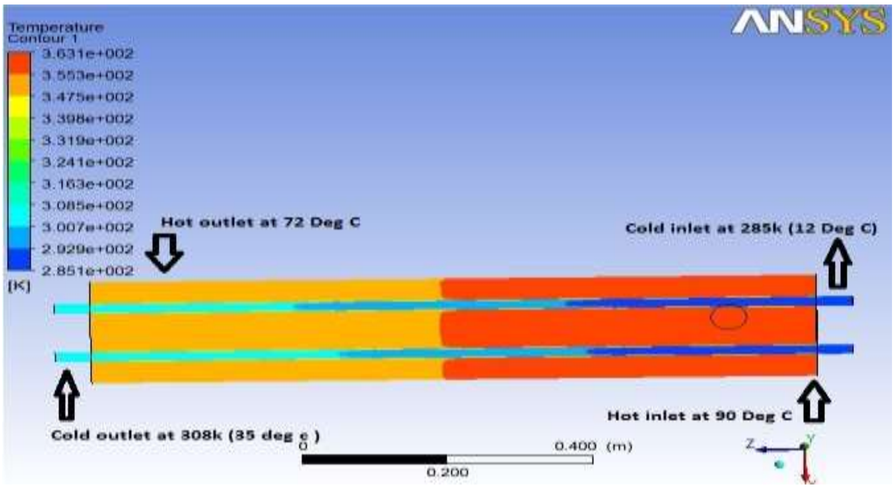

Figure 5: Parallel pipe flow temperature

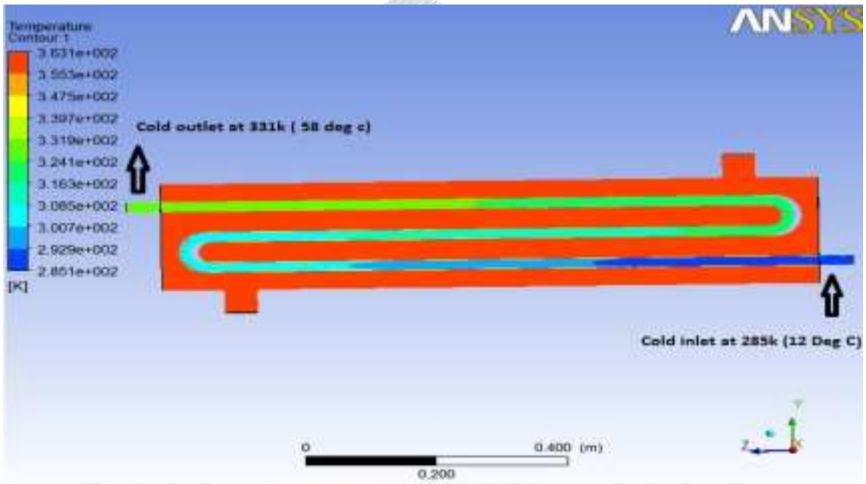

Figure 6: S Pattern pipe flow temperature

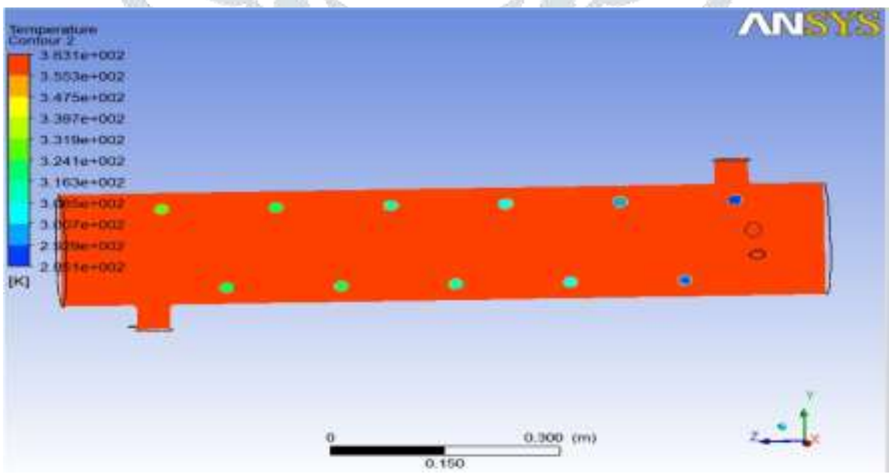

Figure 7: helical Pipe flow temperature

As per study figure shows the variations of temperature difference. In above study three types of heat exchangers designed by using Ansys. Different types of tube used in heat exchanger i.e. Parallel flow tubes, 'S' pattern tubes, and helical flow pattern tubes. After designing cold water inlet temperature given 12°C and hot water temperature given at inlet is 90°C. as per CFD analysis heat exchange in designed exchanger and results optimized.

Table 1: Cold water flow temperatures at inlet and outlet

Pattern of Tubes	Cold inlet (°C)	Cold outlet (°C)
Parallel tubes	12	35
‘S’ pattern tubes	12	65
Helical tubes	12	58

Table 2: Hot water Temperatures at inlet and outlet

Pattern of Tubes	Hot inlet (°C)	Hot outlet (°C)
Parallel tubes	90	72
‘S’ pattern tubes	90	81
Helical tubes	90	82

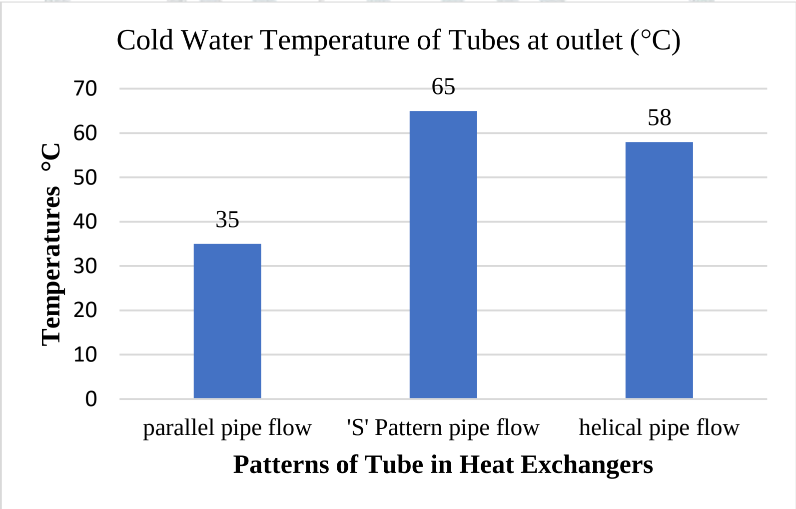

Figure: Cold water temperatures with Patterns of Tube in Heat Exchangers

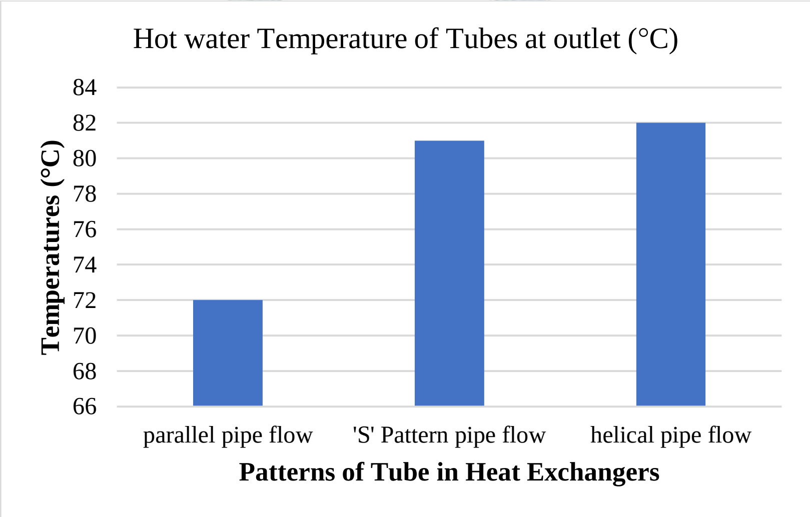

Figure: Hot water temperatures with Patterns of Tube in Heat Exchangers

VII. Conclusion

From this study we select Copper material to the tubes and steel material to shell design then studied in Ansys and optimized the best possible value of temperature variations amongst the discussed materials. so that it may be assumed that no heat transfer is taking place in between shell and surroundings. as per results it is concluded that 'S' pattern tube design gives better heat transfer in heat exchanger as comparison to others.
Parallel flow tubes give 35°C temp at outlet which is decreases from 12°C. 'S' pattern tubes give 65°C temperature at outlet and Helical tube pattern gives 65°C temperature of cold-water flow at outlet. so after comparing these results we found the maximum heat transfer at 'S' pattern of heat exchanger. so 'S' pattern design is optimum design for maximum heat transfer.

References

1. M.D. Rajkamal, M. Mani Bharathi, Shams Hari Prasad, "Thermal Analysis of Shell and Tube Heat Exchanger", International Journal of Pure and Applied Mathematics, Volume 119 No. 12 2018, 14299-14306.
2. Shuvam Mohanty, Shofique Uddin Ahmed, "Performance prediction of Counter flow Heat Exchanger by using CFD technique", IJEDR, Volume 6, Issue 2, ISSN: 2321-9939.
3. Ankush S. Patil, H. S. Farkade, "Advances in Design and Development of Heat Exchangers: A Review", International Research Journal of Engineering and Technology (IRJET), Volume: 04, Issue: 05, May -2017.
4. Chandan Kumar Sethi, "CFD Analysis on Effectiveness of a Plate Type Heat Exchanger Using Sea Water and Engine Oil", International Journal of Advanced Mechanical Engineering, Volume 12, Number 1 (2017) pp. 191-198.
5. Deepa Shrivastava, Rahul Mishra, "Cfd Analysis of Heat Transfer for Tube-In- Tube Heat Exchanger", International Journal of Recent Trends in Engineering & Research (IJRTER), Volume 03, Issue 08, August – 2017.
6. Dipankar De, Tarun K. Pal, Santanu Bandyopadhyay, "Helical baffle design in shell and tube type heat exchanger with CFD analysis", International Journal of Heat and Technology, Vol. 35, No. 2, June 2017, pp. 378-383.
7. Kvenumadhav, Sudhanshu Kumar, Chandrashekar Goud, "Performance Analysis of Different Heat Exchanger Design Using Cfd Simulation", IJRAET, Volume 6, Issue 2 OCT 2017.
8. Mohammed Irshad, Mohammed Kaushar, G. Rajmohan, "Design and CFD Analysis of Shell and Tube Heat Exchanger", IJESC, Volume 7 Issue No.4, 2017.
9. Q. Miao, Z. Wang, L. Wang, "A Failure Analysis of Air Heat Exchanger Based on Modal Calculation and Thermal-Structural Interaction Simulation", International Journal of Materials, Mechanics and Manufacturing, Vol. 5, No. 3, August 2017.
10. Basawaraj S. Hasu, J. Govardhan, Satyanarayana Rao, "Thermal Analysis of Tubular Heat Exchangers Using Ansys By Using Alloy Materials as Tube Materials", Anveshana's International Journal of Research in Engineering and Applied Sciences, Volume 1, Issue 3, 2016.
11. K Ashok Reddy, "A Review of Heat Transfer Studies for Shell & Tube Heat Exchangers", IJSDR, Volume 1, Issue 5, May 2016.
12. J. Bala Bhaskara Rao, V. Ramachandra Raju, "Numerical and heat transfer analysis of shell and tube heat exchanger with circular and elliptical tubes", Rao and Raju International Journal of Mechanical and Materials Engineering Volume 11, Issue 6, DOI 10.1186/s40712-016-0059-x.
13. Mayank Bhola, Vinod Kumar, Satyendra Singh, "Heat Transfer Enhancement in Concentric Tube Heat Exchanger in ANSYS FLUENT", International Journal of Engineering Research & Technology (IJERT), Vol. 4 Issue 04, April-2015.
14. Pankaj D. Lad, Santosh G. Taji, Baban B. Londhe, "Optimization of Concentric Pipe Heat Exchanger as Per Asme Code using Finite Element Analysis", International Journal on Recent Technologies in Mechanical and Electrical Engineering (IJRMEE), Volume: 2 Issue: 10, ISSN: 2349-7947, 2015.
15. Roshan. V. Marode, Ashok. J.Keche, "Thermal Analysis Validation for Different Design Tubes in a Heat Exchanger", International Journal of Engineering Research and General Science Volume 3, Issue 1, January-February, 2015.