Comparative Thermal Performance Analysis of Shell and Tube Heat Exchanger With and Without Fins by Using CFD

Prof. Gajanan P. Nagre[1], Prof. Nandakishor D. Bankar[2], Prof. Dipak V. Kakde[3]

[1,2,3]*Department of Mechanical Engineering, MGM's Polytechnic, Aurangabad (MH, India)*

ABSTRACT

Heat exchanger is a device that used to transfer thermal energy between two or more fluids, between solid surface and fluid or solid particulates and fluid, at different temperature and in thermal contact. The transfer of heat to and from process fluids is an essential part of most chemical processes. The most commonly used type of heat-transfer equipment is the shell and tube heat exchanger. The present work for study is directed towards the three dimensional computational fluid dynamics (CFD) simulation using commercial software ANSYS FLUENT have been performed to study and compare shell side flow distribution, and temperature variations at low shell side flow rates, velocity variations and heat transfer rate are investigated by numerically modeling small heat exchanger , by inserting meshing, boundary condition, running calculations, inserting surface parameters and flow trajectories to visualize the resulting flow field. Thermal analysis of shell and tube heat exchanger involves rating and sizing of heat exchanger. Rating problem deals with determination of rate of heat transfer, outlet temperature etc. While sizing problem involves selection of tube material, flow arrangement, determining physical size of heat exchanger. In this numerical comparison heat exchanger consisting of shell, tubes, baffles are modeled; numerical model predicts the thermal performance with considerable good accuracy and comparing the flow simulation result with experimental data for single segmental baffles. Also extending the work by doing CFD analysis for shell and tube heat exchanger with fins over the tubes and comparing these results with the simulation results obtained from Ansys for without fins.

Keywords: *segmental baffle, shell and tube heat exchanger, CFD etc.*

---

1. INTRODUCTION

Heat exchangers are one of the most widely 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 heating, cooling, condensation, boiling, or evaporation will require a heat exchanger for this purpose. Process fluids, usually are heated or cooled before the process or undergo a phase change. Performance and efficiency of heat exchangers are measured through the amount of heat transfer using least area of heat transfer and outlet temperature. Usually there is lots of literature and theories to design a heat exchanger according to requirement.

2. WORKING PRINCIPLE

Shell and Tube Heat Exchangers are one of the most popular types of exchanger due to the flexibility the designer has to allow for a wide range of pressures and temperatures. A shell and tube heat exchanger consists of number of tubes in cylindrical shell. Two fluids can exchange heat, one fluid flows over the outside of the tubes while the second fluid flows through the tubes. The fluids can be single or two phase and can flow in a parallel or a cross/counter flow arrangement. Shell and tube type heat exchanger, indirect contact type heat exchanger. For this particular study shell is considered, which generally one pass shell.

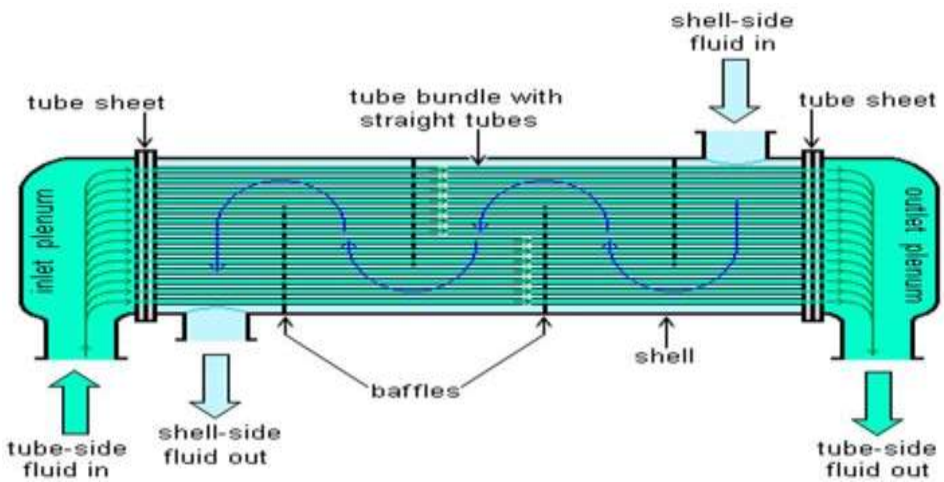


Figure2.1 Shell and tube heat exchanger [8]

A shell is the most commonly used due to its low cost and simplicity. Baffles are used to support the tubes for structural rigidity, preventing the tube vibration and to divert the flow across the bundle to obtain higher heat transfer coefficient. Baffle spacing (B) is the center line distance between two adjacent baffles. In order to improve the thermal performance at a reasonable cost of shell and tube heat exchanger, tubes in the present study are provided with corrugation instead of using plane tubes. Heat exchanger is one of devices that are convenient in industrial and household application. These include power production, chemical industries, food industries, electronics, environmental engineering, manufacturing industry, and many others. Computational fluid dynamic is now an established industrial design tool, offering obvious advantages. In this study, CFD model of shell and tube heat exchanger is considered. By modeling the geometry as accurately as possible, the flow structure and the temperature distribution inside the shell are obtained [1]

3. LITERATURE REVIEW

Dhavalkumar Maheshwari et.al [1] In this work he has done a comparative study on Thermal designing and analysis of U-tube Heat Exchanger with Plain tube and corrugated tube by doing firstly thermal designing calculation by TEMA [Tubular Exchanger Manufacturer Association] Standard of U-tube Heat Exchanger. and then this type of U-tube heat exchanger with Plain tube compare with Corrugated tube for different characteristics parameters like heat transfer rate, overall heat transfer coefficient, efficiency, effectiveness, pressure drop etc. in order to increase heat transfer rate, efficiency, heat transfer area and reduce pressure drop. Corrugated tubes have different characteristics parameters which are Height to Diameter ratio (e/DH), Relative pitch ratio (p/e), Relative helix angle (β).

Digvendra Singh et.al [2] Present paper deals with the Designing and Performance Evaluation of a Shell and Tube Heat Exchanger using Ansys. Heat exchanger has a variety of applications in different industries and in this study one such heat exchanger is taken in to account. The heat exchanger is designed as per the commercial needs of the industry. Kern’s technique is used to design the heat exchanger .The designing procedure results in a shell and tube heat exchanger having 21 tubes, 170mm shell diameter and 610 mm long. As the designing procedure doesn’t include the type of the header to be used, so we have analyzed three types of header which can provide a uniform velocity in the inlet of each tube. Different geometries are included in different positions of the inlet nozzle for the header. CFD simulations are used for the optimum positioning of the inlet nozzle which could be proposed from the uniform distribution of the liquid methanol and the uniform velocity distribution though each and every tube. The main objective of this paper is to verify the heat exchanger designed with the use of the Kern’s technique, by the use of Commercial Computational Fluid Dynamics (CFD) software. For the simulation, purpose a symmetric view of the simplified geometry of the heat exchanger is made using Solid works software. In the present study, CFD simulation is used to study the temperature and velocity profiles through the tubes and the shell. It is found that the heat transfer through the length of the tube is not uniform.

Shweta Y Kulkarni et.al [3] The most commonly used type of heat exchanger is the shell and tube heat exchanger (STHE). In the present study, a comparative analysis of a water to water STHE wherein, hot water flows inside the tubes and cold water inside the shell is made, to study and analyze the heat transfer coefficient and pressure drops for different mass flow rates and inlet and outlet temperatures, using Kern and Bell Delaware methods. This paper purely aims at studying and comparing different methods of Shell and Tube Heat Exchanger and bringing out which method is better for adopting in shell side calculations.

Amol S. Niphade et.al [4] work presented in this paper shall focus on different design alternatives for the heat exchanger with support offered volume of about five thousand liters of milk per day. Mathematical modeling coupled with computational methodology shall be explored for ramping up the volume in excess of twenty thousand litters. ANSYS Fluent shall be deployed for finding solution while mathematical model shall offer alternative methodology for validating the solution. Conventional heat exchanger design method does not predict steady state uniform property performance well; and they are totally unable to predict the influences of time dependence and varying properties or the consequent stresses in the shell and tubes. On the other hand, conventional computational fluid dynamics (CFD) techniques, with their emphasis on body-fitting grids and sophisticated turbulence models, can contribute only to small-scale phenomena such as the velocity and temperature distributions within the space occupied by a few-tube sub-section of a tube bank. Nevertheless, the practical importance of heat exchangers, including those which involve chemical reaction and phase change, is so great that engineers must find design tools which are both economically-affordable and more realistic in prediction than either of the just-mentioned extremes.

Hemant Kumar et.al [5] in this paper, the shell and tube heat exchanger is considered in which hot water is flowing inside one tube and cold water runs over that tube. Computational fluid dynamics technique which is a computer based analysis is used to simulate the heat exchanger involving fluid flow, heat transfer. CFD resolve the entire heat exchanger in discrete elements to find the temperature gradients, pressure distribution and velocity vectors. The turbulence model k-ε is used for accurate results from CFD. The temperature variations are calculated from experiment for parallel and counter flow by varying the mass flow rate of fluid of 2LPM and 3LPM. The solid geometry is made in solid works software and then imported into Gambit which is the pre-processor of the ANSYS 13.0 for meshing the model geometry. Using the post processor FLUENT, the simulated results are computed i.e. temperature contours, pressure contours and velocity vectors. Then, simulated results are validated with the experimental values. The analysis shows that there is a difference between temperatures values computed from the experiment and the simulation by ANSYS 13.0. CFD helps to design the heat exchanger by varying the different variables very easily otherwise it is very difficult if done practically. CFD models or packages provides the contours and data which predict the performance of the heat exchanger design and are effectively used because it has ability to obtain optimal solutions and has work in difficult and hazardous conditions

4. OBJECTIVE OF PAPER

1) The main objective of this project is design and simulation of shell and tube heat exchanger with segmental baffle using CFD and measuring the thermal performance that is heat transfer, temperature variation, velocity etc.

2) Also doing the CFD analysis for plane tube with circular fins over the tubes along with baffle.

3) Comparing these results for both the case with and without fins and finding which is effective for improving the performance of heat exchanger.

5. COMPUTATIONAL FLUID DYNAMIC

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. [4] This software can also build a virtual prototype of the system or device before can be apply to real-world physics to the model, and the software will provide with images and data, which predict the performance of that design. 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. Basic Approach to using CFD[8]

a) **Pre-processor:** Establishing the model
1) Identify the process or equipment to be evaluated.
2) Represent the geometry of interest using CAD tools.
3) Use the CAD representation to create a volume flow domain around the equipment containing the critical flow phenomena.
4) Create a computational mesh in the flow domain.

b) Solver:
1) Identify and apply conditions at the domain boundary.
2) Solve the governing equations on the computational mesh using analysis software.

c) Post processor: Interpreting the results
1) Post-process the completed solutions to highlight findings.
Interpret the prediction to determine design iterations or possible solutions, if needed

6. SYSTEM DEVELOPMENT

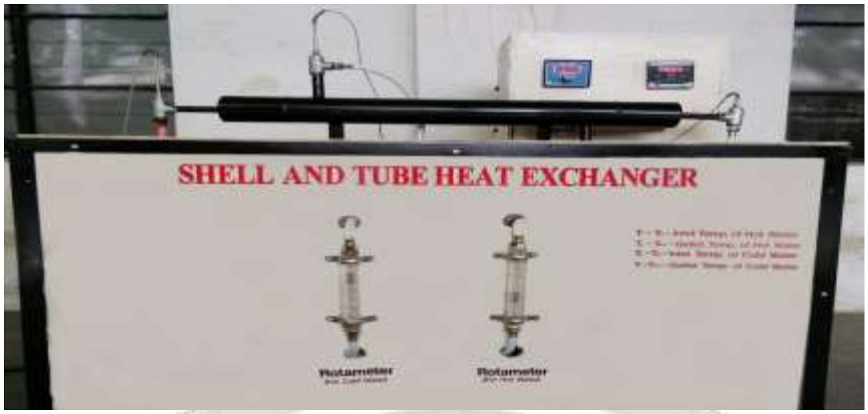
Table 6.1 dimension of shell and tube heat exchanger

Construction Data	
Tube Length	900 mm
Number Of Shells	1
Tube Outside Diameter	16 mm
Tube Outside Diameter	14 mm
Tube thickness	1 mm
Number Of Tubes	07
Heat Transfer Area	0.316 m2
Tube Material	Copper
Number of Tube Passes	1
Tube Pitch & Orientation	30 mm
Shell outer Diameter	99 mm
Shell inner Diameter	94 mm
Shell thickness	2.5 mm
Shell Material	Stainless steel
No. of baffles	04
Baffles Spacing	180 mm
Baffle cut	25% (23.5 mm)
Baffle thickness	3 mm

Specification of experimental setup

Tank with two Heaters
Fiber tank (50 lit) for storing cold & hot water.
Platform made from angles and plywood
Piping system – hose pipes for transportation of cold water.
Piping system – hose pipes for transportation of hot water.
Welding of the heat exchanger to the platform.
Rota meter (5LPM) for measuring the flow.
Magnetic pump – 2 (40 watt

The experimental setup for the counter flow heat exchanger by LMTD method is shown in figure.



Geometry and Mesh
Model is designed according to Tubular Exchanger Manufacturers Association (TEMA)

Geometry creation: CAD model of the shell and tube heat exchanger in .IGES or STEP format. This can be imported in ANSYS ICEM CFD and with few hours of CAD clean-up, the CFD domain can be extracted. The baffle details, including the baffle inclination and baffle spacing, will be provided through scaled drawings. Using these dimensions and with the help of geometry creation tools in ANSYS ICEM CFD the baffles can be included in the shell side domain of heat exchanger.

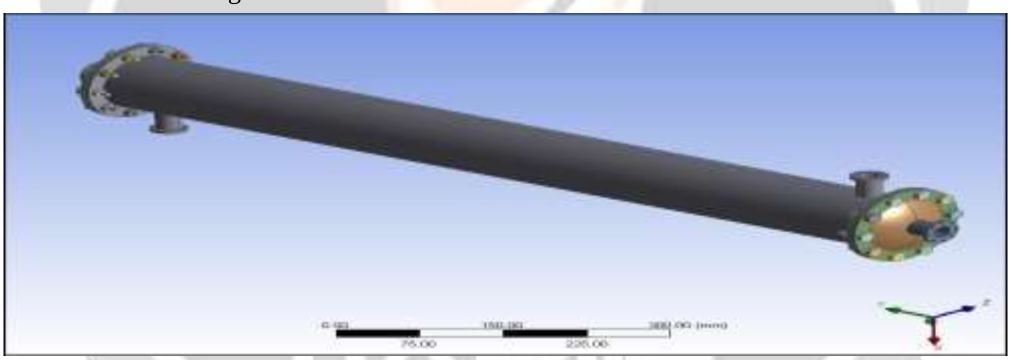


Figure 6.1 isometric view of arrangement of shell and tube heat exchanger with segmental baffle

Meshing:

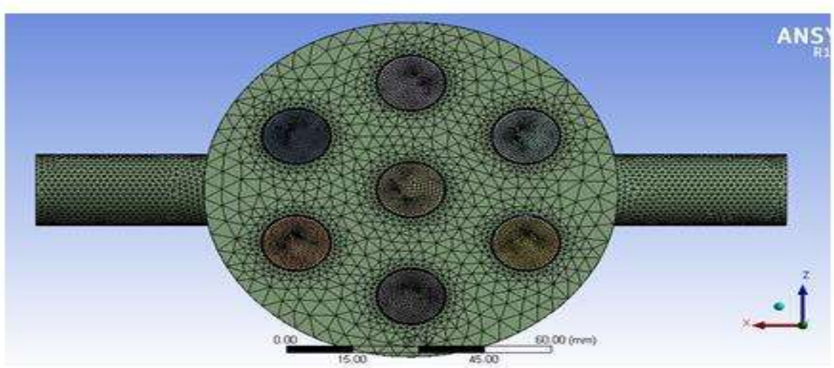


Figure 6.2 meshing diagram of shell and tube heat exchanger

Initially a relatively coarser mesh is generated. This mesh contain mixed cells (Tetra and Hexahedral cells) having both triangular and quadrilateral faces at the boundaries. Care is taken to use structured cells (Tetrahedral) as much as possible, for this reason the geometry is divided into several parts for using automatic methods available in the ANSYS meshing client.it is meant to reduce numerical diffusion as much as possible by structuring the mesh in a well manner, particularly near the wall region

Problem Setup:

Simulation was carried out in ANSYS FLUENT 17. In the fluent solver pressure based type was selected, absolute velocity formation and steady time was selected for the simulation. In the model option energy calculation was on and viscous was set as standard K-e, standard wall function (K-epsilon 2 eqn).

In the cell zone fluid water-liquid was selected. Water-liquid and copper was selected as material for simulation. Boundary condition was selected for inlet and outlet. In the inlet and outlet 0.0175 m/s velocity and temperature was set at 343K .Across each tube 0.0175 m/s velocity and 303K temperature was set. Mass flow was selected in each inlet. In the reference value area as 1m2, density 998 kg/m3, length 1m, temperature 353k, velocity 0.0175 m/s, ratio of specific heat 1.4 was considered

7. RESULT AND DISCUSSION

Contours

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

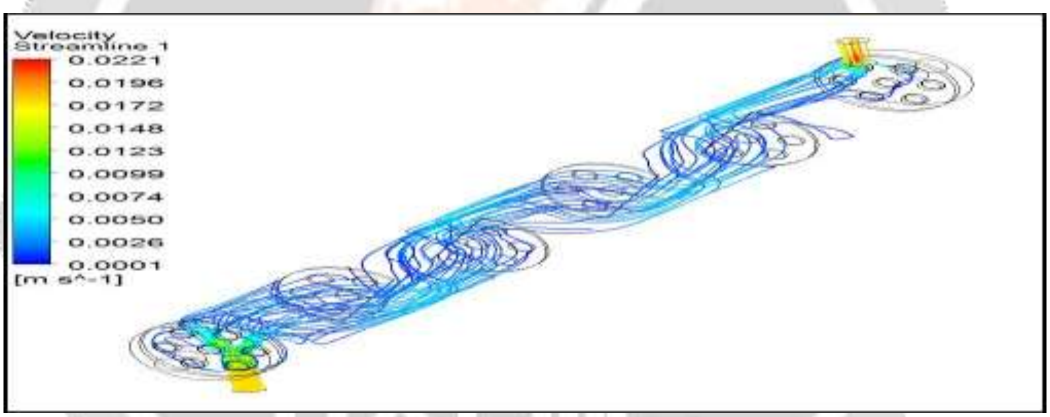


Figure 7.1 velocity streamline of shell for plane tubes without fins

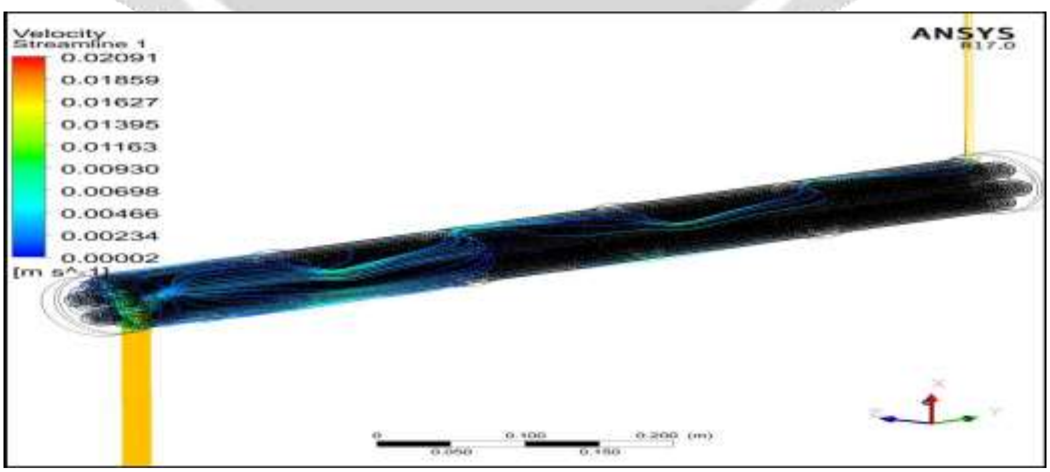


Figure 7.2 velocity streamline of shell for plane tubes with fins

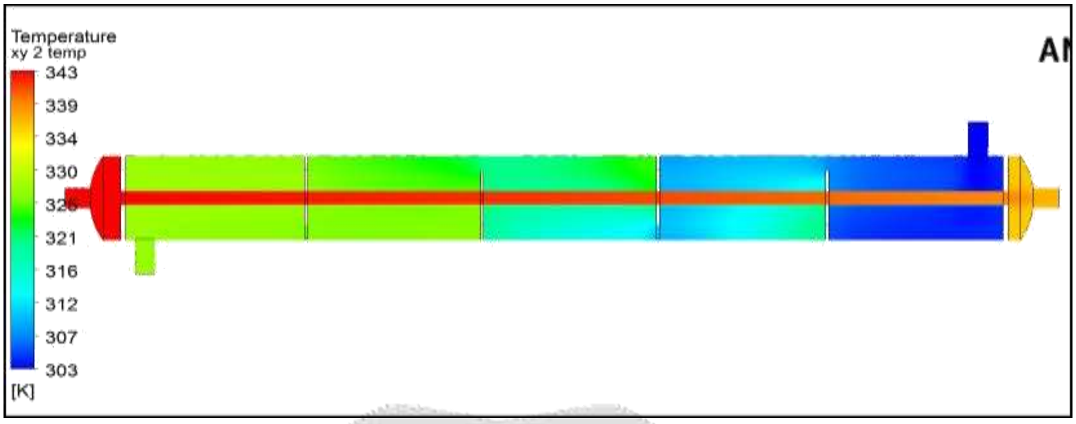


Figure 7.3 contour of temperature for plane tubes without fins

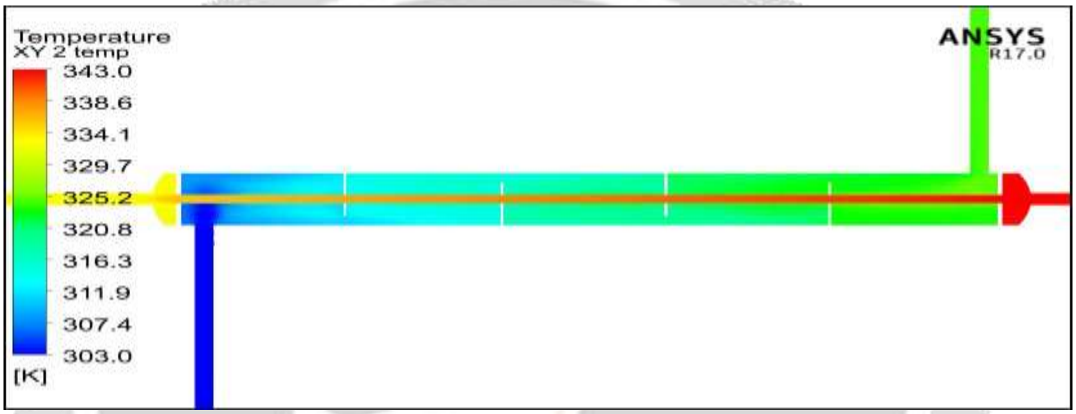


Figure 7.4 contour of temperature for plane tubes with fins

Without fins: As the shell and tube side velocity is kept 0.0175 m/s temperature at outlet of tubes is decreasing and pressure is almost same while temperature of cold water at outlet increases by 24 °C for this simulation was carried out for given model of heat exchanger and desired result are found and statement are concluded. Temperature at inlet is suddenly increasing after passing through some distance temperature is got constant almost so we will get jump in respective graph.

With fins: In this case velocity as input parameter is kept same for both shell side and tube side that is 0.0175 m/s the temperature at outlet of tubes is decreasing because of additional surfaces of fins total time for contacting surface to tubes will be decreasing in this case fins are impeding water flow so less heat is wasted compare to without fins design. With fins at the outlet of tubes the temperature is decreasing but comparing without fins. Decreasing temperature at outlet is always higher heat transfer is increasing at inlet velocity.

Comparison of Result:

Table7.1. Comparison of Ansys result with experimental result

Parameter	CFD result	Experimental result
Cold water inlet temperature	303 K	303 K
Cold water outlet temperature	327 K	317 K
Hot water inlet temperature	343 K	343 K
Hot water outlet temperature	336 K	335 K

Table7.2 comparison of Outlet temperature of cold water for without fins and with fins

Parameter	CFD result for plain tube without fins	CFD result for plain tube with fins and baffle
Cold water inlet temperature	303 K	303 K
Cold water outlet temperature	327.00 K	328.49 K
Hot water inlet temperature	343 K	343 K
Hot water outlet temperature	336.12 K	333.12 K

8. CONCLUSION

From CFD simulation we can observe that the outlet temperature of cold water with same inlet velocity 0.0175 m/s increases in heat exchanger model with fins and that is because we are increasing surface area to increase heat transfer rate values from the table and graph are mean to the results. Maximum difference in temperature i was getting is 1.5°C. Conventional methods used for the design and development of Heat Exchangers are expensive. CFD provide alternative to cost effectiveness speedy solution to heat exchanger design. The Results Shows that the Maximum heat transmission takes place in the circular fin with baffle having the counter flow of the water.

9. FUTURE SCOPE

Future scope from my study is that different types of fins can be used over tubes so as to increase more efficiency of heat exchanger. This work can be extended by using different working fluid for heat exchanger. Analysis can be done in ANSYS workbench by varying the mass flow rate for a given heat exchanger.

10. REFERENCES

[1] Dhavalkumar Maheshwari, Kartik Trivedi, "Experimental Investigation Of U-Tube Heat Exchanger Using Plain Tube And Corrugated Tube", International Research Journal Of Engineering And Technology (IRJET), Volume: 03 Issue: 05 May-2016

[2] Digvendra singh, Dr. Jitendra Pandey, Dr. Abhishek Tiwari, "Designing and Performance Evaluation of a Shell and Tube Heat Exchanger using Ansys (Computational Fluid Dynamics)", International Journal of Scientific Engineering and Applied Science (IJSEAS) Volume-2, Issue-3,March 2016.

[3] Shweta Y Kulkarni , Jagadish S B , Manjunath M B, "analysis comparing performance of a conventional shell and tube heat exchanger using kern, bell and bell Delaware method", International Journal of Research in Engineering and Technology,Volume: 03 Special Issue: 03 May-2014.

[4] Amol S. Niphade, H. A. Chavan,"Design, CFD Analysis of Shell & Tube Heat Exchanger with CFD Analysis for Dairy Application", International Journal of Advance Foundation And Research In Science & Engineering, Volume 2, Issue 9 , March 2016.

[5] Gurbir Singh, Hemant Kumar,"Computational Fluid Dynamics Analysis of Shell and Tube Heat Exchanger", Journal of Civil Engineering and Environmental Technology, Volume 1, Number 3; August, 2014

[6] Praveen Kumar Kanti, Karthika.U.P, Sabeer Ali, Sanath Kumar, N Shyam Chandran C, "CFD analysis of shell and tube heat exchanger", International Journal of Engineering Research, Volume No.5 Issue: Special 6, 20 May 2016

[7] Rahul Singh, Divyank Dubey, Harishchandra Thakur, "CFD Analysis of Shell and Tube Heat Exchanger", International Journal for Scientific Research & Development, Vol. 3, Issue 12, 2016

[8] Rajiv Mukherjee,"effectively design shell and tube heat exchanger", chemical Engineering progress, February 1998.

[9] Mohammad Reza Saffariana, Tooraj Yousefib and Mostafa Keshavarze Moraveji, “Numerical Analysis of Shell and Tube Heat Exchanger with Simple Baffle by CFD”,Indian J.Sci.Res. 7, 2014

[10] Kirubadurai.B, R.Rajasekaran, K.Kanagaraj, P.Selvan “Heat Transfer Enhancement of Shell and Tube Heat Exchanger”, International Journal for Research in Applied Science and Engineering Technology, Vol. 2 Issue VIII, August 2014

[11] Swapnaneel Sarma, D.H.Das “CFD Analysis of Shell and Tube Heat Exchanger using Triangular fins for waste heat recovery processes”, Engineering Science and Technology: An International Journal, Vol.2, No.6, December 2012

[12] Manu P S, “Comparison of Heat Transfer in Shell and Tube Heat Exchanger with Modified Baffles using Ansys Fluent”, International Journal of Engineering Research & Technology (IJERT), Vol. 5 Issue 02, February-2016

[13] Santosh Kansal, Mohd. Shabahat Fateh, “Design and Performance Evaluation of Shell and Tube Heat Exchanger using CFD Simulation”, International Journal of Engineering Research & Technology, Vol. 3 Issue 7, July – 2014.

[14] K.Maheswari Devi, G.V. Nagamani, “Design and Thermal Analysis of Shell and Tube Heat Exchanger by Using Fluent Tool”, International Journal & Magazine of Engineering, Technology, Management and Research,Volume No: 2, 2015

[15] Jitendra K Prajapati, Dilip B Vaghela, Dipendra A Thakur, “Parametric Optimization & CFD Analysis of Shell And Tube Type Heat Exchanger”, International Journal of Advance Engineering and ResearchDevelopment, Volume 2,Issue 8, August -2015

[16] Mayank Vishwakarma,“Thermal analysis of Helical Baffle in Heat Exchanger”, International Journal of Science and Research, Volume 2 Issue 7, July 2013

[17] Eldhose Lomy, Nidheesh P, “Optimization of Shell and Tube Heat Exchanger for Tube Arrangements”, New York, 2nd International Conference on Emerging Trends in Mechanical Engineering, Vol.4, Special Issue 12, September 2015

[18] Avinash D Jadhav, Tushar A Koli, “CFD Analysis Of Shell And Tube Heat Exchanger To Study The Effect Of Baffle Cut On The Pressure Drop”, International Journal Of Research In Aeronautical And Mechanical, Vol.2 Issue.7, July 2014.

[19] S.prabakaran, P.Gopi ,“CFD Integrated Optimum Design and Prototyping of Shell and Tube Heat Exchanger”, Global Research and Development Journal for Engineering,Volume1 Issue 4, March 2016

[20] P.S.Gowthaman and S.Sathish, “Analysis of Segmental and Helical Baffle in Shell and tube Heat Exchanger”, International Journal of Current Engineering and Technology, Special Issue-2, February 2014

[21] G.V.Srinivasa Rao, Dr. C.J.Rao, Dr.N.Haribabu, “Heat Transfer Analysis on Shell and Tube Heat Exchangers”, International Journal Of Research In Aeronautical And Mechanical Engineering, Vol.2 Issue.1, January 2014.

[22] A.Gopi Chand, A. V. N. L. Sharma, G. Vijay Kumar, A.Srividya,“ Thermal Analysis Of Shell And Tube Heat Exchanger Using Mat Lab And Floefd Software”, International Journal Of Research In Engineering And Technology, Volume: 01 Issue: 03, Nov-2012

[23] Holman, J.P. (2004) Heat Transfer, 9th Edition, Tata McGraw-Hill, Inc., New York.