COMPARATIVE THERMAL PERFORMANCE STUDY ON SHELL AND TUBE HEAT EXCHANGER WITH SEGMENTAL BAFFLE BY USING CFD

Vikas Lokawar[1*] Harsh Sharma[2*]
Deogiri Institute of Engineering and Management Studies,Aurangabad

ABSTRACT

In this study an experimental results of a small scale Shell and Tube Heat Exchanger (STHX) were compared with Computational Fluid Dynamics (CFD) simulation results. STHX is a widely used heat exchanger in industries hence it is essential to study and improve its performance. This heat exchanger is of Liquid-Liquid category with cold water flowing through the shell-side and hot water is through the tube-side of this heat exchanger. The cold water at 30° C flows through the shell-side while the hot water at 70°C flows on the tube-side. The cold water flow rate is maintained constant while the hot water flow rate on the tube was varied to study the heat exchange characteristics. The CAD model of STHX is designed according to TEMA (Tubular Exchanger Manufacturers Association) standards and the CFD simulation is done for the same in ANSYS ICEM. The heat transfer calculations were achieved by using Logarithmic Mean Temperature Difference (LMTD) method. We also employed the Kern and Bell-Delaware method which can be used accurately for shell and tube heat exchangers. The calculations and simulations are done for counter flow of the heat exchanger.

Keywords: *Shell and Tube Heat Exchanger, CFD, Design, Analysis*

INTRODUCTION

In an indirect contact type of heat exchanger the heat is exchanged in between two fluids, Shell and Tube heat exchanger is an indirect contact type of heat exchanger in which one fluid flows through the shell side and other flows through tube side. STHX is mainly consisting of shell, tubes and baffles. Baffles play a very important role, it disturbs the shell side flow and increase the contact time with tube side fluid hence heat transfer rate increases. Baffles have different types such as segmental, helical, slotted etc. Segmental baffles are the most commonly used baffle type hence the segmental baffles with 25% cut is used in this study. [10]

Figure 1. Baffle (25% Cut)

While obtaining the solution for any component designs such as Heat Exchangers, the experimental methods are always preferred as compared to the analytical and numerical (CFD or FEA) approaches. This was mainly due to the reason that the component undergoes actual operating conditions as per the specifications. With the application of the actual working conditions, the performance of the components shall be investigated in detail. However, the experiments must conducted by adhering to the relevant standards, data reduction techniques etc to be considered for producing quality results from such a study. This experimental study for the shell and tube heat exchanger was conducted and the measurements such as inlet, outlet temperatures were noted during the experiments. It was critical to record these temperature values after obtaining the steady-state conditions.

Experimental Procedure

In this study cold water flow rate (Shell side) was maintained at constant (4 liter/min) at an inlet temperature of 30 °C while the hot water flow rate (Tube side) was varied as 2, 4, 6 and 8 liter/min with an inlet temperature of 70°C. In order to calculate the heat exchanger performance parameters such as log-mean temperature difference, heat exchanger effectiveness, the outlet temperatures need to evaluate from experiment study. The procedure includes First, Started the cold water supply to the tank and subsequently to the shell side of the heat exchanger simultaneously, water supply to the hot water tank was also imitated. Here, the heater was turned to supply the thermal energy to the water and hence increasing the water temperature. Then the flow rate was controlled with the help of flow control valves. So that the required water flows rates were maintained. The cold water flow rate was kept at 4 l/min while the hot water flow rate was maintained at 2 l/min. Hence, steady condition for the water flow rate was established. At the beginning, the heat exchanger was filled with atmospheric air. As the water starts filling, the air will be removed from the heat exchanger. However, it was ensured that there were no bubbles in the water flow lines by supplying the water continuously. After satisfying this condition the temperature reading has been recorded. With the electrical energy supply, the temperature of the water was increasing. Heat supply was controlled such that the hot water inlet temperature was maintained at 70°C to ensure steady thermal operating conditions were in place. At this stage, with the steady conditions for the flow and thermal operating conditions have been met (time = 0 s), the inlet and outlet temperature of cold as well as hot water were monitored and were noted on the observation table. In order to prevent any errors, this was continued for a certain time period and the temperature measurements were observed for the pre-determined time intervals.

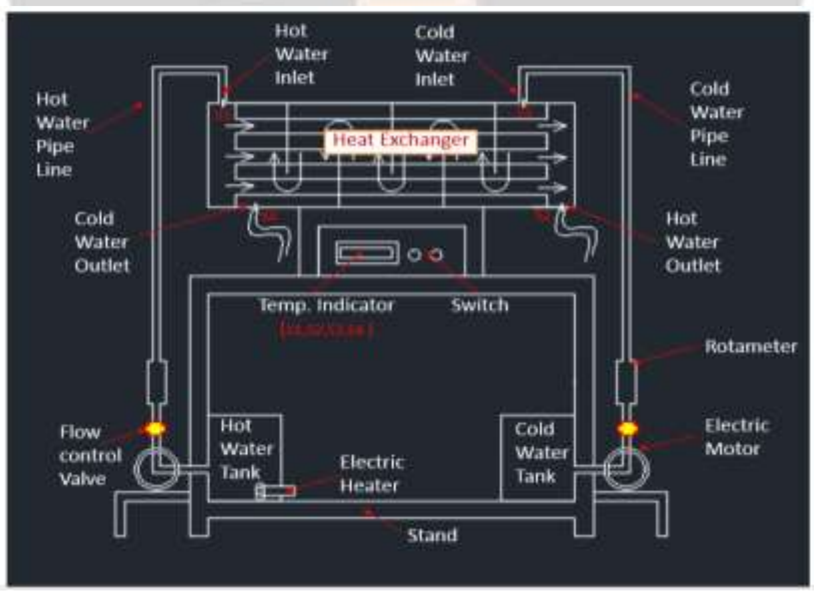

Figure 2. Schematic representation of the experimental setup

CFD (Computational Fluid Dynamics)

Computer based simulations had been gaining popularity in developing engineering components such as Aircraft, IC Engines, Heat Exchangers etc. These computer based simulation methods such as FEA (Finite Element Analysis), CFD (Computational Fluid Dynamics) help in reducing time required for product development as well as cost. Another benefit from these methods is the ability to a detailed results analysis with the advanced post-processing tools that are available in these software's.

Computational Fluid Dynamics (CFD) method employs the concept of solving the discretized flow governing equations (Navier Stokes Equations) in finite shaped cells to obtain the local flow and thermal field. In order to solve these discretized governing equations, advanced computer algorithms had been developed such as Multi-grid schemes etc. In general, CFD approach is based on the three discretization methods such as 1) Finite Difference Methods (FDM) 2) Finite Element Methods (FEM) 3) Finite Volume Methods (FVM).

The modern day CFD software systems are based on the finite volume methods as this approach enables conservation principles by default. There have been many CFD software systems available like ANSYS FLUENT, ANSYS CFX, STAR CCM+, Open FOAM etc. Each of this software has their unique advantages over others for particular applications.

CFD simulation approach provide the following advantage over the traditional experimental approach 1) Cost reduction 2) Faster to obtain the solutions 3) Ability to conduct Parametric Analysis in short time 4) Detailed post-processing. However, the results from the CFD simulations still need to be validated using the experimental methods during the product development.

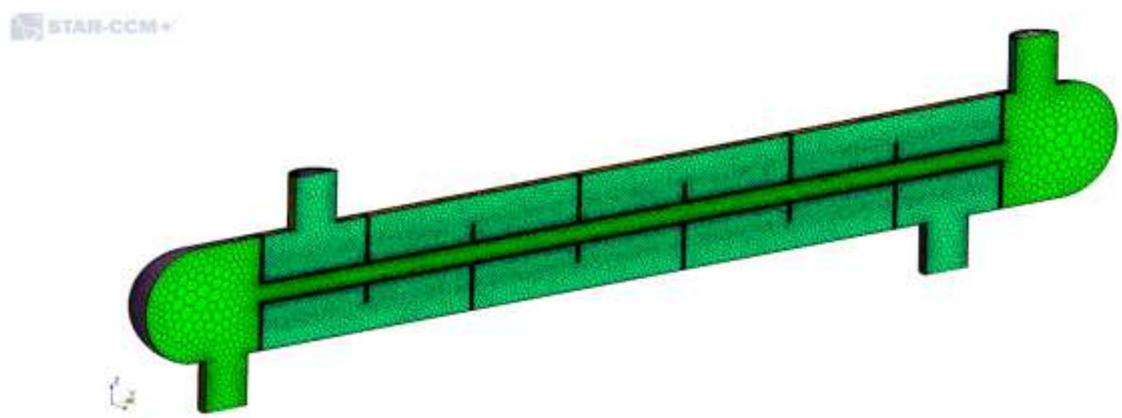

Figure 3. Meshing for the STHX geometry

Rating of shell and Tube Heat Exchanger

A heat exchanger can be designed by the LMTD when inlet and out let conditions are specified. When the problem is to determine the inlet and outlet temperatures for a particular heat exchanger, the analysis is performed more easily by using a method based on effectiveness of the heat exchanger and number of transfer units (NTU). The heat exchanger effectiveness is defined as the ratio of actual heat transfer to the maximum possible heat transfer.

Effectiveness of heat exchanger

€ = actual heat transfer/ maximum possible heat transfer = Q / Qmax _ _ _ _ [02]

Actual heat transfer Q can be determined by energy balance

Q = mh Cph (th1-th2) = mc Cpc (tc2-tc1) _ [02]

Fluid capacity rate C:

mh Cph = Ch = hot fluid capacity rate

mc Cpc = Cc = cold fluid capacity rate

Cmin = minimum fluid capacity rate (Ch or Cc)

Cmax = maximum fluid capacity rate (Ch or Cc)

The effectiveness $€ = C_h (t_{hi} - t_{ho}) / C_{min} (t_{hi} - t_{ci}) = C_c (t_{co} - t_{ci}) / C_{min} (t_{hi} - t_{ci}$ _ _ _ _ _ _ [02]
Tube side heat transfer coefficient by

At = π di2 / 4
Where di = tube inner diameter
Atp = Nt At / No of passes
Where Nt = no. of tubes

Gt = mt / Atp
Where Gt = mass velocity of tube
Atp = heat transfer area based on tube surface

Ut = Gt / ρ
Where ρ = density of fluid at average temperature

Ret = Ut ρ di / μ _ [02]
Where di = tube inner diameter
μ = viscosity
ρ = density of fluid at average temperature

Using the petukhov and Kirillov coorelation
Nu = (f/2)RePr/ 1.07+12.7(f/2)0.5(Pr2/3-1) _ _ _ _ _ _ _ _ _ _ _ _ _ _ _ _ [02]
Where f = friction factor
Re= Reynolds no.
Pr = prandtl no.

Where f = (1.58 In Re -3.28)-2_ [02]
The tube side heat transfer coefficient is then found as
h= Nu*k / di

RESULTS AND DISCUSSION

For the fixed cold water flow rate (4 l/min), the temperature rise increases as the hot water flow rate was increased. This is due to the high energy that was available with the increase in hot water flow rate. In the pressure drop image Figure 5, the pressure drop across the cold side (shell-side of the heat exchanger) remained constant because the cold water flow rate is constant. On the other hand, the pressure drop for the hot side (Tube side of the heat exchanger) is increasing with the increase in the hot water flow rate. A similar observation table for the remaining three configurations – hot water flow rate of 4 l/min, 6 l/min and 8 l/min – were prepared and details and did further calculations – Refer Table 1. The results of experimental study and CFD Simulations are compared and shown in Figure 6.

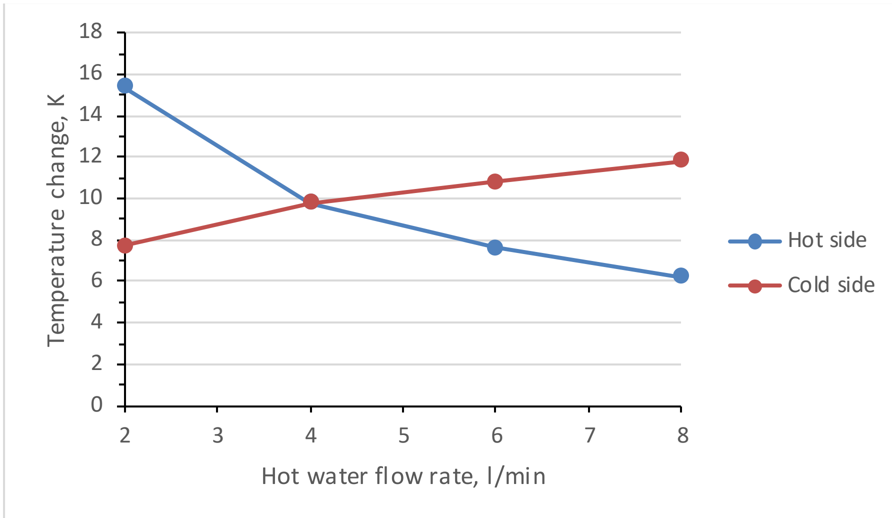

Figure 4. Temperature drop for the STHX

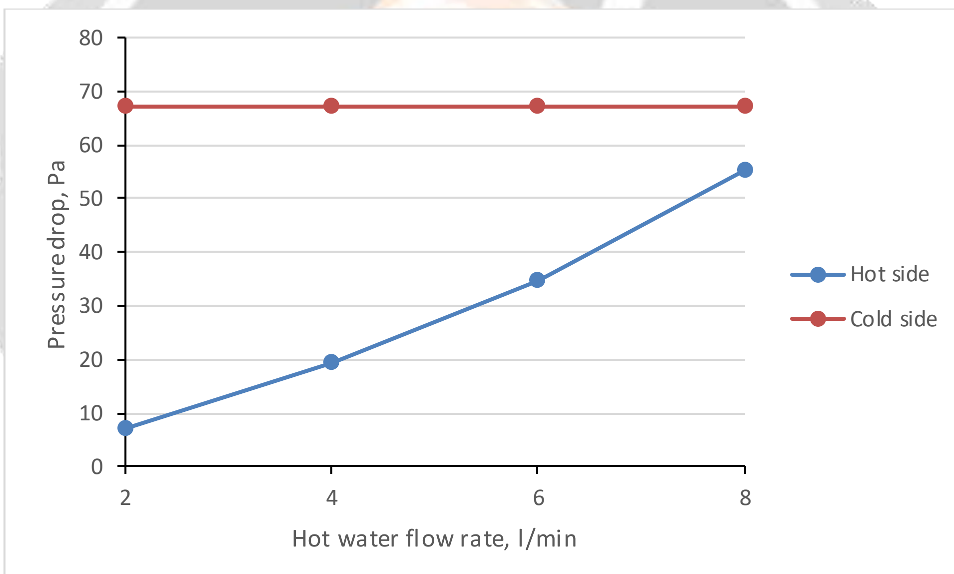

Figure 5. Pressure drop for the STHX

Cold water flow rate		4 liters / minute		
Hot water flow rate		2 liters / minute		
	Cold Side (Shell-side)		Hot Side (Tube-Side)	
Time, s	Inlet Temperature, °C	Outlet Temperature, °C	Inlet Temperature, °C	Outlet Temperature, °C

0	30	30	70	70
120	30	31.8	70	67
240	30	34	70	64.1
360	30	35.4	70	61.2
480	30	36	70	58.3
600	30	36.6	70	56.4

Table 1. Results from the experimental studies

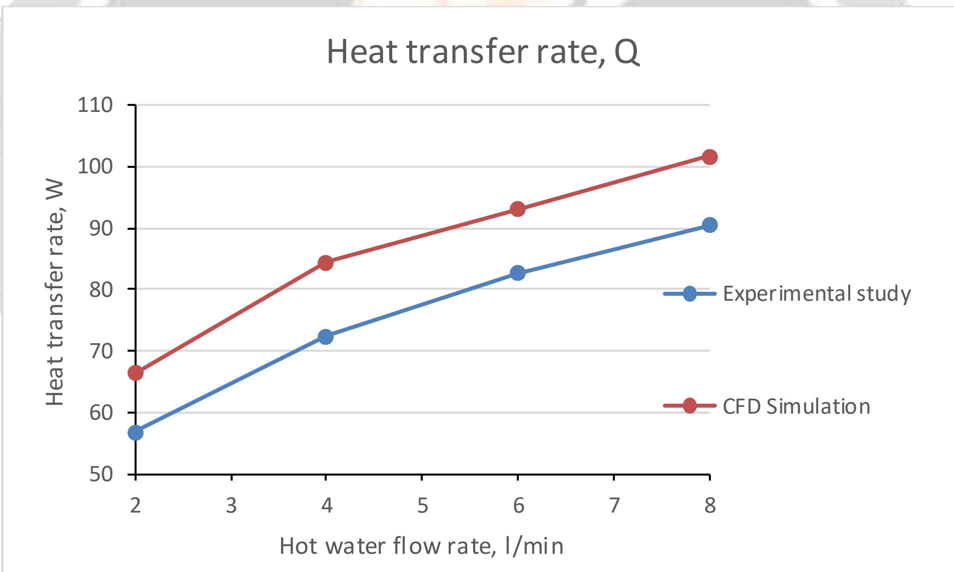

Figure 6. Heat transfer rate vs flow rate

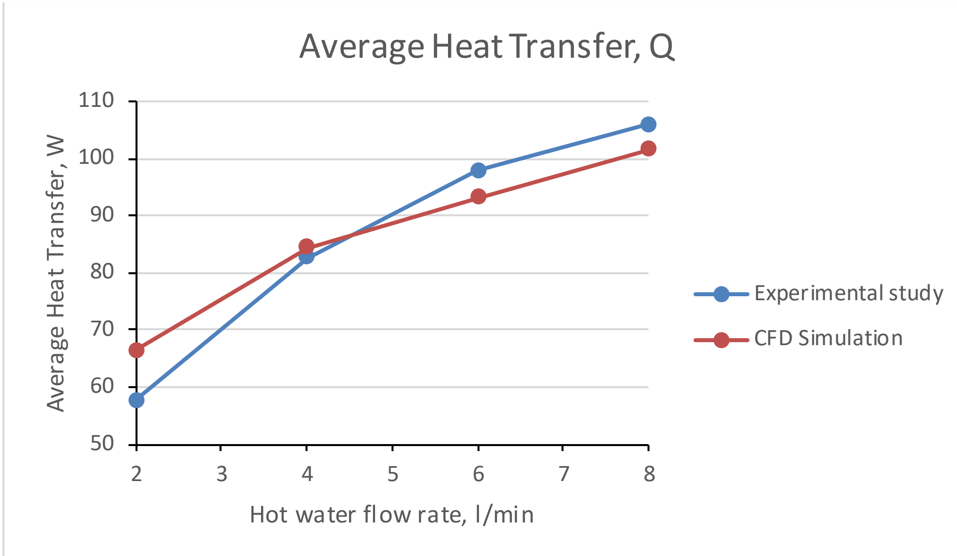

Figure 6. Average Heat transfer rate vs flow rate

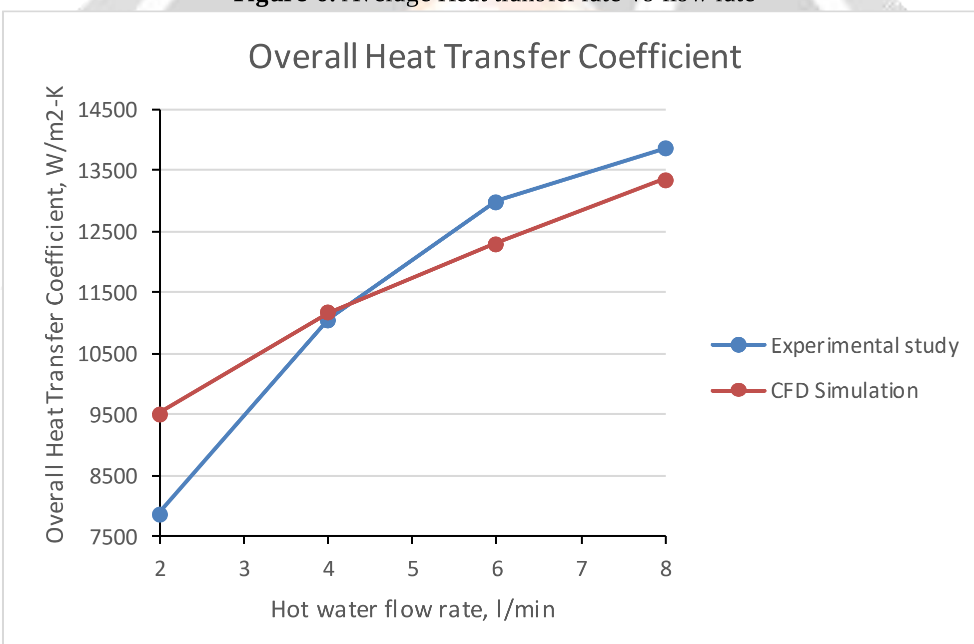

Figure 6. Overall heat transfer coefficient vs flow rate

CONCLUSION

Shell and tube heat exchanger is most widely used heat exchanger in the industries now days. In this comparative experimental study we focused on important factors like Heat transfer rate, average heat transfer and overall heat transfer coefficient and it is found that experimental result are coming within 14% variation with CFD result. The small scale heat exchangers showed numerous flow re-circulation zones as well as un-utilization zones near the segmental baffles. The 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. With this small variation one can do the development in the STHX first in the CFD simulation, if results are good then it can be manufactured for the experimental study.

FUTURE SCOPE

As the Conventional methods used for the design and development of Heat Exchangers is expensive one can do the further study in CFD simulation. One may consider the factors for the future study in CFD simulation is like using fins on the tubes, helical baffle with different inclination instead of Segmental baffle etc.

NOMENCLATURE

B baffle spacing (mm)
B_c baffle cut (%)
C_p Specific heat capacity (J/kg K)
$\dot{m}$ mass flow rate (kg/s)
N_t number of tubes
N_b number of baffles
D_s internal shell diameter (mm)
D_o external tube diameter (mm)
P_t tube pitch (mm)
T_{in} inlet temperature
T_{out} outlet temperature
A_o heat exchanger area based on external diameter of tube (mm^2)
u average velocity(m/s)
L tube total effective length (mm)
ΔP pressure drop (pa)
ΔT_m logarithmic mean temperature difference
Re Reynolds number
U overall heat transfer coefficient
Q_{avg} average heat transfer rate (W)
y^+ diamensional wall distance

Greek Symbols

µ Dynamic viscosity
ρ density (kg/m^3)
ε Turbulent kinetic energy dissipation rate
Ґ generalized diffusion coefficient
υ kinematic viscosity
α thermal diffusivity (m^2/s)

LIST OF ABBREVIATIONS

STHX - Shell and tube heat exchanger
CFD- Computational Fluid Dynamics
TEMA -Tubular Exchanger Manufacturers Association
LMTD-Logarithmic Mean Temperature Difference
FEA-Finite element analysis

REFERENCES

[1] Mohammad Hadi Mohammadi, Hamid Reza Abbasi, Adel Yavarinasab, Hossein Pourrahmani, Thermal optimization of shell and tube heat exchanger using porous baffles, Applied Thermal Engineering,Volume 170, 2020,115005,ISSN 1359-4311, 2020.

[2] 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

[3] Ammar Ali Abd, Mohammed Qasim Kareem and Samah Zaki Naji, Performance Analysis of Shell and Tube Heat Exchanger: Parametric Study, Case Studies in Thermal Engineering, 2019.

[4] Danish Thondiyil, Saijal Kizhakke Kodakkattu, Optimization of a shell and tube heat exchanger with staggered baffles using Taguchi method, Materials Today: Proceedings,Volume 46, Part 19, 2021.

[5] Muhammad Mahmood Aslam Bhutta, Nasir Hayat, Muhammad Hassan Bashir, AhmerRais Khan, KanwarNaveed Ahmad, Sarfaraz Khan5, CFD Applications In Various Heat Exchangers Design: A Review, Department Of Mechanical Engineering, University Of Engineering & Technology, Applied Thermal Engineering, 2011.

[6] Jian-Fei Zhang, Ya-Ling He, Wen-Quan Tao, A Design And Rating Method For Shell-And-Tube Heat Exchangers With Helical Baffles, Journal Of Heat Transfer, May 2010

[7] Rajagapalthundilkaruppa raj and Srikanthganne, "Shell side numerical analysis of a shell and tube heat exchanger considering the effects of baffle inclination angle on fluid flow", ThundilKaruppa Raj, *et al*.: Shell Side Numerical Analysis of a Shell and Tube Heat Exchanger ,THERMAL SCIENCE: Year 2012, Vol. 16, No. 4, pp. 1165-1174.

[8] Sachin Kallannavar, Suresh Mashyal, Manik Rajangale, Effect of tube layout on the performance of shell and tube heat exchangers,Materials Today: Proceedings,Volume 27, Part 1,2020.

[9] M. Thirumarimurugan, T.Kannadasan and E.Ramasamy, Performance Analysis Of Shell And Tube Heat Exchanger Using Miscible System, American Journal Of Applied Sciences 5 (5): 548-552, 2008

[10] Usman Ur Rehman, Heat Transfer Optimization of Shell-And-Tube Heat Exchanger through CFD Studies, Chalmers University of Technology, 2011

[11] A.O. Adelaja, S. J. Ojolo and M. G. Sobamowo, "Computer Aided Analysis of Thermal and Mechanical Design of Shell and Tube Heat Exchangers", Advanced Materials Vol. 367 (2012) pp 731-737 © (2012) Trans Tech Publications, Switzerland.

[12] Apu Roy, D.H.Das, CFD Analysis Of A Shell And Finned Tube Heat Exchanger For Waste Heat Recovery Applications, National Institute Of Technology, 2011.

[13] Dhavalkumar A. Maheshwari, Kartik M. Trivedi, "A Review on Experimental Investigation of U-Tube Heat Exchanger using Plain Tube and Corrugated Tube", International Journal of Engineering Development and Research (IJEDR), ISSN:2321-9939, Volume.3, Issue 4, pp.255-259, 31 October 2015,