International Journal of Engineering, Science and Mathematics
Vol. 6 Issue 1, March 2017,
ISSN: 2320-0294 Impact Factor: 6.765
Journal Homepage: http://www.ijmra.us, Email: editorijmie@gmail.com
Double-Blind Peer Reviewed Refereed Open Access International Journal - Included in the International Serial Directories Indexed & Listed at: Ulrich's Periodicals Directory ©, U.S.A., Open J-Gage as well as in Cabell's Directories of Publishing Opportunities, U.S.A

EFFECT OF BAFFLE CUT SIZES ON TEMPERATURE AND PRESSURE DROP AT VARIOUS MASS FLOW RATE IN A SHELL AND TUBE HEAT EXCHANGER

A. A. Musilim [*]
A. Nwagwo [**]
O. K. Uche *

Abstract

Computational Fluid Dynamics (CFD) can be very useful to visualize the flow and the temperature fields on the shell side. In this present study, attempts were made to investigate the effects of varying baffle cut sizes at various mass flow rates on temperature and pressure drop of a shell-and-tube heat exchanger for zero degree (0°) baffle inclination angle. From the CFD results obtained it can be deduced that for an increase in baffle cut size percentage i.e. (decrease in baffle diameter) there is a reduction in the pressure drop and temperature differences in the shell and tube heat exchanger. This pressure drop and temperature differences adversely affects the overall performance of the heat exchanger and must be kept as little as possible thus showing that the performance of the shell and tube heat exchanger would increase if there is an overall increase in the baffle cut size as this would aid the flow and proper mixing and the heat transfer of the fluids.

Key words: Baffle, Temperature field, Temperature difference, Pressure drop, Heat transfer.

[*] **Department of Mechanical Engineering, Yaba College of Technology, Yaba,**

[**] **Department of Mathematics, Yaba College of Technology, Yaba, Nigeria.**

1. Introduction.

Heat exchangers have always been an important part to the life cycle and operation of many systems. A heat exchanger is a device built for efficient heat transfer from one medium to another in order to carry and process energy [1]. Typically one medium is cooled while the other is heated. They are widely used in petroleum refineries, chemical plants, petrochemical plants, natural gas processing, air-conditioning, refrigeration, and automotive applications. One common example of a heat exchanger is the radiator in a car, in which it transfers heat from the water (hot engine-cooling fluid) in the radiator to the air passing through the radiator [2].

Baffles are used for directing the flow inside the shell from the inlet to the outlet while maintaining effective circulation of the shell side fluid, hence providing effective use of the heat transfer area. Segmental baffles are introduced inside the cover pipe of a typical shell and tube heat exchanger to increase the heat transfer rate [3 – 5]. Furthermore, performance of tubular heat exchanger can be improved by helical baffles instead of conventional segmental baffles [6 – 7]. Also, proper baffle inclination angle and orientations can also provide an optimal performance of heat exchanger [8 – 9]. Baffle is provided with a cut (%) which is expressed as the percentage of the segment height to shell inside diameter. In general, baffle cut can vary between 15% and 45% of the shell inside diameter. This cut allows the fluid to pass through in parallel or counter flow direction. A number of baffles are placed along the shell in alternating orientations (cut facing up, cut facing down, cut facing up again, etc.) in order to create flow paths across the tube bundle (forming cross flow windows). Baffle is provided with a cut (BC) which is expressed as the percentage of the segment height to shell inside diameter. Baffle cut can vary between 15% and 45% of the shell inside diameter. In the present study 25%, 35%, and 45% BC is considered [10].

This heat exchanger shown in Plate I is generally built of a round tubes mounted in a cylindrical shell with the tube axis parallel to that of the shell [11]. One fluid flows inside the tubes, the other flows across and along the tubes. The major components of this exchanger are tubes, shell, and front-end head, rear-end head, baffles and tube sheets. A variety of different internal constructions is used in shell-and -tube exchangers, depending on the desired heat transfer and pressure drop performance and the methods employed to reduce thermal stresses, to prevent leakages [12].

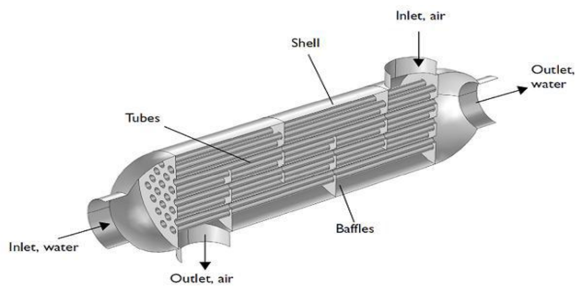

Plate I: Shell and tube heat exchangers

2. Computational Modelling.

Computational model requires the application of extensive computational resources in studying the behaviour of complex systems by computer simulation. The system under study is often a complex nonlinear system for which simple, intuitive analytical solutions are not readily available. Rather than deriving a mathematical analytical solution to the problem, experimentation with the model is done by adjusting the parameters of the system in the computer, and studying the differences in the outcome of the experiments. Operation theories of the model can be deduced from these computational experiments.

Computational modelling includes three key steps which are pre-processing, processing and post-processing.

2.1 Geometry modelling.

The geometries of the shell and tube heat exchanger and baffle designs are described below. The modelling process was carried out with the aid of computer aided design software namely CATIA V5 and the design model was saved as a .igs file to be imported later into ANSYS CFX. The model geometry considers a single pass fluid namely water with inlet and outlet and baffle cavities are represented as cuts in the fluid cavity. The cavity model is designed according to TEMA (Tubular Exchanger Manufacturers Association) standards guidelines as properly stated in the table below [13]:

Table- 2.1: Geometries of the shell and tube heat exchanger and baffle designs.

Heat exchanger length, L	600 mm
Shell inner diameter, D_i	90 mm
Tube outer diameter, D_o	20 mm
Tube bundle geometry and pitch triangular	30 mm
Number of tubes, N_t	7
Number of baffles, N_b	6
Central baffle spacing, B	86 mm
Baffle inclination angle, ϴ	0°

Further parameters were also implied in the model design. These parameters refer to the baffle cut sizes that have been varied in the model geometry. The baffle cuts sizes are gotten by percentage reduction of the shell inner diameter. The table below presents the data on the baffle cut percentage and corresponding baffle diameters.

Baffle Cut Percentage	Baffle Diameter
25% baffle cut	67.5 mm
35% baffle cut	58.5 mm
45% baffle cut	49.5 mm

Table-2.2: Data on the baffle cut percentage and corresponding baffle diameters.

2.2 Mesh generation.

In order to effectively analyze the fluid flow in the heat exchanger, the flow domains are splits into sub domains comprising hexahedral and tetrahedral mesh elements. The discretized governing equations are applied in each of the domains grid. Mesh setting for three fluid domains viz: fluid inlet, fluid shell and fluid outlet are detailed below:

Mesh Settings	Details
Type of Analysis	3D
Type of Mesh Elements	Tetrahedral, Wedges, and Pyramids
Physics preference	CFD
Solver preference	ANSYS CFX
Use Advance Size of Function	On Curvature
Relevance Centre	Medium
Smoothing	Medium

Table-2.3: ANSYS cfd Mesh Setting

The computational mesh was generated accordingly on application of the above mesh settings as shown below.

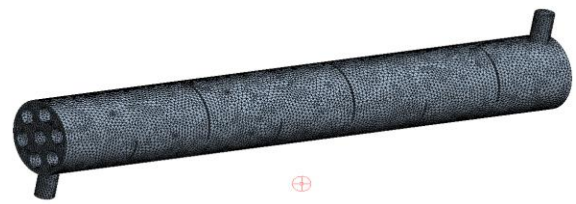

Plate II**:** Meshing of Heat Exchanger

2.3 Governing equations.

The governing equations of the flow are modified according to the conditions of the simulated case. Since the problem is assumed to be steady, time dependent parameters are dropped from the equations are:

The Navier Stokes Equation:

$$\underbrace{\rho\left(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)}_{\text{Inertial terms}}=-\underbrace{\frac{\partial p}{\partial x}}_{\text{Pressure gradient}}+\underbrace{\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right)}_{\text{Viscous terms}}+\underbrace{F_x}_{\text{Body force terms}}$$

$$\rho\left(\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=-\frac{\partial p}{\partial y}+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right)+F_y$$

$$\rho\left(\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=-\frac{\partial p}{\partial z}+\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right)+F_z$$

The energy equation

$$\rho c_p\left(\frac{\partial T}{\partial t}+u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}+w\frac{\partial T}{\partial z}\right)=\Phi+\frac{\partial}{\partial x}\left[k\frac{\partial T}{\partial x}\right]+\frac{\partial}{\partial y}\left[k\frac{\partial T}{\partial y}\right]+\frac{\partial}{\partial z}\left[k\frac{\partial T}{\partial z}\right]+\left(u\frac{\partial p}{\partial x}+v\frac{\partial p}{\partial y}+w\frac{\partial p}{\partial z}\right)$$

where Φ is the dissipation function given by:

$$\Phi = 2\mu\left[\left(\frac{\partial u}{\partial x}\right)^2 + \left(\frac{\partial v}{\partial y}\right)^2 + \left(\frac{\partial w}{\partial z}\right)^2 + 0.5\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right)^2 + 0.5\left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y}\right)^2 + 0.5\left(\frac{\partial w}{\partial x} + \frac{\partial u}{\partial z}\right)^2\right] - \frac{2}{3}\mu\left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z}\right)^2$$

In these equations, u,v,w are the velocity components in the x,y,z directions, ρ is the density, T is the temperature, p is the pressure, μ is the viscosity and c_p is the specific heat at constant pressure. The continuity equation applies to all fluids, compressible and incompressible flow. Newtonian and non-Newtonian fluids. It expresses the law of conservation of mass at each point in a fluid and must therefore be satisfied at every point in flow field.

2.4 Turbulence modeling.

Turbulence modeling constitutes a large, if not the greatest, part of CFD. The following section briefly discusses the background of the models and continues to demonstrate their versatility as well as shortcomings [17]. Factors that need to be considered when evaluating the effectiveness of different turbulence models are accuracy, stability, solution time, flexibility and resource intensiveness. The literature also gives other insights into selection considerations, namely application suitability, flow pattern/geometry, boundary values and performance. The following modeling approach are being considered:

i. Reynolds-averaged Navier-Stokes (RANS) models
ii. Large-eddy simulation (LES) models
iii. Detached-eddy simulation (DES) models
iv. Direct numerical simulation (DNS) models.

2.4.1 Boundary conditions.

The following conditions are considered:

1. The working fluid of the shell side is water,
2. The shell inlet temperature is set to 300 K,
3. The constant wall temperature of 450 K is assigned to the tube walls,
4. Zero gauge pressure is assigned to the outlet nozzle,
5. The inlet velocity profile is assumed to be uniform,
6. No slip condition is assigned to all surfaces.

3. Result and Discussion

3.1 Validation.

Given in table-3.1 below are the simulation results obtained for different mass flow rates and baffle cut sizes of shell side fluid domain ranging from 0.5kg/s, 1kg/s, 2kg/s and 25%, 35%, 45% respectively:

Baffle (%)	Baffle Diameter (mm)	Mass Flow Rate = 0.5kg/s		Mass Flow rate = 1kg/s		Mass Flow Rate =2kg/s	
		Outlet Temperature [K]	Pressure Difference [Pa]	Outlet Temperature [K]	Pressure Difference [Pa]	Outlet Temperature [K]	Pressure Difference [Pa]
25	67.5	343.5	4687.7	340.5	21644.5	337.5	90,880
35	58.5	342.5	3860.4	336.5	15292.9	333.5	65,651
45	49.5	329.5	3205.9	327.5	12780.0	324.5	56,440

Table - 3.1 Outlet temperature and shell-side pressure drop values

It could be seen from fig.-1 that the temperature gradually increases from 300 k (approximately) at the inlet to 343.5 k (approximately) at the outlet of the shell side. The average temperature at the outlet surface is nearly 338.5 k for this model. The pressure drop is less for 0.5kg/s mass flow rate compared to other two mass flow rates. From the contour of pressure in fig.-2, it was found that recirculation near the baffles induces turbulence eddies which would result in more pressure drop for this model.

From the result in table-3.1, it was found that the shell outlet temperature decreases with increase in mass flow rates. Also the increment in percentage baffle cut sizes leads to decrease in outlet temperature at various mass flow rates as well as pressure drop. For this particular analysis the mass flow rate must be above 2 kg/s for a steady fluid flow, however, if it is decreased below 0.5 kg/s the pressure drops rapidly.

For streamline of temperature for 25% baffle cut size (0.5kg/s mass flow rate)

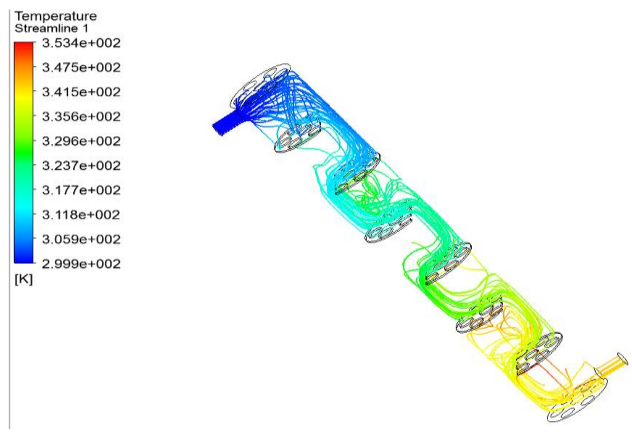

Fig. – 1 Streamline of Temperature

Pressure contours for 25% baffle cut size (0.5kg/s mass flow rate)

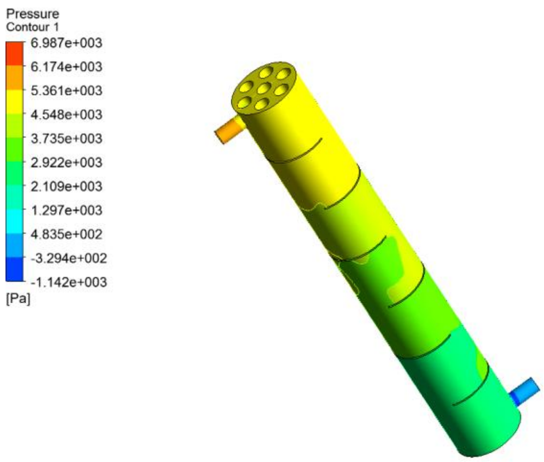

Fig. – 2 Contours of pressure

Moreover, considering figs.-1 and -2, the 25% baffle cut size (67.5 mm in diameter of shell), the fluid turbulence is more compare to other baffle cut size considered.

For streamline of pressure for 35% baffle cut size (0.5kg/s mass flow rate)

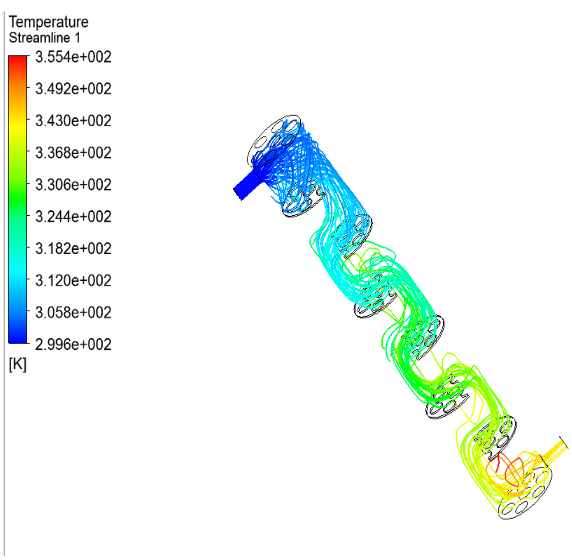

Fig. – 3 Streamline of Pressure

Contours of pressure for 35% baffle cut size (0.5kg/s mass flow rate)

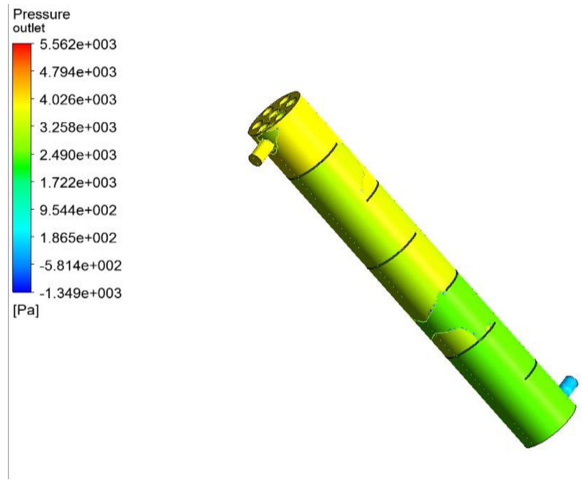

Fig. – 4 Contours of pressure

Also, considering figs. 3 and -4, 35% baffle cut size (58.5 mm in diameter of shell), the fluid turbulence is less compare to that considered in figs.-1 and -2.

For streamline of temperature for 45% baffle cut size (0.5kg/s mass flow rate)

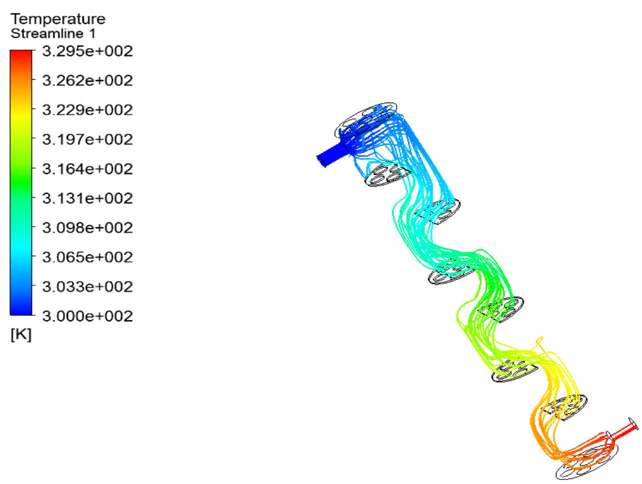

Fig. – 5 Streamline of Temperature

For contours of pressure for 45% baffle cut size (0.5kg/s mass flow rate)

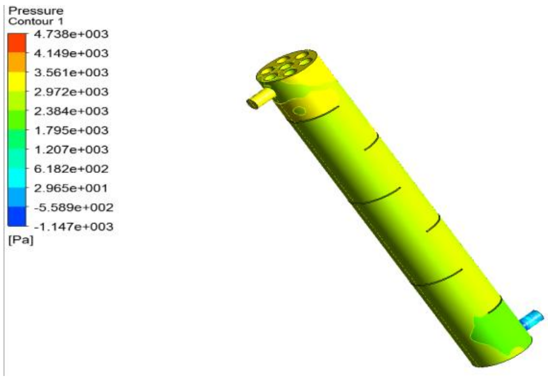

Fig. – 6 Contours of Pressure

And finally, considering figs. -5 and -6, the 45% baffle cut size (49.5 mm in diameter of shell), the fluid is more of steady state compare to other baffle cut size considered.

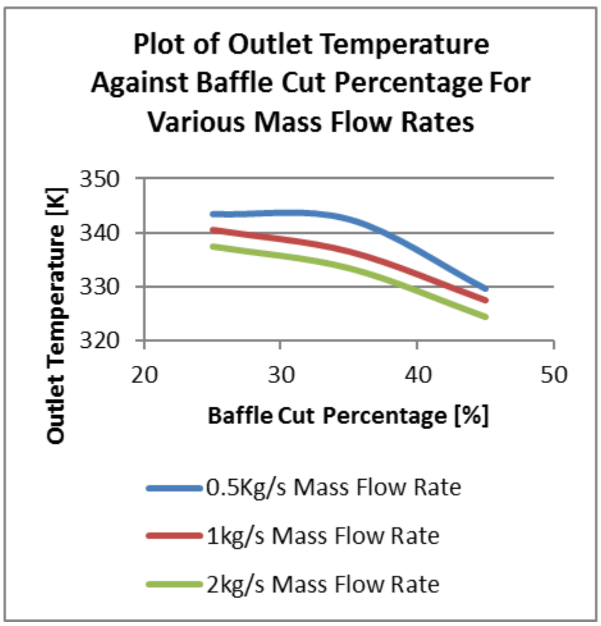

Fig. – 7 Graph of Outlet Temperature against Baffle Cut % for Various Mass Flow Rates.

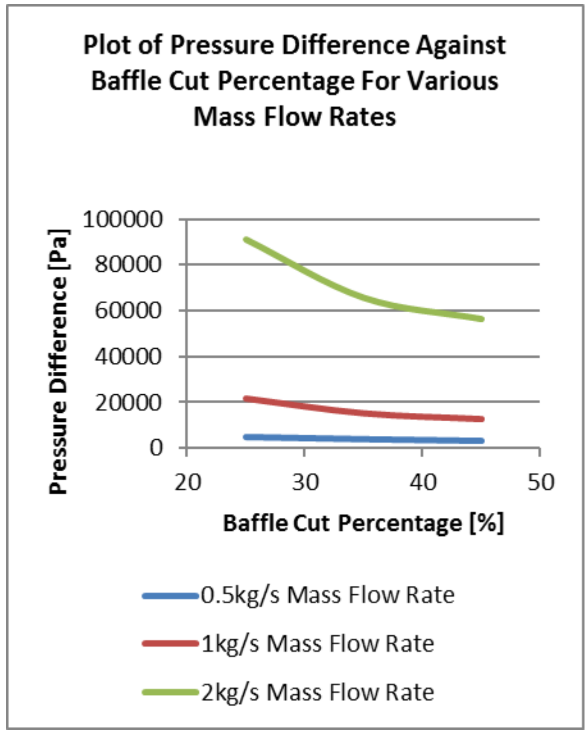

Fig. – 8 Graph of Pressure Differences against Baffle Cut % for Various Mass Flow Rates.

Figs. -7 and -8, showed the graph obtained from the simulation at various mass flow rates and baffle cut sizes.

This section have been able to highlight the results obtained from the CFD simulation by depicting them in fig.-1 to fig.-6 using post-processing sub-application ANSYS CFD-POST. Figs.- 7 and 8 are the EXCEL plots in order to show the variations in the outlet temperature of the baffle cut percentages at the three considered mass flow rates.

4. CONCLUSION

The variation in the baffle diameter of shell and tube heat exchanger has been found to have a direct effect on the performance of the heat exchanger in terms of the outlet fluid temperature and pressure drop. This study has shown that increasing baffle cut percentage (i.e. reducing baffle diameters) demonstrates significant improvement on the flow conditions and overall cooling performance of the heat exchanger. Hence, further study in this field is likely to reveal more interesting results in heat exchanger design and operations.

REFERENCES

[1] B.T.L. Alawa and V. Egwanwo, "Numerical Analysis of Heat Transfer in Heat Exchanger", *International Journal of Applied Science and Technology*, vol. 2, no. 4, pp. 60 – 64, 2012.

[2] Clarke R.H., (1990) "Computational Fluid Dynamics: The Basic with Applications" *Mc Graw- Hill Book Company Inc., New York.* 1990.

[3] A. Singh and S.S. Sehgal, "Thermohydraulic Analysis of Shell-and-Tube Heat Exchanger with Segmental Baffles", *ISRN Chemical Engineering*, vol. 2013, Article ID 548676, 5 Pages, 2013.

[4] S.T. Gajusingh, N. Shaikh, and K. Siddiqui, "Influence of a Rectangular Baffle on the Downstream Flow Structure", *Experimental Thermal and Fluid Science*, vol. 34, no. 5, pp. 590 – 602, 2010.

[5] V.K. Chalwa and N. Kadli, "Study of Variation for Pressure Drop and Temperature Distribution in a Shell and Tube Heat Exchanger in Case of Vertical Baffle", *Mechanical Confab*, vol. 2, no. 1, pp. 17 – 25, 2013.

[6] S.S. Shinde, S.S. Joshi and S. Pavithran, "Performance Improvement in Single phase Tubular Heat Exchanger using continuous Helical Baffles", *International Journal of Engineering Research and Applications (IJERA)*, Vol. No.2, pp. 1141-1149, Jan 2012.

[7] M. Vishwakarma, K.K. Jain, "Performance Evaluation of a Helical Baffle Heat Exchanger", *International Journal of Engineering Research and Technology, IJERT*, vol. 2, Issue (6), 2013.

[8] R. Thundil, K. Raj, and S. Ganne, "Shell Side Numerical Analysis of a Shell and Tube Heat Exchanger Considering the Effects of Baffle Inclination Angle on Fluid Flow", *International Journal of Thermal Science*, vol.16, no. 4, pp. 1165 – 1174, 2012

[9] S.P. Walde, "Review of Heat Transfer Enhancement in Different Types of Baffles and Their Orientations", *International Journal of Engineering and Technology*, vol. 4, no. 4, pp. 1367 – 1372, 2012

[10] Achenbach E. (1971), "On the Cross Flow through In-Line Tube Banks with Regard to the

Effect of Surface Roughness", Wärme- und Stoffübertragung, Vol. 4, pp. 152-155, 1971.

[11] Ball, J., "Construction Basic of Shell and Tube Heat Exchanger, *API Heat Transfer*, March 2000.

[12] M.M.A. Bhutta, N. Hayat, M.H. Bashir, A.R. Khan, K.N. Ahmad, "CFD Applications in Various Heat Exchangers Design: A Review, Department Of Mechanical Engineering, University Of Engineering & Technology", *Applied Thermal Engineering*, 2011.

[13] E.S. Gaddis, "Pressure drop on the shell side of shell and tube heat exchangers with segmental baffles". Chemical Engineering and Processing, Vol. 36, No. 2, pp. 149-159, 2007

[14] Fernandes et al., "In Turbulence, Classic Papers on Statistical Theory" *Inter Science, New York*, 2007.