Thermo-Hydraulic Performance Evaluation of Shell & Tube Heat Exchanger with Different Tube Geometries

Nikhil M Gadave, Pramod P Kothmire

***Abstract** A heat exchanger is equipment that transfers heat energy from one fluid stream to another fluid stream across a solid surface by conduction and convection. Heat exchangers are used in air conditioning & refrigeration systems, power plants, automotive industries, chemical processing, waste heat recovery systems, and food industries. Shell & tube heat exchangers are the most widely used heat exchanger. Earlier many types of studies were carried out on baffle of heat exchanger, as the hydraulic performance of shell side of exchanger relies on baffle elements such as changing baffle types, baffle segments, baffle angles, baffle cuts, etc. are introduced. But only a few researches are concentrated on the tube side. In this paper, efforts have been made to design a shell & tube heat exchanger by using the kern method & referring TEMA standards. Also, the fluid flow behavior & heat transfer mechanism of shell & tube heat exchanger with four different cross-sections of the tubes i.e. Circular, Rectangular, Square & Triangular is numerically investigated using ANSYS-fluent. Numerical simulation was carried out for a single tube pass shell & tube heat exchanger with 25% baffle cut. Finally, from the simulation results, suggestions are made for the best geometry which gives the best thermo-hydraulic performance.*

Keywords: ANSYS-Fluent, CFD, Shell & tube heat exchanger

I. INTRODUCTION

Shell & tube heat exchangers are most widely used in industries such as petroleum refineries, power plants, chemical industries, aerospace industries, cooling towers, etc. They can offer versatile applications for liquid as well as gaseous medium in large pressure & temperature ranges. Shell & tube heat exchanger consist of a bundle of tubes enclosed within a cylindrical shell. Shell is usually cylindrical in the shape with a circular cross-section, although in specific applications shells of different shape are used. One fluid flow through the tubes and another fluid flows through the gap between the tubes and the shell. The baffle plays a vital role in STHX, which supports the tube bundle and also ensures uniform mixing of fluid in the shell side. Shell & tube heat exchangers have more than 40% market share in different industries. Therefore, it is necessary to concentrate on this device, to enhance the efficiency of these exchangers. Baffles and tube configuration and their arrangement have a deep impact on the performance of these type of heat exchanger.

Revised Manuscript Received on October 15, 2019.

Nikhil M Gadave, M.tech Student, School of Mechanical and Civil Engineering, MIT Academy of Engineering Alandi(D), Savitribai Phule Pune University, Pune, India, 412105. Email: nmgadave@mitaoe.ac.in

Pramod P Kothmire, Assistant Professor, School of Mechanical and Civil Engineering, MIT Academy of Engineering Alandi(D), Savitribai pune University, Pune, India, 412105. Email: ppkothmire@mech.maepune.ac.in

However, to reduce the power consumption and increase cost-benefit, conventional heat exchangers with circular tubes are usually not competent enough due to they got low and inefficient thermal performance. To overcome this problem, different strategies of heat transfer enhancement were put forward. Besides, in the past few years, more focus has been given to cross-flow heat exchangers with non-circular tube configuration. The primary objective of this research work is to obtain insight into the heat transfer and fluid flow structure of STHX using different tube geometries: circular, rectangular, square and triangular.

II. LITERATURE REVIEW

Ngoctan Tran, Chi-Chuan Wang [1] in their paper, numerically investigated, impact of tube geometries on the heat transfer mechanism and fluid flow structure of the shell-and-tube heat exchangers having circular, rectangular, equilateral- and isosceles-triangular, square and elliptical shapes, found that an elliptical tube's thermal performance is better compared to circular tubes. A locally optimized triangular tube is described with the highest effectiveness of 92.46%. The Nusselt numbers subjected to tube varieties examined are described in a novel correlation. In addition to current round tube correlations, the suggested comparison also provides excellent predictions for other tube shapes with a peak deviation of less than 7.1 %. Mohammad Reza Saffarian et al. [2] studied different cross-sections of tubes i.e. circular, elliptical with two different attack angles of 90^0 and 0^0 in their paper. They found that the maximum rate of heat transfer was achieved with ellipsoidal tubes located close to the shell having angle of attack of 90^0 and circular tubes situated in the middle of the shell than with circular tubes and elliptical tubes with an attack angle of 90^0 and 0^0. Also, tubes placed close the shell has been shown to have a higher effect on heat transfer than tubes in the middle of the shell.

With regard to tube geometry, circular tubes were usually used in the beginning because they are easy to manufacture. This shape, however, creates significant segregation, large wakes and consequently increases pressure drop. Hence, in recent years, the use of non-circular tubes in heat exchangers has gained attention. Najla El Gharbi et al. [3] obtained an overview of heat transfer & fluid flow characteristics in different tube bundles layouts: circular, ellipsoidal and wing-shaped, considering bulk and local entropy generation.

They conducted a numerical study on two dimensional CFD model with the use of finite volume discretization method to assess the performance of these models. For various conditions of tube shapes, heat transfer correlations are obtained & the best geometry suggestions are made that minimize the total entropy generation. The results showed that the optimum Reynolds number value exists if the tube geometry is fixed & also there is no best tube geometry shape that is valid for all flow conditions. Also concluded that circular shape has worst performance than the other two geometries when $Re > 1.5 * 10^4$ the elliptic shape is the best for $Re > 2.3 * 10^4$, although with little advantage as compared to the wing-shaped tubes.

Ala Hasan [4] experimentally studied the thermo-hydraulic performance of five oval tubes & comparison is done with a circular tube in a cross-flow of air. The nominal axis ratio of three of the tested oval tubes are 2, 3 & 4. Other two configurations of oval tubes are, an oval tube with axis ratio of 3 having two wires attached to it's lower & upper top positions & a cut-oval shaped tube. The drag C_d coefficient for the tubes is evaluated and the combined thermo-hydraulic performance of the oval tubes is defined by the Nu_D/C_d ratio. A. Nouri-Borujerdi & A.M. Lavasani [5] experimentally analyzed cam shaped tube in cross-flow with forced convection & investigated heat transfer mechanism and fluid flow structure of this shape. Findings indicated that over the whole range of the Reynolds numbers, the average heat transfer coefficient is a maximum at about $\alpha = 900$. It is found that, except for $\alpha = 900$ and 1200, the performance of cam-shaped tube is superior to that of a circular tube having the same surface area. Ali Akbar Abbasian Arania & Reza Moradia [6] performed experimental & numerical analysis of heat transfer & fluid flow behavior of shell and tube heat exchanger with segmental baffle (SB-STHE) & optimized using a combination of baffle and longitudinal ribbed tube configuration. circular and triangular ribbed tubes are used with a combination of disk baffle shell and tube heat exchanger (DB-STHE) and combined segmental disk baffle shell and tube heat exchanger (CSDB-STHE). From the experiment, it is concluded that the average heat transfer coefficient of shell side of disk baffle with triangular rib (DB-TR) & combined segmental disk baffle with triangular rib (CSDB-TR) are 31.9% and 26.6% higher than (CSDB-CR) and (DB-CR) respectively. Another performance evaluation criterion $Q/\Delta P$ is selected for same mass flow rate, 7.12%, 24.2%, 40.5% & 42.8% difference compared to conventional baffles are given by DB-CR, CSDB-CR, CSDB-TR & DB-TR respectively. Based on the obtained results, disk baffle with a triangular rib gave the best performance among the analyzed combinations. Vijaya Kumar Reddy et al. [7] studied tube in tube helical coiled heat exchanger with different mass flow rates in inner as well as outer tubes. The results showed that, as the flow rate in inner tubes increased from 400 to 600 LPH, with a constant flow rate of 700 LPH in the outer tube, LMTD increased by 1.33% As the flow rate in outer tube increases with constant flow rate in an inner tube, the LMTD decreases.

Pranita Bichkar [8] numerically analyzed shell & tube heat exchanger with different types of baffles & studied their effect on pressure drop. i.e. single segmental, double segmental and helical baffles. Single segmental baffles showed the formation of dead zones. Compared with single segmental baffles, double segmental baffles showed to reduce vibrational hazard. The use of helical baffles indicated a reduction in the pressure drop as the dead zones were eliminated. The comparative analysis showed that helical baffles are more efficient than the other two baffle configurations. Guo-Yan Zhou [9] numerically studied forced convection characteristics of heat transfer of shell & tube heat exchanger having trefoil hole baffles results obtained were compared to experimental results. The study showed that fluid flow was fully developed ahead of the first trefoil-hole baffle. The secondary flow is generated on both sides of the baffles when the fluid flows through the trefoil hole baffle. The secondary flow & jet flow and decreased the boundary layer thickness which enhanced the heat transfer.

Shuai Xie [10] numerically studied fluid flow structure & heat transfer mechanism of the enhanced tube with cross ellipsoidal dimples. Further, the effects of dimples depth, pitch & axis ratio are also investigated. For carrying out the numerical simulations, the realizable k-e turbulence model was used with the Re range from 5000 to 30,000. From this study, it is concluded that dimple tubes showed an increased rate of heat transfer compared to ordinary plain tubes, a beneficiary technique that can be employed for heat transfer augmentation applications.

From literature survey it can be concluded that many kinds of researches have been focused on changing baffle types & tube arrangement but very few studies were carried out to study the effect of tube shapes on heat transfer mechanism & fluid flow structure for shell & tube heat exchangers. However, the supplied fan power may be restricted for the energy recovery applications, and thus the use of tubes or shells of different shapes can be more advantageous as per energy savings is concerned. This recent demand for energy saving has motivated the current research to study the impact of tube shape on heat transfer & fluid flow structure applicable to shell & tube heat exchanger. In this study, four different tube shapes are studied & decision is made for optimum geometry which gives better thermal & pressure drop performance.

III. METHODOLOGY

A. *Physical Models*

All the physical models used are presented in fig. below. In the current research, shell & tube heat exchanger having a single tube pass & 25% baffle cut is used for the analysis. The specification of the used model is given in table 1. The hot fluid which at 65^0C flows inside the tubes & cold fluid which is at 25^0C flows inside the shell. The inner diameter of shell is 192 mm outside diameter is 200 mm in all cases.

The dimensions of different tube shapes are mentioned in table I. Stainless steel is used as a shell material & copper material is used for tubes.

All tubes used in the present research have the same outer surface area & are arranged in 45^0 triangular layout. 21 number of tubes are there in each model. Baffles are made of stainless steel. Baffle spacing & baffle thickness is 125 mm & 5 mm respectively. Working fluid is chosen as water for both the shell side & tube side since it is used in the current manufacturing process in steel industries. The design of STHX is done by using Kern Method. The geometry of STHX is prepared in three dimensions using CATIA (ver.5) software. The numerical simulations are carried out on CFD software package ANSYS-Fluent (Ver. 18.2) & based on simulation results; the best geometry is selected. Simulations are performed for each tube geometries with different mass flow rates. Several empirical correlations have been established with the help of the kern method for the calculation of the shell-side coefficient of heat transfer and pressure drop. In this study, CFD simulations results are compared with analytical results obtained from the kern method for each model with different mass flow rates.

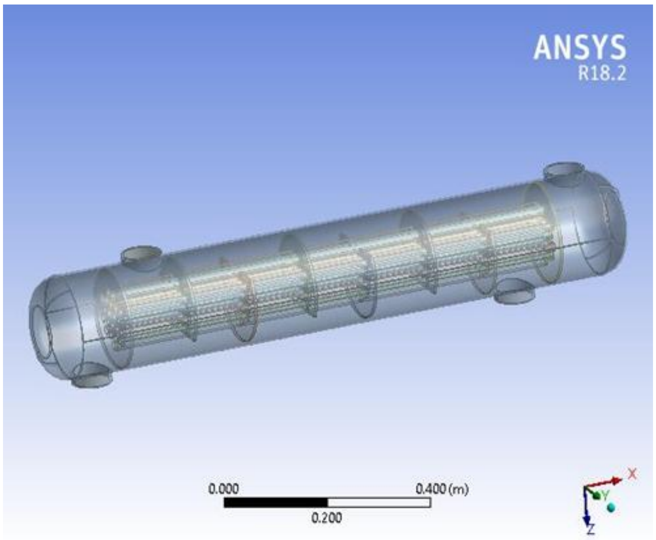

Fig. 1. Shell & tube heat exchanger geometry

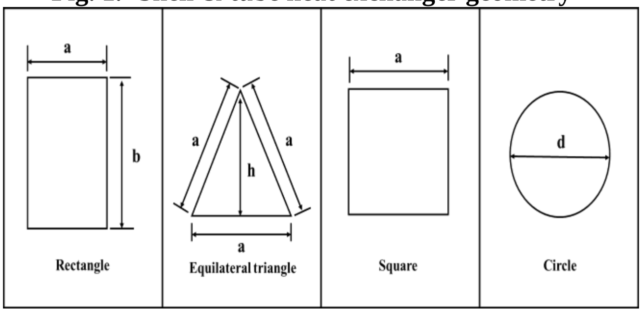

Fig. 2. Different tube geometries used for the analysis

Dimensions of tube geometries are calculated such that the outer surface area of all the models will remain the same. In all the heat exchangers examined in this study, all factors such as the governing equations, type of flow, baffles, and type of fluid were unchanged and only the geometries of tubes were changed.

Table- I: Dimensions of different tube geometries

Names of tube shape	Dimensions
Circular	d_0= 19.05 mm, d_i= 15.75 mm
Rectangular	a= 12 mm, b= 18 mm
Square	a= 15 mm
Triangular	a= 20 mm, h= 17.32 mm

Table- II: Geometric specification of the heat exchanger

Shell inside diameter (D_s)	0.192 m
Shell outside diameter	0.2 m
Tube inside diameter (D_i)	0.01575 m
Tube outside diameter (D_o)	0.01924 m
Pitch (P_t)	0.024 m
Length of shell (L_s)	1.2 m
Length of tube (L_t)	0.9 m
Number of baffles (N_b)	6
Baffle spacing	0.125 m
Number of tubes (N_t)	21
Number of shell passes (n_s)	1
Number of tube passes (n_t)	1
Bundle to shell diametrical clearance (Δb)	0.031 m
Shell to baffle diametrical clearance (Δsb)	0.004 m
Total heat transfer area (A)	1.13 m^2

B. Governing Equations

Water is used as both shell & tube side fluid which is treated as an incompressible fluid. Flow is assumed to be turbulent. No-slip condition is assumed for the walls. Based on these assumptions, governing equations of continuity, momentum & energy for incompressible fluids are represented as follows:

Continuity Equation:

$$\frac{\partial(\rho_f u)}{\partial x} + \frac{\partial(\rho_f v)}{\partial y} + \frac{\partial(\rho_f w)}{\partial z} = 0 \qquad (1)$$

Momentum Equation:

X momentum:

$$u\frac{\partial(\rho_f u)}{\partial x} + v\frac{\partial(\rho_f u)}{\partial y} + w\frac{\partial(\rho_f u)}{\partial z} = \frac{\partial\left(\mu_f \frac{du}{dx}\right)}{\partial x} + \frac{\partial\left(\mu_f \frac{du}{dy}\right)}{\partial y} + \frac{\partial\left(\mu_f \frac{du}{dz}\right)}{\partial z} - \frac{\partial p}{\partial x} \qquad (2)$$

Y momentum:

$$u\frac{\partial(\rho_f v)}{\partial x} + v\frac{\partial(\rho_f v)}{\partial y} + w\frac{\partial(\rho_f v)}{\partial z} = \frac{\partial\left(\mu_f \frac{dv}{dx}\right)}{\partial x} + \frac{\partial\left(\mu_f \frac{dv}{dy}\right)}{\partial y} + \frac{\partial\left(\mu_f \frac{dv}{dz}\right)}{\partial z} - \frac{\partial p}{\partial y} \qquad (3)$$

Z momentum:

$$u\frac{\partial(\rho_f w)}{\partial x} + v\frac{\partial(\rho_f w)}{\partial y} + w\frac{\partial(\rho_f w)}{\partial z} = \frac{\partial\left(\mu_f \frac{dw}{dx}\right)}{\partial x} + \frac{\partial\left(\mu_f \frac{dw}{dy}\right)}{\partial y} + \frac{\partial\left(\mu_f \frac{dw}{dz}\right)}{\partial z} - \frac{\partial p}{\partial z} \qquad (4)$$

Energy Equation for solid domain:

$$K_s\left(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}+\frac{\partial^2 T}{\partial z^2}\right) \quad (5)$$

Energy Equation for fluid domain:

$$u\frac{\partial(\rho_f C_f T_f)}{\partial x}+v\frac{\partial(\rho_f C_f T_f)}{\partial x}+w\frac{\partial(\rho_f C_f T_f)}{\partial x}=k_f\frac{\partial^2 T_f}{\partial x^2}+k_f\frac{\partial^2 T_f}{\partial y^2}+k_f\frac{\partial^2 T_f}{\partial z^2} \quad (6)$$

Considering the high swirl on the shell side fluid flow with non-isotropic turbulence, Realizable k − ε turbulence model along with standard wall function is employed for the numerical simulations. The conservation equations of turbulence kinetic energy and its dissipation rate for realizable k-ε turbulence model are represented as follows:

Turbulence kinetic Energy k:

$$\frac{\partial(\rho_f k)}{\delta t}+\frac{\partial(\rho_f k u_j)}{\delta x_j}=\frac{\partial\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]}{\delta x_j}+G_k+G_b-\rho\epsilon-Y_M+S_k \quad (7)$$

Turbulence kinetic energy dissipation rate ε:

$$\frac{\partial(\rho_f \varepsilon)}{\delta t}+\frac{\partial(\rho_f \varepsilon u_j)}{\delta x_j}=\frac{\partial\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right]}{\delta x_j}+\rho C_1 S_\epsilon-\rho C_2\frac{\epsilon^2}{k+\sqrt{v\epsilon}}+C_{1\epsilon}\frac{\epsilon}{k}C_{3\epsilon}G_b+S_\epsilon \quad (8)$$

Where,

$$C_1=max\left[0.43,\frac{\eta}{\eta+5}\right] \quad \eta=S\frac{k}{\epsilon} \quad S=\sqrt{2S_{ij}S_{ij}}$$

The turbulent viscosity is given by: -

$$\mu_{t=}\rho C_\mu\frac{k^2}{\epsilon}$$

& model constants are,

$$C_{1\epsilon}=1.44,\ C_2=1.9, \sigma_k=1, \sigma_s=1.2$$

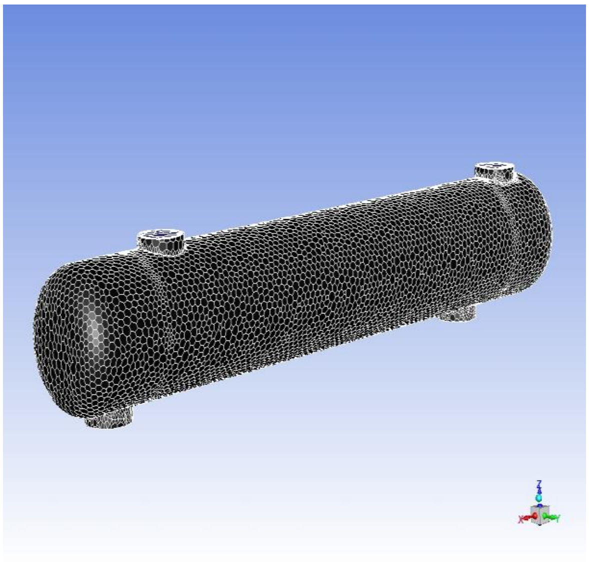

Fig. 3. Meshed Geometry

C. The Computational Domain & Boundary Conditions

To perform the simulations, a three-dimensional discretized grid system is generated by using commercial software package ANSYS (Ver. 18.2). The discretization has been done with unstructured polyhedral elements. The total cell count of four different geometries i.e. circular, rectangular, square, & triangular is 5.3 million, 5.3 million, 5.5 million, & 4.9 million respectively. The grid independency test has been carried out by adopting four different mesh sizes of 3.5 million, 4 million, 5 million & 6 million to maintain accuracy of calculation & it is observed that the pressure loss difference between the last two mesh models is less than 4%. Thus, the range of mesh size employed for the simulation is in between 5 to 5.5 million considering the computational time & solution accuracy.

For the present study, the following boundary conditions are set:

1. The momentum boundary condition of no penetration & no slip is applied for all the solid walls.
2. The shell wall is set with a thermal boundary condition of zero heat flux.
3. The coupling thermal boundary condition is used for the tube walls.
4. The temperature of the hot fluid at the inlet is kept constant at 338 K.
5. The temperature of the cold fluid at the inlet is kept constant at 298 K.
6. The shell & tube side inlets are set as mass flow inlet.
7. The outlets are set as pressure outlets so the inlet pressure is the pressure drop.
8. The heat generated internally within the model is neglected.

The Finite volume method is employed for discretization of governing equations & these equations are solved by using SIMPLE algorithm with pressure-velocity coupling. All the variables are treated with first-order upwind scheme, expect the energy with second-order upwind scheme. Numerical simulations are performed with the pressure-based solver. The convergence criterion was set as a relative residual of 10^{-3} for the flow equations & 10^{-6} for the energy equation.

D. Thermo-physical Properties of Water at Atmospheric Pressure

Table III: - Thermo-physical properties of shell & tube side water at 298K & 338K

Properties	Water at 298K	Water at 338K	Units
Density (ρ)	997.1	980.6	(kg/m^3)
Specific heat (C_p)	4183	4184	(J/kgK)
Dynamic viscosity (μ)	0.0008905	0.0004334	(Pa.s)
Thermal conductivity (K)	0.5948	0.6455	(W/mK)

IV. RESULT AND DISCUSSION

A. *Validation Results*

Mass flow rate (m/s)	CFD Results		Kern method results		Difference	
	Q(w)	ΔP(Pa)	Q(w)	ΔP(Pa)	Q(w) Diff.	ΔP(Pa) Diff.
3	81812	1545	92236	1398	11.30%	-10.51%
4	101274	2655	119875	2479	15.51%	-7.1%
5	104950	3881	130278	3425	19.44%	-13.31%

The study was carried out taking five different mass flow rates and results obtained from CFD simulations are compared with kern method results. From the above table, it is concluded that there is good agreement between analytical & CFD results.

The graph of tube side heat transfer coefficient & heat transfer vs mass flow rate of four studied heat exchanger is presented in fig. 4 & 5.

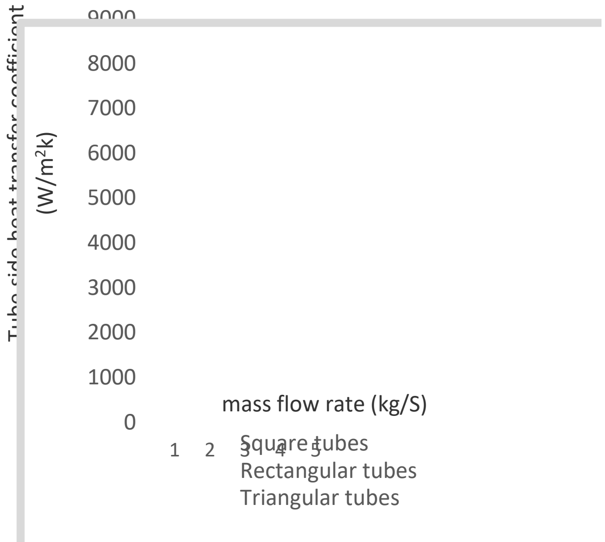

Fig. 4. Tube side heat transfer coefficient vs mass flow rate of four studied tube configurations.

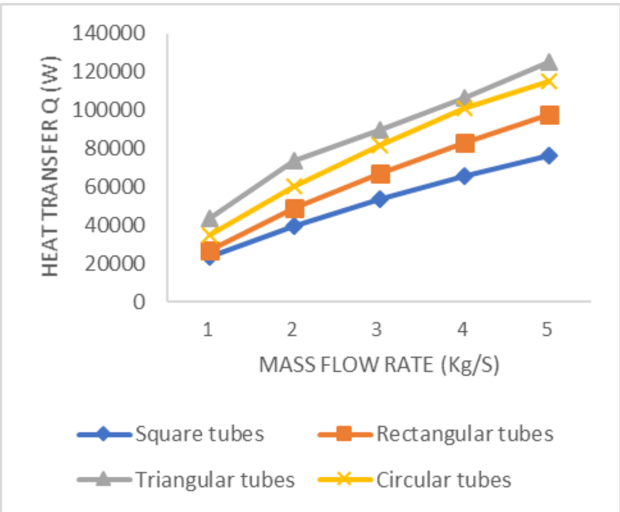

Fig. 5. Heat transfer vs mass flow rate for four studied types of tube geometries.

From the graph (fig. 4.) it is observed that triangular tubes have a higher tube side heat transfer coefficient. There are slight variations in tube side heat transfer coefficients of circular & square tubes. Rectangular tubes have a lower tube side heat transfer coefficient than the other geometries studied. The average value of the tube side heat transfer coefficient of circular and triangular tubes is 4744.43 W/m2k and 6015.88 W/m2k. This shows that there is a 26.79% increase in heat transfer coefficient compared to circular tubes.

From the above plot (fig. 5.) it is seen that; triangular tubes give better heat transfer compared to other tube shapes analyzed. The average value of heat transfer rate of shell & tube heat exchanger having circular, rectangular, square and triangular tubes is 78580, 64634, 51710 and 87744 W. This shows that triangular tubes have best heat transfer performance. Also, the average value of inlet and outlet temperature difference of circular, rectangular, square and triangular tubes is 4.65 K, 3.98 K, 3.26 K, and 5.23 K respectively. Therefore, it can be seen that the shell & tube heat exchanger having triangular tubes gives the best heat transfer performance.

The graph of tube side pressure drop & effectiveness vs mass flow rate of four studied heat exchanger is represented in fig. 6 & 7.

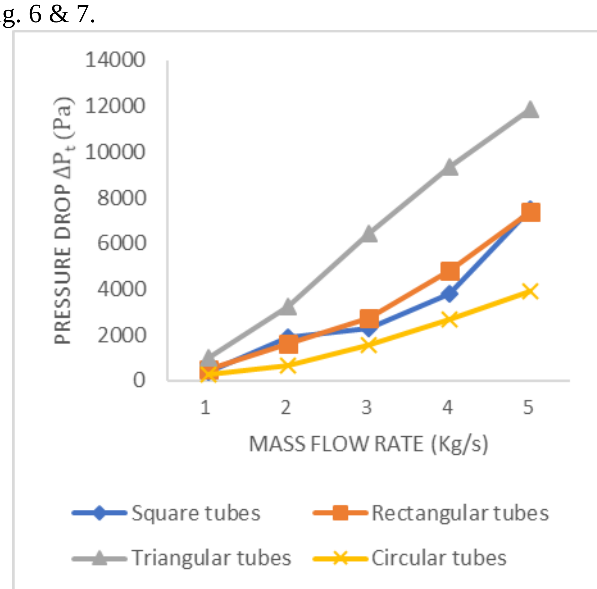

Fig. 6. Tube side pressure drop vs mass flow rate for four different tube cross-sections studied.

The average value of pressure drop for circular, rectangular, square and triangular tubes is 1795.66, 3171.21, 3399.32 and 6377.78 Pa respectively. Hence, it is concluded that triangular tubes are having a higher pressure drop compared to other tube configurations studied.

From the above graph (fig. 6.), it is seen that there is a slight variation in pressure drop values of rectangular and square tubes. While the circular tubes have a lower pressure drop and triangular tubes have a higher pressure drop compared to other tube geometries.

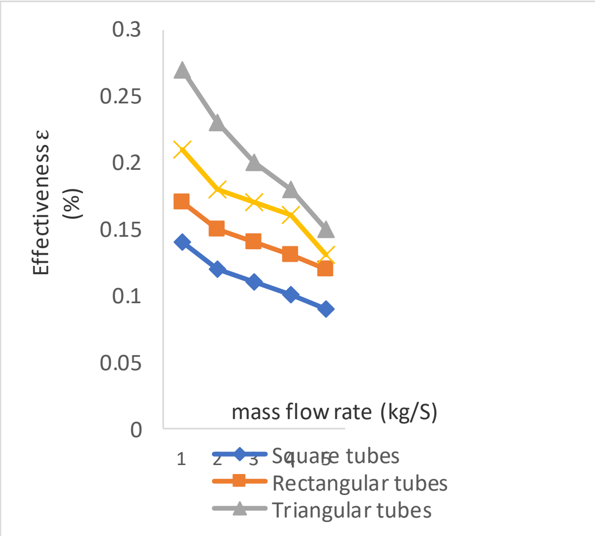

Fig 7. Effectiveness vs mass flow rate of four studied tube shapes.

The efficiency of the heat exchanger depends mainly on two parameters i.e. heat transfer and pressure drop. In the above graph, the efficiency of heat exchanger using different tube geometries plotted vs different mass flow rates. From the graph (fig. 7.), it can be seen that triangular tubes are having maximum effectiveness while square tubes are having the lowest effectiveness. The maximum effectiveness of 0.27 is achieved with triangular tubes at a mass flow rate of 1 kg/s.

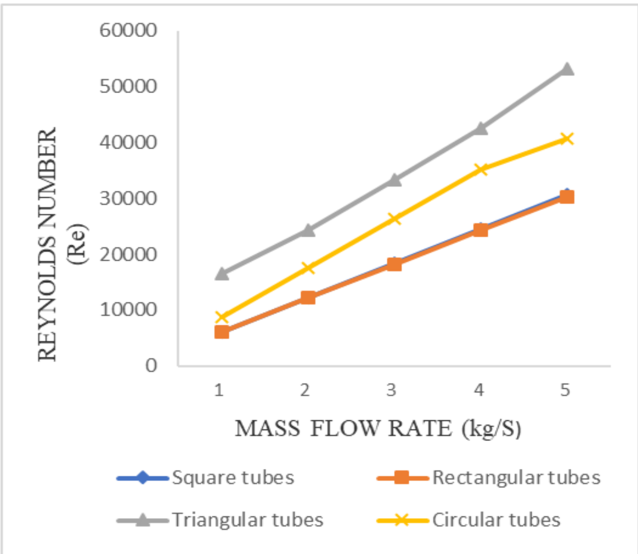

Fig. 8. Tube side Reynolds number vs mass flow rate of four studied tube shapes.

The above graph (fig. 8.) represents the tube side Reynolds number vs mass flow rate for four different types of tube shapes. From the above graph, it is observed that there is a marginal difference between tube side Reynolds number of square and rectangular tubes shell & tube heat exchangers. The triangular tubes have maximum tube side Reynolds number while rectangular and square tubes have minimum tube side Reynolds number.

B. *Temperature Variations*

The temperature variations on the shell surface of four different geometries will give a detailed idea about the heat transfer phenomenon. Below are the temperature contour plots of shells of four different tube geometries.

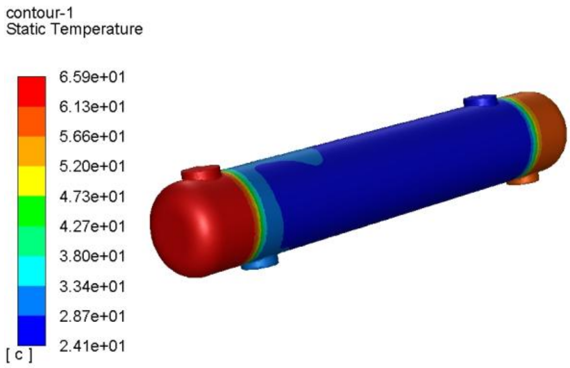

Fig. 9. Temperature contour of shell having circular tubes

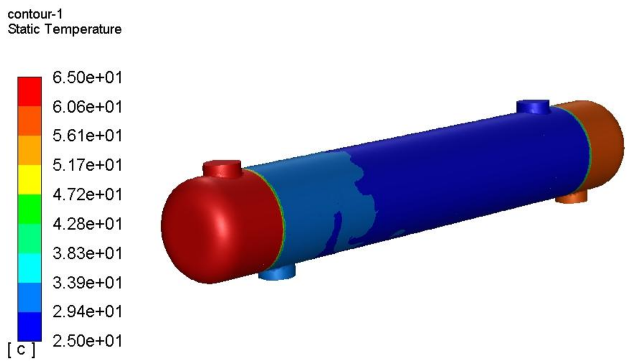

Fig. 10. Temperature contour of shell having rectangular tubes

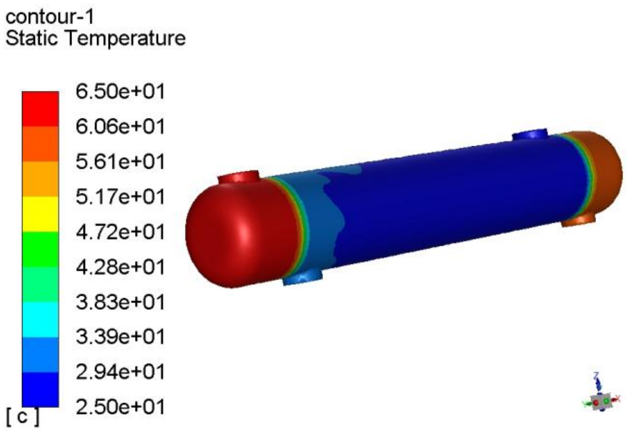

Fig. 11. Temperature contour of shell having square tubes

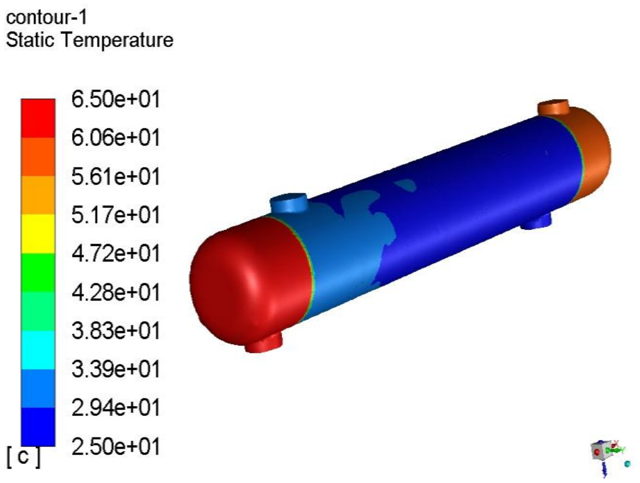

Fig. 12. Temperature contour of shell having triangular tubes

C. *Pressure Variations*

The pressure variations inside the shell of four different geometries will give a detailed idea about the fluid flow structure. Below are pressure vector plots of shells of four different geometries.

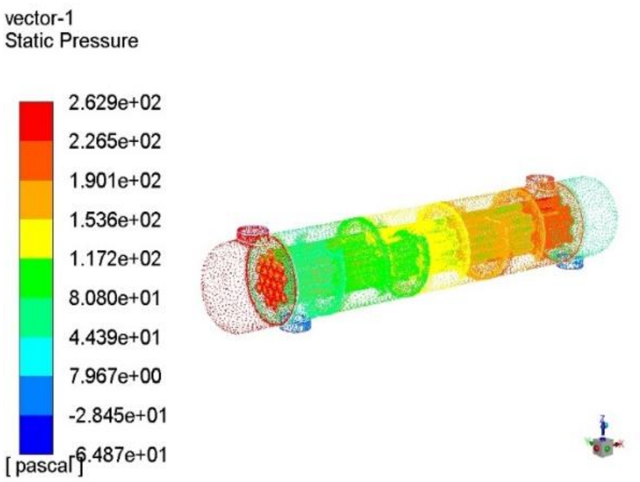

Fig. 13. Pressure vectors inside the shell having circular tubes

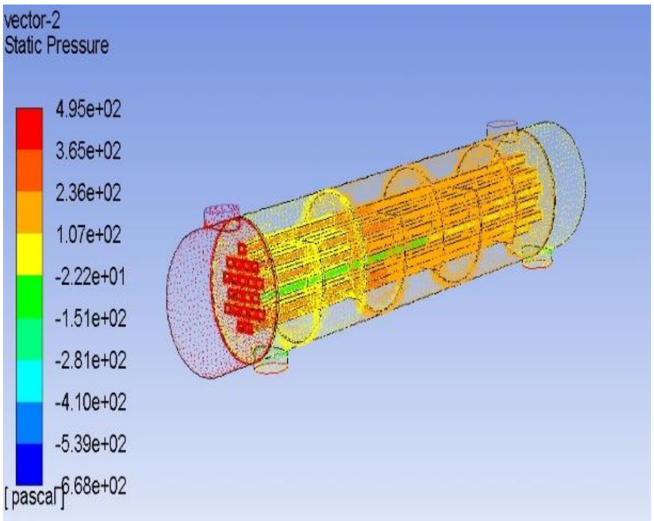

Fig. 14. Pressure vectors inside the shell having rectangular tubes

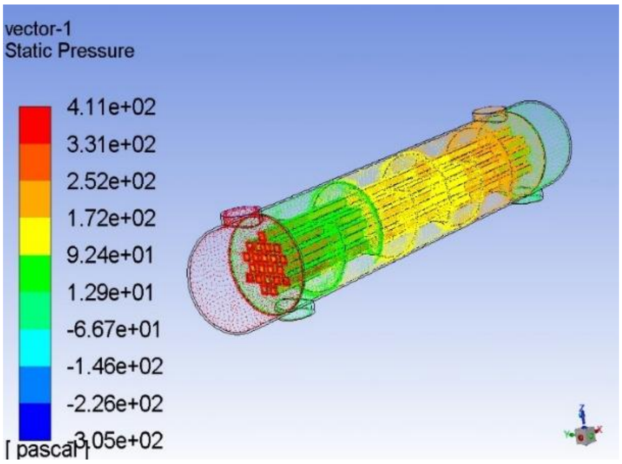

Fig. 15. Pressure vectors inside the shell having square tubes

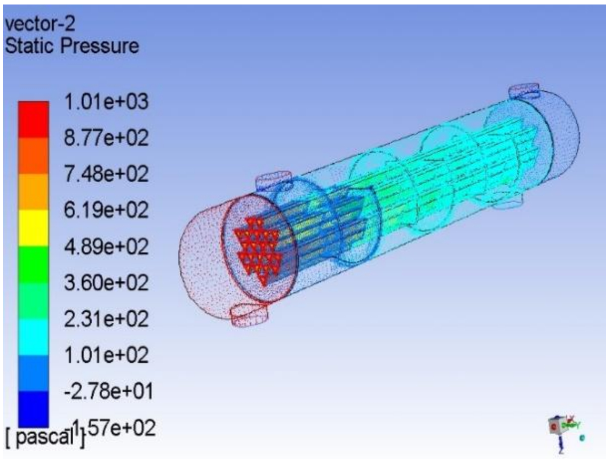

Fig. 16. Pressure vectors inside the shell having triangular tubes

All the above figures represented are taken when the mass flow rate is 1 kg/s for both the shell & tube side inlets.

V. CONCLUSION

In the present research, CFD software ANSYS-Fluent (ver. 18.2) is used for predicting the thermo-hydraulic performance of shell & tube heat exchanger for different mass flow rates. Based on obtained outcomes, analysis & comparison following conclusions can be drawn:

- The thermal performance of triangular tubes is superior to that of other studied tube geometries & there is an 11.66% increase in heat transfer rate compared to circular tubes. But the pressure drop is also on the higher side.
- The thermal performance of rectangular tubes is superior to that of square tubes whereas the pressure drop is less in the case of square tubes than the rectangular one.
- Circular tubes have the lowest pressure drop & moderate thermal performance than the studied models.
- The highest effectiveness of 0.27 is obtained using triangular tubes & it decreases with an increase in mass flow rate.

- The average value of the tube side heat transfer coefficient of circular and triangular tubes is 4744.43 W/m^2k and 6015.88 W/m^2k. This shows that there is an 26.79% increase in heat transfer coefficient compared to circular tubes.
- Finally, the tube side heat transfer coefficient, rate of heat transfer & effectiveness is highest for triangular shaped tubes. Hence, one should go for triangular tubes while designing a shell & tube heat exchanger.

This research work can be further extended by varying the dimensions of tube geometries keeping the same outer perimeter & boundary conditions & by using different fluids for accurate prediction of fluid flow & heat transfer characteristics.

REFERENCES

1. Ngoctan Tran, and Chi-Chuan Wang, "Effects of tube shapes on the performance of recuperative and regenerative heat exchangers", Energy, 169, pp. 1-17, 2019.
2. Mohammad Reza Saffarian, Farivar Fazelpour, and Mehrzad Sham, "Numerical study of shell and tube heat exchanger with different cross-section tubes and combined tubes", International Journal of Energy and Environmental Engineering, 46, pp.10:33, 2019.
3. Najla El Gharbi, Abdelhamid Kheiri, Momammed El Ganaoui and Ryan Blanchard, "Numerical optimization of heat exchangers with circular and non-circular shapes", Case Studies in Thermal Engineering, 2015.
4. Ala Hasan, "Thermal-hydraulic performance of oval tubes in a cross-flow of air", Heat Mass Transfer, 41, pp. 724-733, 2005.
a. Nouri-Borujerdi, and A.M. Lavasani, "Experimental study of forced convec tion heat transfer from a cam shaped tube in cross flows.", International Journal of Heat and Mass Transfer, 50, pp. 2605-11, 2005.
5. Ali Akbar Abbasian Arania, and Reza Moradi, "Shell and tube heat exchanger optimization using new baffle and tube configuration", Applied Thermal Engineering, 157,113736, 2019.
6. Vijaya Kumar Reddy K, Sudheer Prem Kumar B, Ravi Gugulothu, Kakaraparthi Anuja, and Viajaya Rao P, "CFD Analysis of a Helically Coiled Tube in Tube Heat Exchanger", Materials Today: Proceedings,4, pp. 2341-49, 2017.
7. Pranita Bichkar, Ojas Dandgaval, Pranita Dalvi, Rhushabh Godase, and Tapobrata Dey, "Study of Shell and Tube Heat Exchanger with the Effect of Types of Baffles", Procedia Manufacturing, 20, pp. 195-200, 2018.
8. Guo-Yan Zhou, Jingmei Xiao, Lingyun Zhu, Juntao Wang, and Shan-Tung Tu, "A numerical study on the shell-side turbulent heat transfer enhancement of shell-and-tube heat exchanger with trefoil hole baffles" Energy Procedia, 75, pp. 3174-79, 2015.
9. Shuai Xie, Zheng Liang, Liang Zhang, and Yulin Wang, "A numerical study on heat transfer enhancement and flow structure in enhanced tube with cross ellipsoidal dimples", International Journal of Heat and Mass Transfer, 125, pp.434-44, 2018.
10. K. Somasekhar, K.N.D. Malleswara Rao, V. Sankararao, Raffi Mohammed, M. Veerendra, and T. Venkateswararao, "A CFD Investigation of Heat Transfer Enhancement of Shell and Tube Heat Exchanger Using Al2o3-Water Nanofluid", Materials Today: Proceedings, 5, pp. 1057-62, 2018.
11. 12.J. Bala Bhaskara Rao, and V. Ramachandra Raju, "Numerical and heat transfer analysis of shell and tube heat exchanger with circular and elliptical tubes", International Journal of Mechanical and Materials Engineering, 11:6, 2016.
12. 13.Kakac, S, Liu H.T and Pramuanjaroenkij. Heat Exchangers: Selection, Rating and Design, CRC Press, Boca Raton, FL, 2012.
13. 14.J.P. Holman. Heat Transfer. Tata Mc Graw Hill international publishers 9th edition, 2002.
14. 15.Donald Q Kern. Process Heat Transfer. Tata Mc Graw Hill international publisher's edition, 1997.
15. 16.Incropera, F.P, Dewitt D.P. Fundamentals of Heat and Mass Transfer. LTC, Guanabara Dois, Rio de Janeiro. 5th Edition, 2001.

AUTHORS PROFILE

Mr. Nikhil Mallikarjun Gadave (nmgadave@gmail.com or nmgadave@mitaoe.ac.in) is a student of Masters of Technology in Heat Power from School of Mechanical and Civil Engineering, MIT Academy of Engineering Alandi(D), Savitribai Phule Pune University, Pune, India, 412105. He has received a Bachelor of Engineering degree in the Department of Mechanical Engineering from Solapur University, Maharashtra in the year 2016. His research interest includes Computational Fluid Dynamics and Heat Transfer, Shell & tube heat exchanger, Fluid Mechanics, Thermal Engineering.

Qualification: Pursuing M. Tech. (Heat Power), MIT Academy of Engineering Alandi(D), Savitribai Phule Pune University, Pune, India, 412105.

Mr. Pramod P Kothmire is an Assistant Professor in the School of Mechanical and Civil Engineering (ppkothmire@mech.maepune.ac.in), MIT Academy of Engineering Alandi (D), Savitribai Phule Pune University, Pune, India, 412105. He is a student of a Ph.D. in the Indian Institute of Technology, Bombay, India Since 2014 which is a joint research program between IITB and MIT Alandi Pune. He has received a Master of Technology in Thermal Engineering in Sardar Patel College of Engineering, Andheri, Mumbai India, 2013. He has received a Bachelor of Engineering in the Department of Mechanical Engineering from Fr. C. Rodrigues Institute of Technology, Vashi, Navi Mumbai, India. 2008. His interdisciplinary research interests cover Electrohydrodynamics, Interfacial Engineering and also HVAC projects. He is passionate to design, develop and improve the performance of Oil and refinery industries products. His forefront area of researches is Fluid Mechanics, Heat transfer, Electrohydrodynamics, Interfacial Engineering, Thermal Engineering. He has 7 years of teaching experience, 2.2 years of industrial working experience and 1.2 years of research work.