Study of Flow and Heat Transfer Analysis in Shell and Tube Heat Exchanger using CFD

Sugin Elankavi R[1], Uma Shankar T[2]

[1,2]UG Scholars, Department of Mechanical Engineering, PERI Institute of Technology, Tamilnadu, India

---

Abstract - *Heat exchangers are used to transfer heat from fluid at high temperature to fluid at lower temperature. Heat exchangers are used in industrial purposes in chemical industries, nuclear power plants, refineries, food processing, etc. Sizing of heat exchangers plays very significant role for cost optimization. Also, efficiency and effectiveness of heat exchangers is an important parameter while selection of industrial heat exchangers. Methods for improvement on heat transfer have been worked upon for many years in order to obtain high efficiency with optimum cost. In this research work, design of shell & tube heat exchanger with single segmented baffles and analyze the flow and temperature field inside the shell using Autodesk Simulation CFD 2015. When comparing the CFD analysis with experimental results, it was well correlation with negligible percentage of error. Thus, the series of baffles results in a significant increase in heat transfer coefficient per unit pressure drop in the heat exchanger.*

Key Words: **Flow, Heat Transfer, Shell & Tube Heat Exchanger, CFD, Baffles.**

1. INTRODUCTION

Shell & Tube heat exchanger is one of the most widely used equipment in process industry like as in Oil refinery unit, milk dairy & also used in large chemical processes. Heat exchangers are used to transfer heat between two process streams. They are used for different applications such as heating, condensation, cooling and boiling or evaporation purpose. They give name according to their different application e.g. Heat exchanger is used for cooling are called as condenser and It used for heating or boiling are known as boiler. The required amount of heat transfer provides an insight about the capital cost and power requirement of heat exchanger. The tube side and shell side fluids are separated by tube sheet. Thin baffles are used for diverting the flow, support the tube for rigidity and obtained higher heat transfer coefficient. Helical baffles give the better performance than the single segmental baffles, but their manufacturing, maintenance and installation cost is high. Computational fluid dynamics is now industrial design tool having many advantages. CFD models of shell and tube heat exchanger is considered here. The flow structure and heat distribution are obtained by modelling the geometry. This software builds a virtual prototype of the system or device before can be apply to real word physics and chemistry to the model and the software will provide with images and data, which predict the performance of that design.

It can be concluded that a lot of work has bee done in the field of Design & analysis of heat exchanger. An experimental analysis is carried out to study the heat transfer phenomenon in conical coil heat exchanger with cone angle 90 degree [1], The experimental investigation of the effect of wedge-shaped tetrahedral vortex generator on a gas liquid finned tube heat exchanger was studied using irreversibility analysis [2], The statistical analysis is used as an invaluable tool for investigation of performance of a shell and tube heat exchanger during fouling[3], The non-uniform liquid flow among the tubes of a shell and tube apparatus has to be taken into account in determining the efficiency of heat transfer. The authors of this paper have proposed a method for taking this non-uniformity into account and for analyzing its effect on the intensity of heat transfer [4], The improvement in the normal and longitudinal flow, we get more efficiency and low resistance heat exchanger[5], The heat transfer rate of the external tube surface of the heat exchanger for a closed wet cooling tower can be divided into sensible and latent heat transfer rates. These in turn are expressed by heat and mass transfer coefficients [6], Minimization of the thermal surface area for a particular service involving discrete variables [7]. Somasekhar et al. [8] found that adding nanoparticles to the base fluid (distilled water) causes the significant enhancement of heat transfer characteristics. Priyanka et al. [9] predicted the performance of the heat exchanger by considering different heat transfer fluid and the result so obtained have been compared. Pranita Bichkar et al. [10] shows the effect of baffles on pressure drop in shell and tube heat exchanger. Santhisree et al. [11] enhanced the heat transfer rate and maintained uniform distance between the tubes by placing baffle in the shell to force the fluid to flow across the shell. Bansi D. Raja et al. [12] investigated many objective optimizations of shell and tube heat exchanger. Ammar Ali Abd et al. [13] used many parameters to achieve high enough heat transfer coefficient and the pressure drop within specification. Nitheesh Krishnan et al. [14] gave a detailed overview on shell and tube heat exchanger. Gabriel Batalha Leoni et al. [15] highlights the importance of baffle clearances which reduce pressure drop on baffle window. Julia C. Lemos et al. [16] indicated a better performance when compared to conventional approaches. Kyle A. Palmer et al. [17] estimated thermal fouling resistance at uncertain operating conditions, explored in a series of case conditions. Eshita Pal et al. [18] presented the effect of flow field with a comparison of analytical methods. Labbadlia et al. [19] showed that the arrangement of the tubes has a significant influence on the flow distribution. Abazar Vahdat Azad et al. [20] reduced the cost of heat exchanger to 50% based on constructal theory.

In this study, Copper tubes, single segmental baffles as aluminium and shell as PVC pipe were the materials used for the process. Fouling is reduced to get better heat transfer and the size is reduced to decrease the cost of the heat exchanger. The experimental and the CFD analysis are compared and they are in well correlation with negligible percentage of error.

2. EXPERIMENTAL WORK

2.1 EXPERIMENTATION

The materials and equipment's used for the experimentation were shell (PVC is used), copper tubes, submersible pumps and single segmental baffles made of aluminium.

2.1.1 FABRICATION TECHNIQUES

PVC pipe is used as the shell. The shell contains the core of the heat exchanger. Shell consists of cylindrical cross section. The standard ratio of diameter of shell to length of shell is 1:4. The shell is drilled with a hole of required diameter is drilled on the circumference of the shell on both the ends at opposite directions. A cup with same diameter with a hole on its cover is used to join all the tube to make the flow equal in all the tubes as shown in fig 1 & 2.

Figure 1. Shell when cups are opened

Figure 2. Shell when cups are closed

In fig 3 & 4 it is shown that the single segmented baffles are placed at equal distance to support the tube and to regulate the flow. It offers low pressure drop and highest potential heat-transfer rate. The single segmented baffles are drilled with wholes to pass the copper tubes. The holes are drilled with a triangular pitch of 25mm for better heat transfer. The baffles do not have any contact with the PVC pipe this is to avoid fouling and stagnation of the fluid.

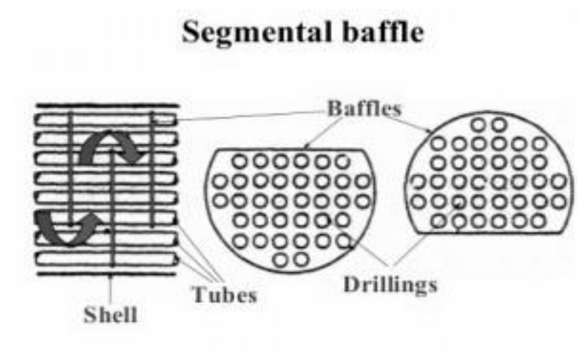

Figure 3. Single segmental baffle

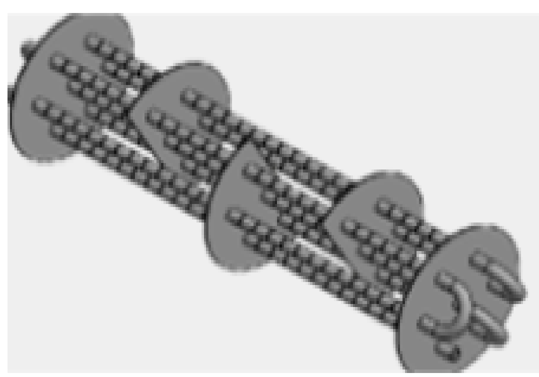

Figure 4. Baffle fixed setup

The copper tubes of same length that of the shell is taken. The tubes are arranged in the circumference of the support plate of the required radius. The copper tubes are corrugated to improve the heat transfer rate as shown in fig 5. Corrugation is the process of creating a series of rings on the outer surface of the tube. The support plates are drilled with holes of same radius of tubes. The support plate and tubes are joined using gas welding as shown in fig 6.

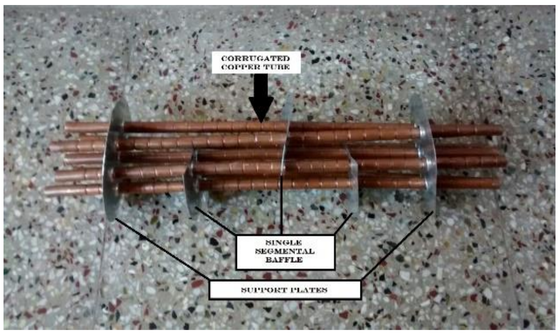

Figure 5. Corrugated copper tubes

Figure 6. Inner setup of heat exchanger

The submersible pump is used to pump the fluid to the copper tube and the shell. Submersible pump is a device which has a sealed motor close-coupled to pump body. The whole assembly is submerged in the fluid to be pumped. The exploded view and the assembly of shell and tube heat exchanger is shown in fig 7&8.

Figure 7. Exploded view of shell and tube heat exchanger

Figure 8. Assembly of shell and tube heat exchanger

2.1.2 EXPERIMENTAL TEST

The hot fluid is passed into the copper tubes and the cold fluid is passed in the shell. This is because the cold fluids have more surface area of contact. This increases the heat transfer rate and hence the effectiveness also increases. The hot fluid is pumped from the tank into the tube. The hot fluid is passed into one side of the cup and comes out by other side. The cold fluid from the tank is pumped into the upper pipe of the shell and it gets heat from hot gas and flows to pipe on the down side of the shell. The flow is made as counter flow. The cold fluid will gain heat from hot fluid and comes out of the shell. The hot fluid will lose heat and comes out of the tube.

2.2 CFD ANALYSIS

Computational fluid dynamics (CFD) starts with the construction of desired geometry and mesh for modelling. Generally, geometry is made simple for the CFD studies. Meshing is the discretization of the domain into small volumes where the equations are solved by the help of iterative methods. Modelling starts with the describing of the boundary and initial conditions for the dominion and leads to modelling of the entire system. Finally, it is followed by the analysis of the result, discussions and conclusions.

2.2.1 MODELING

Heat exchanger is built in the Autodesk Inventor Professional 2015 as shown in fig 9. It is a counter-flow heat exchanger. First, a new design page is created. Then all the parts are created and assembled. The specification of the model is given in table 1.

TABLE 1: Model specifications

Parameters	Specifications
Diameter of the shell (Ds)	90 mm
Diameter of the tube (Dt)	70 mm

Length of the shell (L)	350 mm
Baffle spacing (B)	70 mm
Pitch (P)	25 mm
Number of passes (Nt)	7
Inside tube material	Copper
Outer shell material	PVC

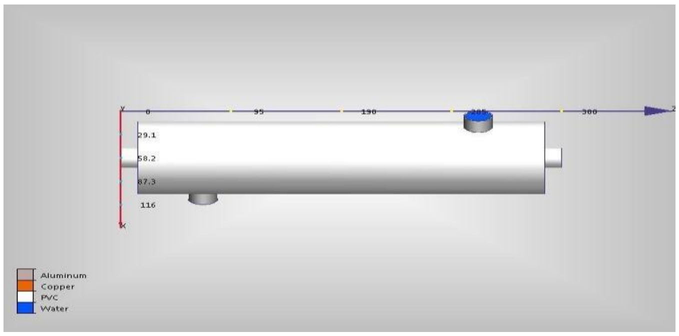

Figure 9. Designed Model

2.2.2 GEOMENTRY

The created model is imported to the Autodesk simulation CFD 2015 and various geometric features are defined. The CFD created volumes are made to represent fluids. The properties applied are given in table 2.

TABLE 2: Component properties

COMPONENTS NAME	PROPERTIES	
PVC	X-Direction	0.25 W/m-K
	Y-Direction	Same as X-dir.
	Z-Direction	Same as X-dir.
	Density	1400.0 kg/m³
	Specific heat	1250.0 J/kg-K
	Emissivity	0.92
Water	Density	Piecewise Linear
	Viscosity	0.001003 Pa-s
	Conductivity	0.6 W/m-K
	Specific heat	4182.0 J/kg-K
	Compressibility	2185650000.0 Pa
	Emissivity	1.0
Aluminium	X-Direction	204.0 W/m-K
	Y-Direction	Same as X-dir.
	Z-Direction	Same as X-dir.
	Density	2707.0 kg/m³
	Specific heat	896.0 J/kg-K
	Emissivity	0.2
Copper	X-Direction	Piecewise Linear
	Y-Direction	Same as X-dir.
	Z-Direction	Same as X-dir.
	Density	8939.58 kg/m³
	Specific heat	380.718 J/kg-K
	Emissivity	0.6

2.2.3 BOUNDARY CONDITIONS

Boundary conditions are used according to the need of the model. The inlet and outlet conditions are defined as mass flow inlet and pressure outlet. The walls are separately specified with respective boundary conditions. No slip condition is considered for each wall. Except the tube wall other walls are set to zero heat flux condition. The boundary conditions are given below,

- Inlet hot fluid temperature = 53^0 C
- Inlet cold fluid temperature = 33^0 C
- Hot fluid flow rate = 0.000277 m^3/s
- Cold fluid flow rate = 0.000277 m^3/s

2.2.4 MESH

Initially coarser mesh is generated. This mesh contains mixed Tetra and Hexahedral cells at the boundaries. It is made sure that structured hexahedral is used. It is meant to reduce numerical diffusion as much as possible by structuring the mesh in a well manner, particularly near the wall region as shown in fig 10.

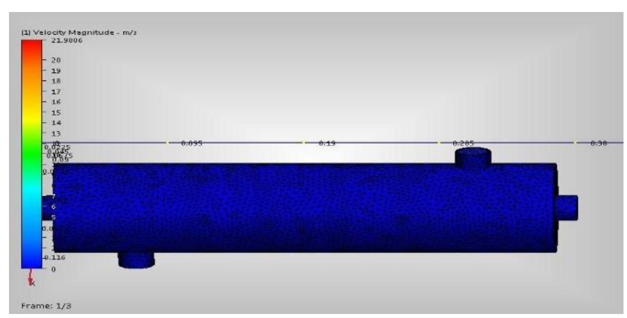

Figure 10. Mesh Model

3. RESULTS AND DISCUSSIONS

3.1 EXPERIMENTAL RESULTS

The results from the experimental test are achieved by using the following formulae,
For shell side,
Area of cross flow (A_c) = (P-D_t) B(B/P)

Area of baffle (A_b) = $F_b\frac{\pi D_s^2}{4}$ - $N_t\frac{\pi D_t^2}{4}$

For tube side,

Flow area per tube (A_t) = $\frac{\pi D_t^2}{4}$

Total flow area of tube = $N_t A_t$

Heat transferred from hot fluid (Q_h) = $m_h C_{ph}(T_1\text{-}T_2)$

LMTD = $\frac{(T_1-t_2)-(T_2-t_1)}{In[(T_1-t_2)-(T_2-t_1)]}$

Effectiveness of heat exchanger (€) =Q/Q_{max}

Table 3: Experimental value

	Field	Experimental Value
Temperature value	T_1	53^0C
	T_2	43.4^0C
	t_1	33^0C
	t_2	38.8^0C

3.2 CFD RESULTS

The results which are obtained is represented by various contours, vectors and plots. It is shown below,

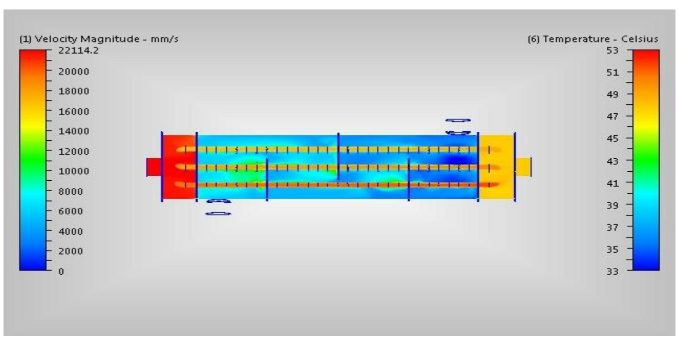

Figure 11. Temperature Distribution

The fig 11 shows the temperature distribution along the heat exchanger and it also shows the reheat chamber which is indicated in red and this reheat chamber helps in increasing the heat transfer rate.

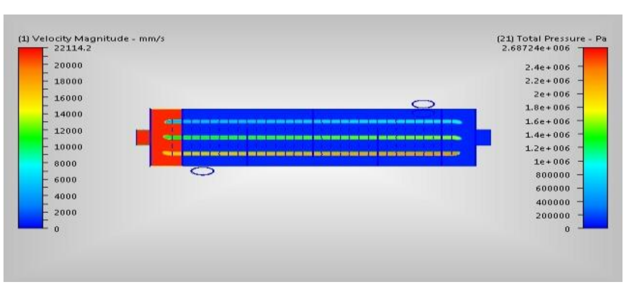

Figure 12. Pressure Distribution

The fig 12 shows the pressure distribution across the heat exchanger and the change is indicated from red to blue which shows the lowering of pressure. Here, the red is indicated as the highest value.

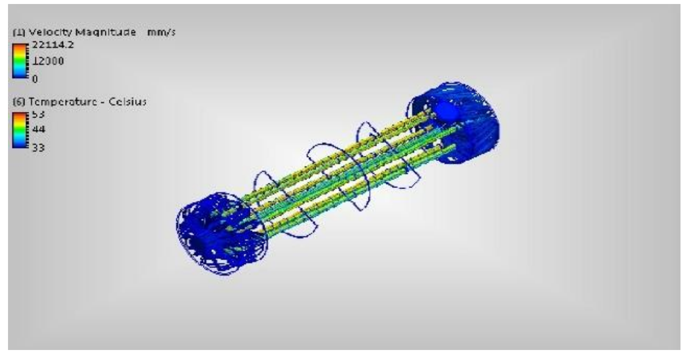

Figure 13. Hot Fluid Flow Simulation

The fig 13 & 14 shows the fluid flow simulation of the hot and cold fluid in the heat exchanger by showing all the ways the fluids are passed.

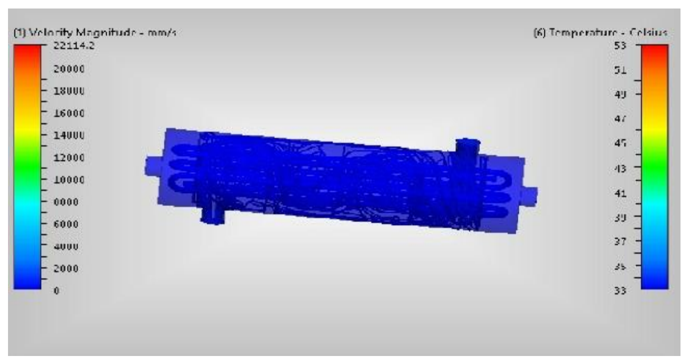

Figure 14. Cold Fluid Flow Simulation

The fig 15 shows the force occurred on the wall during simulation.

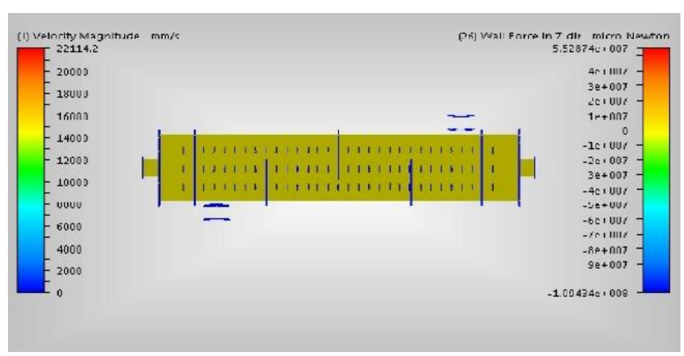

Figure 15. Wall Force

Table 4: CFD value

	Field	Analyzed Value
Temperature value (average)	T_1	53^0C
	T_2	47.7^0C
	t_1	33^0C
	t_2	39.5^0C
Fluid force on wall (average)	Pressure in X	-18.272 N
	Pressure in Y	2.2753 N
	Pressure in Z	2348.6 N
	Shear in X	-0.013275 N
	Shear in Y	-0.17678 N
	Shear in Z	2.9093 N

The results from both experimental and simulation are compared and there is a negligible percentage of error as shown in fig 16.

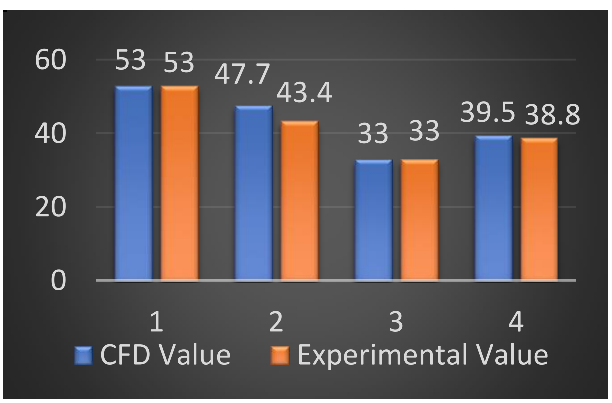

Figure 16. Comparison of CFD and experimental value

4. CONCLUSIONS

A study was done on flow and heat transfer analysis using CFD. The modeling was done using Autodesk Inventor Professional 2015 and the simulation was carried out using Autodesk Simulation CFD 2015. Based on the experimental and CFD results it can be concluded as follows:

- The heat transfer rate was better this is due to the corrugated copper tubes that were used to improve the contact surface of the tube.
- Compared to the other types of shell & tube heat exchangers the fouling is very less this was achieved due to the clearance between the baffles and the PVC pipe which causes shear stress on shell wall. This stress reduces fouling.
- Low pressure drops and high potential heat transfer rate were achieved. This was due to the single segmental baffles which was used to regulate the flow of the fluid and they are very easy to replace.
- The shell & tube heat exchanger cost was reduced this was due to the design and materials used in the heat exchanger.

REFERENCES

[1] Pramod S. Purandare, Mandar M. Lele, Raj Kumar Gupta "Investigation on thermal analysis of conical coil heat exchanger", International Journal of Heat and Mass Transfer, 90,2015,118-1196.

[2] M. Ghazikhani, I. Khazaee, S. M. S. Monazzam, J. Taghipour "Experimental Investigation of the Vortex Generator Effects on a Gas Liquid Finned Tube Heat Exchanger Using Irreversibility Analysis", Arabian journal for science and engineering, 39,3,2014,2107-2116.

[3] Dillip Kumar Mohanty, Pravin Madanrao Singru "Numerical Method for heat transfer and fouling analysis of a shell and tube heat exchanger using statistical analysis", Korean Journal of Chemical Engineering, 29,9,2012,1144-1150.

[4] A. I. Zinkevich, V. N. Sharifullin, A. V. Sharifullin "Analyzing the effect on heat transfer due to nonuniform distribution of liquid flow among the tubes of a shell and tube heat exchanger", Thermal Engineering, 57,9,2010,807-809.

[5] Wei Liu, ZhiChun Liu, YingShuang Wang, SuYi Huang "Flow mechanism and heat transfer enhancement in longitudinal flow tube bundle of shell and tube heat exchanger", Science in China Series E: Technological Sciences, 52,10,2009,2952-2959.

[6] Seong-Yeon Yoo, Jin-Hyuck Kim, Kyu-Hyun Han "Thermal performance analysis of heat exchanger for closed we cooling tower using heat and mass transfer analogy", Journal of Mechanical Science and Technology, 24,4,2010,893-898.

[7] Andre L. H. Costa, Eduardo M. Queiroz "Design optimization of shell and tube heat exchangers", Applied Thermal Engineering, 28,14-15,2008,1798-1805.

[8] K. Somasekhar, K. N. D. Malleswara Rao, V. Sankararao, Raffi Mohammed, M. Veerendra, T. Venkateswararao "A CFD Investigation of Heat Transfer Enhancement of Shell and Tube Heat Exchanger Using Al_2O_3-Water Nanofluid", Materials Today Proceedings, 5,2018,1057-1062.

[9] Priyanka G, M. R. Nagaraj "CFD Analysis of Shell and Tube Heat Exchanger with and without Fins for Waste Heat Recovery Applications", IJSR, 3,7,2014,1437-1440.

[10] Pranita Bichkar, Ojas Dandgaval, Pranita Dalvi, Rhushabh Godase and Tapobrata Dey "Study of Shell and Tube Exchanger with the Effects of Types of Baffles", Procedia Manufacturing, 20,2018,195-200.

[11] N Santhisree, M Prashanthkumar, G Priyanka "Thermal analysis of shell and tube heat exchanger", IJMET, 8,5,2017,596-606.

[12] Bansi D. Raja, R. L. Jhala, Vivek Patel "Man-objective optimization of shell and tube heat exchanger", Thermal science and Engineering progress, 2,2017,87-101.

[13] Ammar Ali Abd, Samah Zaki Naji "Analysis study of shell and tube heat exchanger for clough company with reselect different parameters to improve the design", Case Studies in Thermal Engineering, 10,2017,455-467.

[14] Nitheesh Krishnan M. C, Suresh Kumar B "An overview on shell and tube heat exchanger", IJESC, 6,10,2016,2632-2636.

[15] Gabriel Batalha Leoni, Tania Suaiden Klein, Ricardo de Andrade Medronho "Assessment with computational fluid dynamics of the effects of baffle clearances on the shell side flow in a shell and tube heat exchanger", Applied thermal engineering, 112,2017,497-506.

[16] Julia C. Lemos, Andre L. H. Costa, Miguel J. Bagajewicz "Linear method for the design of shell and tube heat exchangers including fouling modeling", Applied thermal engineering, 125,2017,1345-1353.

[17] Kyle A. Palmer, William T. Hale, Kyle D. Such, Brian R. Shea, George M. Bollas "Optimal design of tests for heat exchanger fouling identification, Applied thermal engineering, 95,2016,382-393.

[18] Eshita Pal, Inder Kumar, Jyeshtharaj B. Joshi, N. K. Maheshwari "CFD Simulation of shell side flow in a shell and tube heat exchanger with and without baffles", Chemical engineering science, 143,2016,314-340.

[19] O. Labbadlia, B. Laribi, B. Chetti, P. Hendrick "Numerical study of the influence of tube arrangement on the flow distribution in the header of shell and tube heat exchangers", Applied thermal engineering, 126,2017,315-321.

[20] Abazar Vahdat Azad, Majid Amidpour "Economic optimization of shell and tube heat exchanger based on constructal theory", Energy, 36,2,2011,1087-1096.

AUTHORS

Sugin Elankavi R
Finished UG in PERI Institute of Technology and went for higher studies.

Uma Shankar T
Finished UG in PERI Institute of Technology and now works as a Solution Expert at KKM Soft Pvt. Ltd.