Experimental Analysis of Thermal and Hydrodynamic Performance in Shell and Tube Heat Exchangers

Dr.G.Sulochana[1], Ponakala Hema [2], Ch.V.Prasad[3] Pidatala Indrasena[4], Korra Mahesh Babu[5], Ratsa Bhavani prasad [6]

[1 and 3] Assistant Professor, Department of Mechanical Engineering,

[1 and 3] University College of Engineering Narasaraopet-JNTUK, India Velagapudi Ramakrishna Siddhartha Engineering College, India
[2,4,5,6] Department of Mechanical Engineering, University College of Engineering, Narasaraopet-JNTUK, India

---***---

Abstract

Improving the heat transfer rate is a crucial aspect of improving the performance and efficiency of a shell and tube heat exchanger. Continuous efforts are made to optimize its effectiveness by analysing the impact of various parameters such as mass flow rate, hot and cold fluid temperatures and overall heat transfer. Based on the results from the developed model, these parameters are adjusted to achieve better performance. To accomplish this, multiple factors are modified, including fluid flow rate, tube diameter, and number of tubes, baffle configuration and baffle spacing. The experiment is conducted on a shell and tube heat exchanger with parallel fluid flow and counter fluid flow and readings are recorded. Using this data, the overall heat transfer coefficient is calculated for different flow rates using both the LMTD and NTU methods, and their results are compared. Additionally, a CATIA V5 model is created, and CFD analysis is performed using ANSYS R20.1 for one set of readings under both parallel and counterflow conditions. Observed the maximum wall heat transfer coefficient in both hot and cold domains is determined. Furthermore, additional baffles are introduced, and all results are analysed to validate the design improvements.

Key words: Shell and tube Heat Exchanger, Double segmental baffles, CATIA V5 Overall Heat Transfer Coefficient, CFD, LMTD and NTU.

1. INTRODUCTION

A heat exchanger is a device designed to transfer thermal energy from a hot fluid to a cold fluid efficiently, with minimal cost and maintenance. The efficiency of heat transfer depends on factors such as the thermal conductivity of the separating wall and the convective heat transfer coefficient between the fluids and the wall. Additionally, boundary conditions like insulation or adiabatic constraints affect the rate of heat exchange. Heat transfer in a heat exchanger typically involves convection in both fluids and conduction through the separating wall. Heat exchangers can be categorized based on several factors, including the transfer process, fluid flow direction, and number of fluids, surface compactness, heat transfer mechanism, and construction type.

[1] Young-Seok Son, Jee-Young Shin, explained about “Performance of a Shell and Tube Heat Exchanger with Spiral Baffle Plates”. In a conventional shell-and-tube heat exchanger, fluid contacts with tubes flowing up and down in a shell, therefore there is a defect in the heat transfer with tubes due to the stagnation portions. Fins are attached to the tubes in order to increase heat transfer efficiency, but there exists a limit. Therefore, it is necessary to improve heat exchanger performance by changing the fluid flow in the shell. In this study, a highly efficient shell-and-tube heat exchanger with spiral baffle plates is simulated three-dimensionally using a commercial thermal-fluid analysis code, CFX4.2. In this type of heat exchanger, fluid contacts with tubes flowing rotationally in the shell. It could improve heat exchanger performance considerably because stagnation portions in the shell could be removed. It is proved that the shell-and-tube heat exchanger with spiral baffle plates is superior to the conventional heat exchanger in terms of heat transfer. [2] Kaushik Parmar, Osma Gora, Kashyap Desai, Niraj Kumar C Mehta, Heat Exchanger is a device used intensively for heat transfer form fluid. Thus, all various type of heat exchanger. Our concentration is on shell and tube type heat exchanger. We will design the heat exchanger by bell Delaware method to increase heat transfer by using various material and geometries. Shell-and-tube heat exchangers are widely used in many industrial areas, and more than 35-40% of heat exchangers are of this type due to their robust geometry construction, easy maintenance, and possible upgrades. [3] Harishchandra Thakur, Rahul Singh, “CFD Analysis of Shell and Tube Heat Exchanger “, in this present study, attempts are made to investigate the impacts of new type baffle on fluid flow and the heat transfer characteristics of a shell-and-tube heat exchanger. The shell side design has been numerically investigated by modelling a shell-and-tube heat exchanger. The study is concerned with a single shell side pass parallel flow heat exchanger. The flow and temperature fields inside the shell are studied using CFD software tool ANSYS FLUENT 12.0. The heat exchanger performance is investigated by varying mass flow rate and baffle cut on new type baffle. From the CFD simulation results, the shell side outlet temperature, pressure drop, optimal mass flow rate and the optimum baffle cut for the given heat exchanger geometry are determined. The results are compared with the segmental baffle, which shows that the use of new type baffle reduces the pressure drop and improves the performance of heat exchanger.

1.2 Shell and tube heat exchanger

A shell and tube heat exchanger consists of round tubes arranged inside a cylindrical shell, with the tubes aligned parallel to the shell's axis. One fluid flows through the tubes, while the other moves around and along the tubes within the shell. The main components of a shell and tube heat exchanger include, baffles, direct the fluid flow and enhance heat transfer, tube sheets, secure the tubes at both ends, shell, the outer casing that encloses the tube bundle, front end head, The entry section for the tube-side fluid, rear end head, the exit section for the tube-side fluid, tube bundle, a collection of tubes where heat transfer occurs.

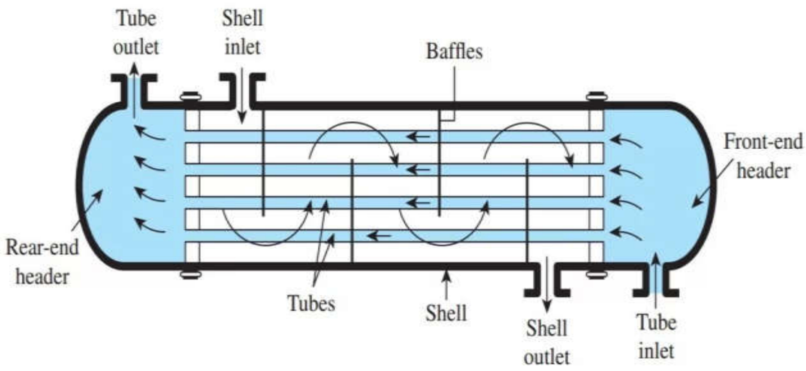


Fig 1: Shell and Tube Heat Exchanger

1.3 Types of heat exchangers

1.3.1 Parallel Flow

In a shell and tube heat exchanger, parallel flow (or co-current flow) occurs when the hot and cold fluids enter the heat exchanger at the same end and move in the same direction. The temperature difference between the hot and cold fluids decreases along the length of the exchanger. Here's how it behaves, at the inlet the temperature difference is maximum because both fluids enter from the same side, with one being hot and the other cold, as they flow together the cold fluid heats up while the hot fluid cools down, reducing the temperature difference gradually, at the outlet the temperature difference is minimum, and both fluids may approach a nearly equal temperature, limiting heat transfer efficiency.

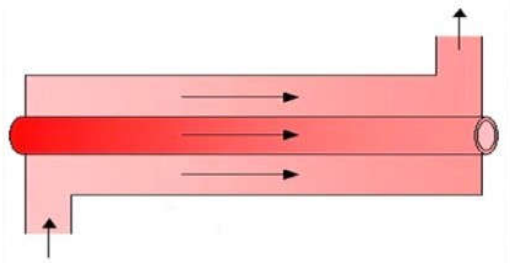


Fig.2. Parallel flow

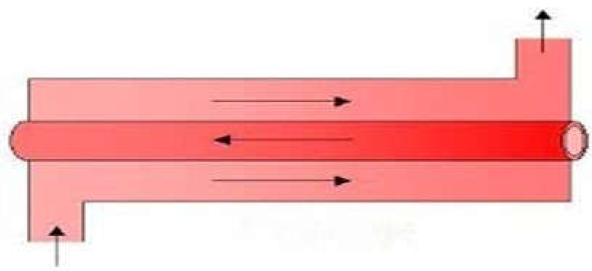


Fig.3. Counter flow

1.3.2 Counter Flow

Counter flow in a shell and tube heat exchanger occurs when the hot and cold fluids move in opposite directions. Characteristics of counterflow in a shell and tube heat exchanger, higher heat transfer efficiency, the temperature difference between the fluids remains more uniform along the exchanger, allowing better heat transfer. Greater temperature change, the cold fluid can reach a temperature closer to the inlet temperature of the hot fluid, which is not possible in parallel flow. More effective use of surface area, since the temperature difference is more evenly distributed, the heat exchanger operates more efficiently. This is a crucial factor that makes counterflow heat exchangers more efficient and preferable compared to parallel flow heat exchangers.

2. MATERIAL AND METHODOLOGY

2.1 Material: Water is used as the working fluid in both the shell and tube sides of the heat exchanger due to its excellent thermal properties, including high specific heat capacity and thermal conductivity. Water is commonly chosen for heat exchangers as it facilitates efficient heat transfer while maintaining a stable and predictable flow behaviour. Mild steel is used in shell. The reasons for using mild steel are cost-effective, good mechanical strength, high toughness, excellent weldability, good machinability, availability, coating compatibility. Copper is used in tubes. The reasons for using copper are excellent thermal conductivity which allows for efficient heat transfer between the fluids, corrosion resistance, anti-fouling properties, malleability and workability, durability and reliability, cost-effectiveness.

2.2 Methodology: The analysis is performed using ANSYS, a powerful simulation tool for computational fluid dynamics (CFD) and thermal analysis. The methodology begins with geometric modelling in ANSYS space claim or design modeler, where the shell, tubes, and baffles are designed based on specific dimensions, next, the model undergoes meshing, where it is discretized into small elements to ensure accurate numerical calculations. A fine mesh is applied near the tube walls and heat exchange surfaces to capture flow and temperature variations effectively. Once the meshing is complete, boundary conditions are set, including inlet temperature, mass flow rate, and pressure conditions for both hot and cold-water streams. The simulation is conducted in ANSYS. Finally, post-processing is performed to analyse temperature distribution, velocity profiles, pressure drop, and heat transfer efficiency. The results are compared with theoretical calculations or experimental data to validate the model. This methodology helps in optimizing the heat exchanger design, ensuring enhanced thermal performance and energy efficiency.

3. EXPERIMENTAL SETUP

Fig 4: Experimental Setup of Shell and Tube Heat Exchanger

3.1 Table 1: Specification of shell and tube heat exchange

S.No.	Parameters	Values	Material
1.	Shell		
	Inner Diameter	200mm	Mild Steel
	Thickness	5mm	Mild Steel
	Length	1000mm	Mild Steel
2.	Tubes		
	Outer Diameter	8mm	Copper
	Inner Diameter	5.5mm	Copper
	No of Tubes	15	Copper
	Length	1020mm	Copper

3.2 Methods of Evaluation

3.2 Table 2: Observation readings of parallel flow heat exchanger

SR. NO.	Hot Water	Temperature of Hot Water		Cold Water	Temperature of Cold Water	
	Flow Rate (Ips)	Inlet Temp. (°C)	Outlet Temp. (°C)	Flow Rate (Ips)	Inlet Temp. (°C)	Outlet Temp. (°C)
1	0.028	50	46	0.023	32	34
2	0.022	49	44	0.0228	31	34
3	0.03	51	42	0.178	33	37
4	0.035	43	41	0.166	31	35

3.2 Logarithmic Mean Temperature Difference (LMTD) method: In heat exchanger analysis, the average temperature differential between two fluids one hot and one cold along the heat exchanger's length is represented by the Logarithmic Mean Temperature Difference (LMTD). Since the hot fluid cools and the cold fluid heats up as it flows, the temperature differential between the fluids in the majority of heat exchangers is not constant. LMTD provides a more accurate value that reflects the gradual changes in temperature by using a logarithmic formula rather than a simple average, which can be imprecise. This value is significant because the magnitude of the temperature differential between the two fluids determines the rate of heat transfer. A greater driving force for heat transfer is indicated by a larger LMTD.

* The heat transfer rate may be obtained from the overall energy balance for hot side

$$Q_h = m_h\, c_h\ (T_{hi} - T_{ho}) \quad , \quad Q_c = m_c\, c_c\ (T_{co} - T_{ci})$$

* Hot Fluid Lost the heat = Cold Fluid Gain the Heat

$$m_h\, c_h\ (T_{hi} - T_{ho}) = m_c\, c_c\ (T_{co} - T_{ci}),\ Tco = \frac{Qc}{m_c\, Cc} + Tci \quad , \ \Delta T_1 = T_{hi} - T_{ci} \quad , \Delta T_2 = T_{ho} - T_{co}$$

* Logarithmic Mean Temperature Difference can be calculated by using following formula: -

$$\Delta Tm = \frac{\Delta T1 - \Delta T2}{\ln \frac{\Delta T1}{\Delta T2}} \quad . K$$

* The overall heat transfer coefficient can be calculated by using following formula: -

$Q = U_o A \Delta T_m$, $U_o = \frac{Q}{A \Delta T_m}$

3.3 Number of transfer units (NTU) METHOD: NTU stands for Number of Transfer Units, and it is a dimensionless parameter used in the analysis and design of heat exchangers. It indicates how effectively heat is being transferred between two fluids in a heat exchanger. The NTU value depends on the heat exchanger's surface area, the overall heat transfer coefficient, and the heat capacity rate of the fluid with the lower heat capacity. A higher NTU value generally means more efficient heat transfer. The NTU method is particularly useful when the outlet temperatures of the fluids are not known in advance, making it suitable for complex heat exchanger configurations like shell and tube types. It allows engineers to calculate the effectiveness of the heat exchanger and understand how well it performs under specific operating conditions.

* The heat capacity rates are calculated by following formula

$C_h = m_h c_h$, $Cc = m_c c_c$, If $m_h c_h < m_c c_c$: $C = \frac{m_h C_h}{m_c Cc}$, $Q_{max} = C_{min} (T_{hi} - Tci)$

* Actual heat transfer rate is calculated by: $Q_{act} = m_h c_h (T_{hi} - T_{ho})$

* Effectiveness of Heat Transfer can be calculated by: $\varepsilon = \frac{Qact}{Qmax}$

* Effectiveness of Heat Transfer by parallel flow: $\varepsilon = \frac{1-\exp[-NTU(1+C)]}{(1+C)]}$

* The Overall Heat Transfer Coefficient can be calculated by: $NTU = \frac{UA}{Cmin}$, $U = \frac{NTU \times Cmin}{A}$

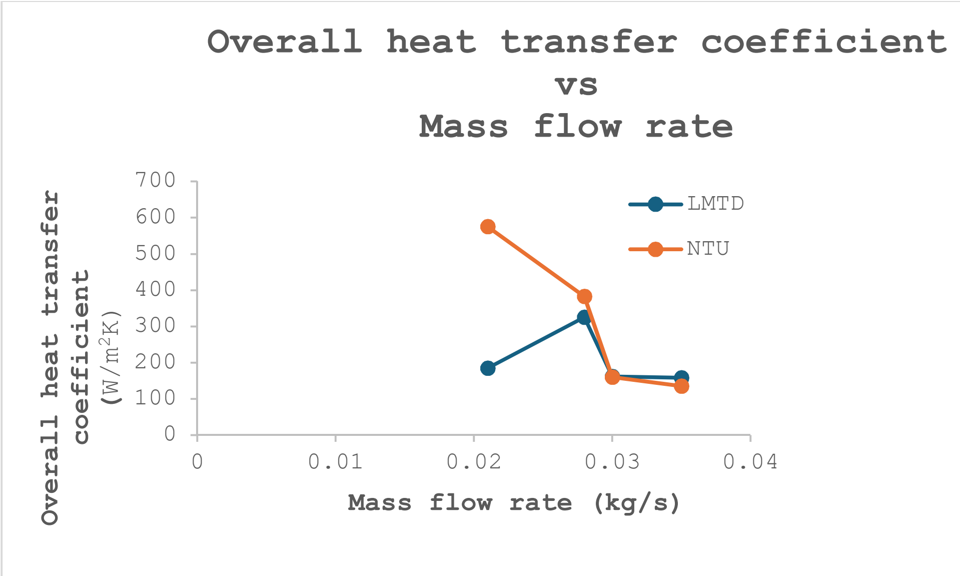
3.3. Table 3: Overall Heat Transfer Coefficient by LMTD Method

S. NO	Mass Flow Rate (kg/s)	Overall Heat Transfer Coefficient [W/m^{2}K]
1	0.021	185.123
2	0.028	325.338
3	0.03	161.926
4	0.035	158.321

3.3. Table.4. Overall Heat Transfer Coefficient by NTU Method

S. NO	Mass Flow Rate(kg/s)	Overall Heat Transfer Coefficient [W/m^{2}K]
1	0.021	575.92
2	0.028	382.64
3	0.03	160.23
4	0.035	135.13

The Comparison of the overall heat transfer coefficient is shown by graph



Graph: Overall heat transfer coefficient v/s Mass Flow Rate

From Table 4, it is observed that overall heat transfer coefficient is higher in NTU method than LMTD at 0.021 kg/s. In heat exchanger analysis, both the NTU (Number of Transfer Units) method and the LMTD (Log Mean Temperature Difference) method are used to determine the heat transfer rate. In certain instances, though, the NTU approach might seem to forecast a greater total heat transfer rate than the LMTD approach. This discrepancy results from the approaches taken by each method to the problem. The NTU method focuses on the effectiveness of the heat exchanger and assumes ideal flow arrangements to estimate the maximum possible heat transfer, often based on the minimum heat capacity rate and the largest potential temperature difference. However, the LMTD approach tends to provide a more conservative estimate because it is solely based on the fluids' actual inlet and outlet temperatures.

3.3 Computational Fluid Dynamics

Computational Fluid Dynamics (CFD) is a field of study that deals with the simulation and analysis of fluid flow using computers. It involves the use of mathematical models, numerical methods, and algorithms to understand how fluids such as liquids and gases move and interact with their surroundings. Instead of performing physical experiments, CFD allows engineers and scientists to create virtual models and analyse the behaviour of fluids in various conditions. This helps in predicting flow patterns, pressure distribution, heat transfer, and other related phenomena. CFD is widely used in industries like aerospace, automotive, mechanical, and environmental engineering to improve designs, enhance performance, and reduce the need for costly experimental testing.

3.3.1 Design of Model

Using the actual dimensions of the experimental setup geometrical model is created using CATAIA V5 Software. Now two more models were created adding 2 and 4 baffles in the existing model as a modification. Export the geometry in a compatible format (e.g., IGES or STEP) for CFD software. Import the model into the CFD environment and establish the computational domain. Define precise boundary conditions such as inlets, outlets, walls, and symmetry planes. Assign material properties and fluid characteristics that match the experimental data. Generate a refined mesh with adequate resolution in regions of high gradients.

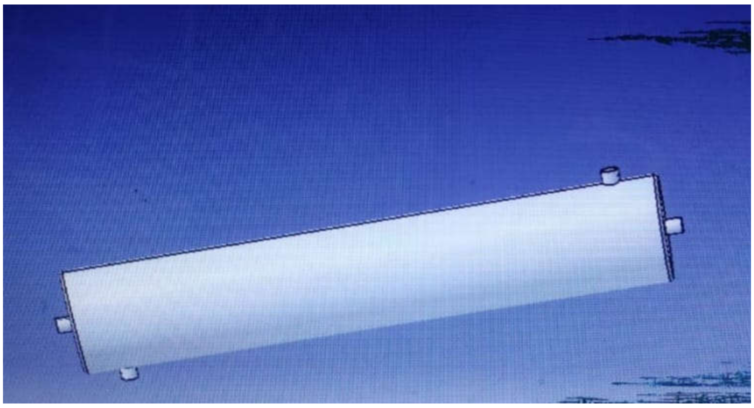


Fig 5: 3-Dimensional Model

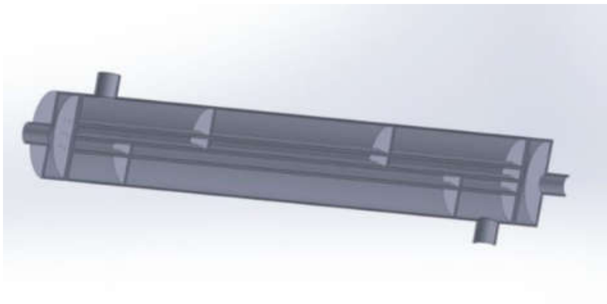


Fig 6: Heat Exchanger with 2 baffles

Fig 7: Heat Exchanger with 4 baffles

3.3.2 Meshing

The meshing part is done in the Meshing software package of ANSYS. The mesh was generated with a high smoothing and fine sizing. The complex geometry created in CATIA V5 must be discretized into a finite number of cells (mesh elements). A well-refined mesh is critical in regions with steep gradients (e.g., near walls or sharp edges) to capture the flow details accurately. Maintaining high mesh quality (e.g., orthogonality, aspect ratio, skewness) is essential. Poor-quality meshes can lead to numerical instability and inaccuracies. Adaptive meshing techniques might be used in critical areas to improve resolution without overly increasing the computational cost.

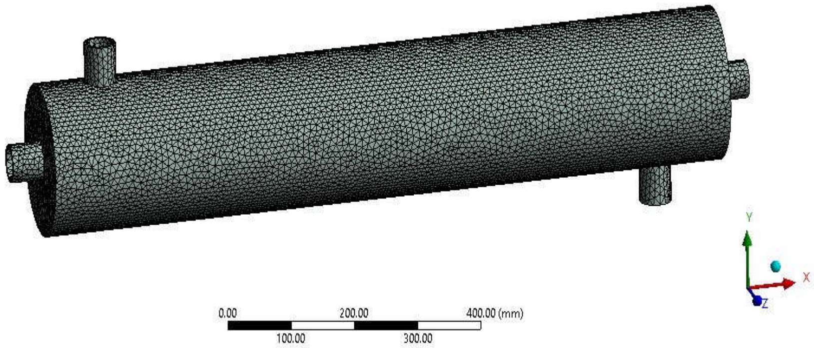


Fig 8: 3-D Model Mesh

3.4 Baffle plates in heat exchanger

Baffle plates are an essential component in a shell and tube heat exchanger, primarily used to improve the efficiency of heat transfer between the fluids. These plates are installed inside the shell and serve multiple purposes. First and foremost, baffles direct the shell-side fluid to flow across the tubes in a zigzag or cross-flow pattern, rather than allowing it to flow straight through. This increases the turbulence of the fluid, which enhances the heat transfer coefficient by reducing the thermal boundary layer around the tubes. Secondly, baffles help support and hold the tubes in place, preventing vibration and potential damage caused by high fluid velocities. Additionally, they promote a more uniform distribution of fluid across the tube bundle, ensuring better thermal performance and minimizing the risk of hot spots or poor heat transfer areas. By improving fluid dynamics and structural support, baffle plates play a critical role in increasing the overall effectiveness and reliability of a shell and tube heat exchanger.

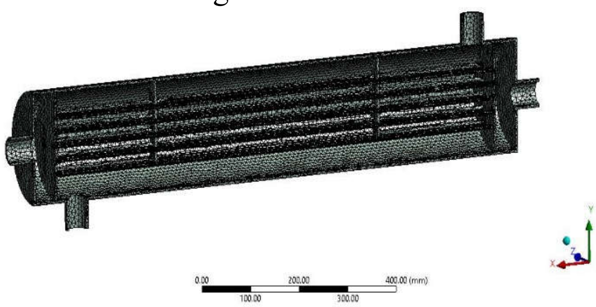


Fig 9: Mesh of Heat Exchanger with 2 Baffles

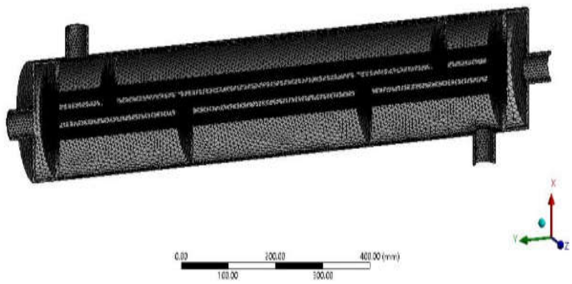


Fig 10: Mesh of Heat Exchanger with 4 Baffles

3.4.1 Impact of the absence of baffle on shell and tube heat exchanger

A shell and tube heat exchanger's performance and efficiency are greatly diminished if baffles are not used. Instead of being forced to flow across the tubes, the shell-side fluid travels in a smooth, parallel path through the shell when there are no baffles. This lowers the rate of heat transfer by reducing fluid turbulence, which results in poor mixing and the development of a thick thermal boundary layer surrounding the tubes. Furthermore, the tubes are more likely to vibrate as a result of fluid flow without the structural support of baffles, which over time may result in failure, noise, or mechanical damage. Additionally, the absence of baffles causes uneven fluid distribution.

3.5 Results of CFD

Maximum Value of Wall Heat Transfer Coefficient on hot and cold domain is calculated in ANSYS for parallel and counter flow with and without baffles for constant flow rate of 0.028kg/s of hot fluid and 0.021kg/s of cold fluid.

Parallel flow without baffles

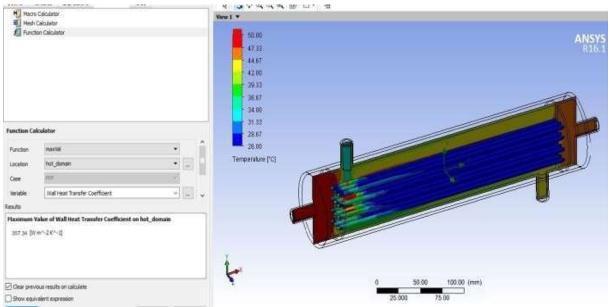


Fig 11: Maximum Value of Wall Heat Transfer Coefficient on Hot domain

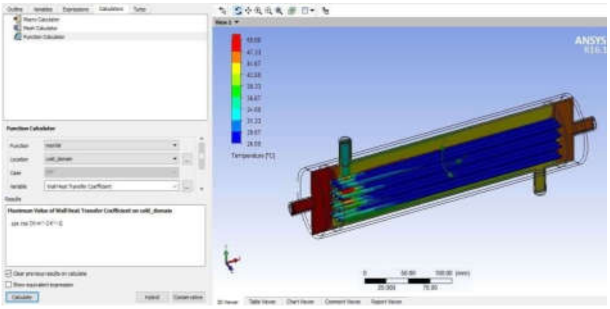


Fig 12: Maximum Value of Wall Heat Transfer Coefficient on Cold domain

Counter flow without baffles

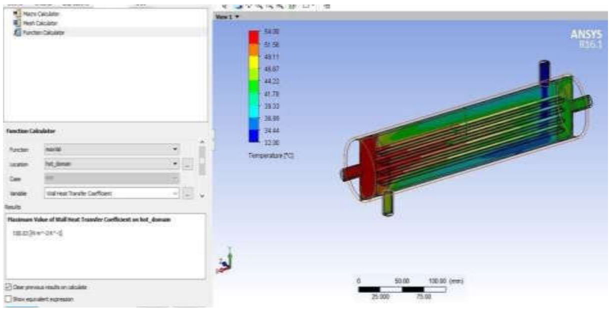


Fig 13: Maximum Value of Wall Heat Transfer Coefficient on Hot domain

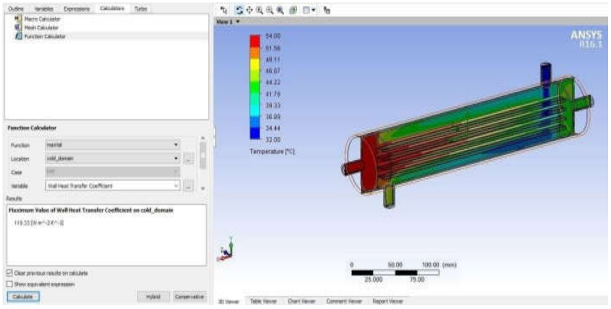


Fig 14: Maximum Value of Wall Heat Transfer Coefficient on Cold domain

Parallel Flow with 2 Baffles

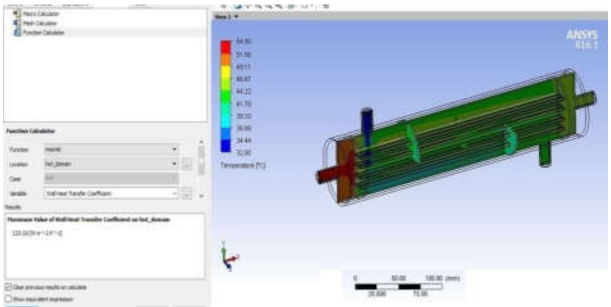


Fig 15: Maximum Value of Wall Heat Transfer Coefficient on Hot domain

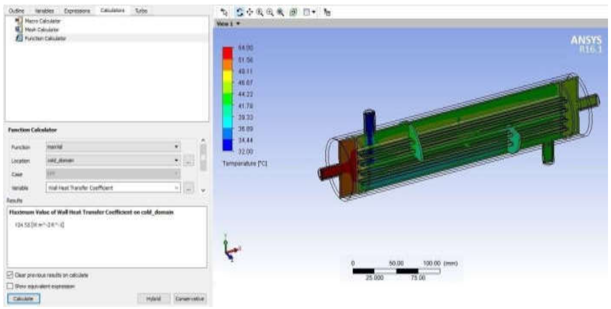


Fig 16: Maximum Value of Wall Heat Transfer Coefficient on Cold domain

Counter Flow with 2 baffles

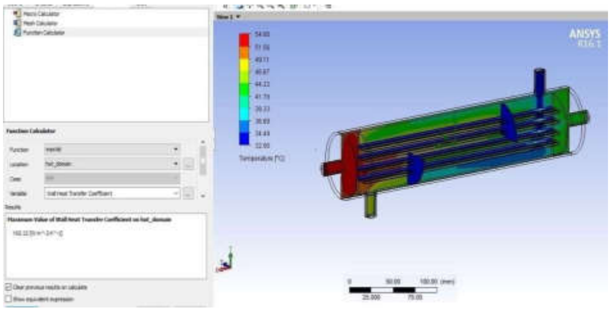


Fig 17: Maximum Value of Wall Heat Transfer Coefficient on Hot domain

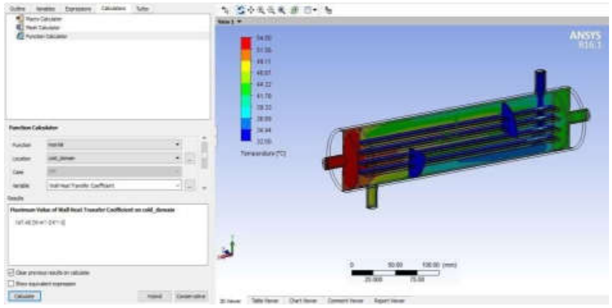


Fig 18: Maximum Value of Wall Heat Transfer Coefficient on Cold domain

Parallel flow with 4 baffles

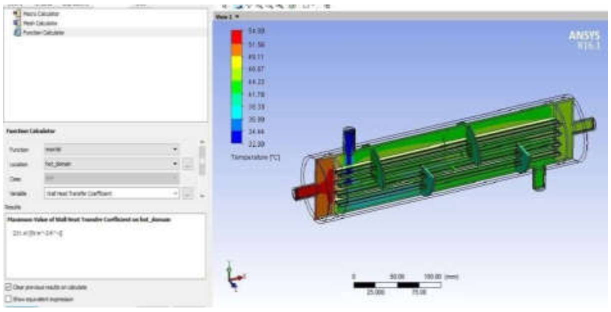


Fig 19: Maximum Value of Wall Heat Transfer Coefficient on Hot domain

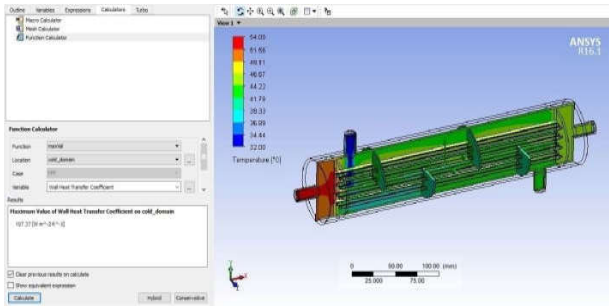


Fig 20: Maximum Value of Wall Heat Transfer Coefficient on Cold domain

Counter flow with 4 baffles

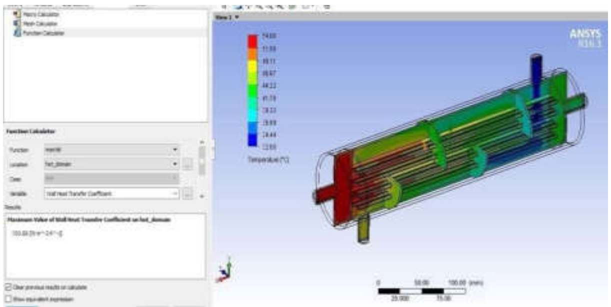


Fig 21: Maximum Value of Wall Heat Transfer Coefficient on Hot domain

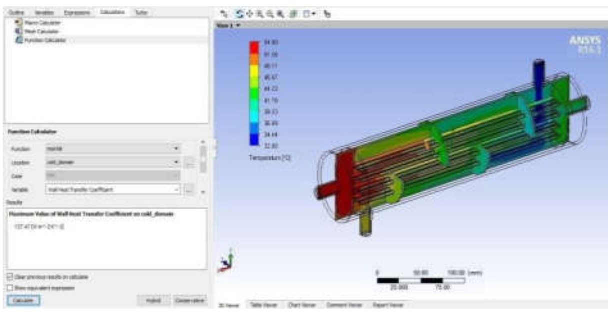


Fig 22: Maximum Value of Wall Heat Transfer Coefficient on Cold domain

4. Result and Discussion

Here the analysis was performed for six arrangements. The readings obtained are shown in Table. It is clear from the result that for same flow rate and same boundary conditions for different arrangement the overall heat transfer coefficient has a noticeable change.

The heat transfer coefficient for hot domain and cold domain is obtained.

SR. NO.	Type of Flow and Configuration	Maximum Value of Wall Heat Transfer Coefficient on Hot Domain [W/m²K]	Maximum Value of Wall Heat Transfer Coefficient on Cold Domain [W/m²K]	Overall Heat Transfer Coefficient [W/m²K]
1	Parallel flow	357.34	234.739	141.673
2	Counter flow	188.83	119.33	73.121
3	Parallel flow with 2 baffles	228.89	194.58	105.172
4	Counter flow with 2 baffles	168.32	147.49	78.60
5	Parallel flow with 4 baffles	231.41	197.37	106.51
6	Counter flow with 4 baffles	150.89	137.47	71.93

Table 4: Result obtained from analysis

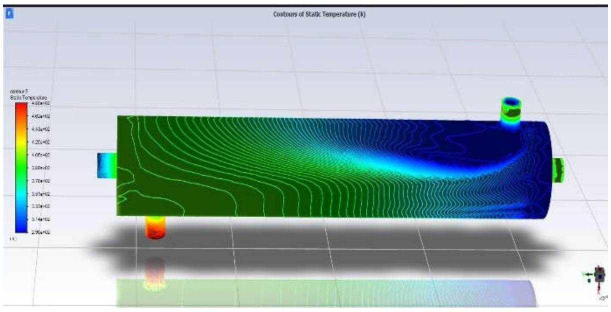


Fig 23: Contours of static temperature parallel flow

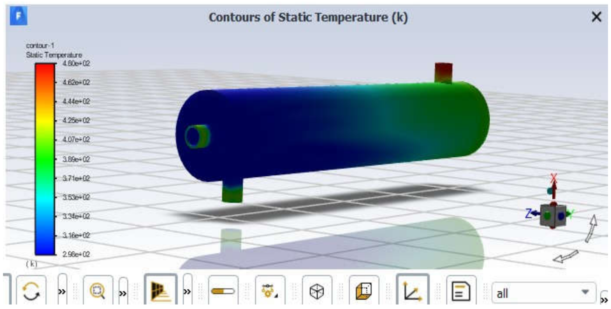


Fig 24: Contours of static temperature

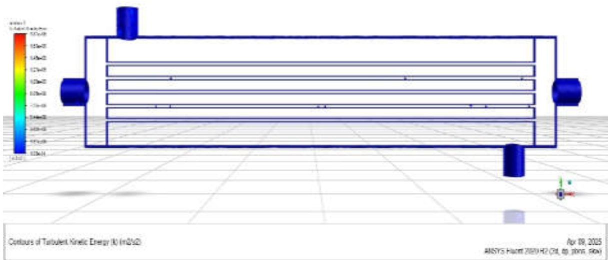


Fig 25: Contours of turbulence kinetic energy

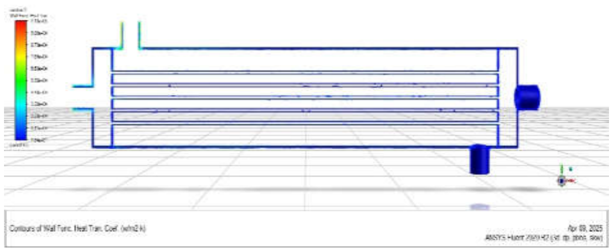


Fig 26: Contours of wall function heat Transfer coefficient

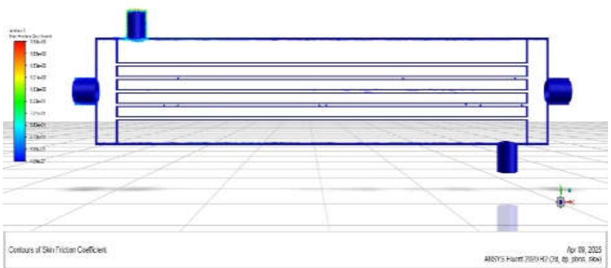


Fig 27: Contours of skin friction coefficient

Fig 28: Contours of surface heat transfer

Coefficient

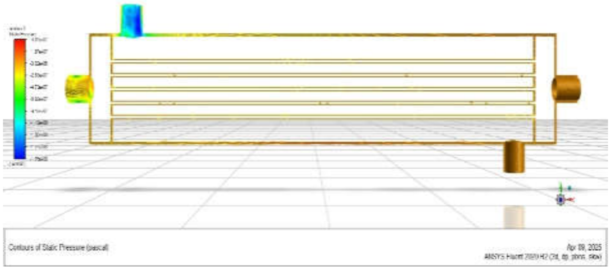


Fig 29: Contours of static pressure

Fig 30: Contours of enthalpy

Types of flow configuration	Contours of static temperature (°C)		Contours of turbulence kinetic energy (kw/ m^{2}K)		Contours of wall function heat transfer coefficient (kw/ m^{2}K)		Contours of skin friction coefficient		Contours of static pressure (M Pascal)		Contours of enthalpy (KJ/kg)	
	Minimum	Maximum	Minimum	Maximum	Minimum	Maximum	Minimum	Maximum	Minimum	Maximum	Minimum	Maximum
Parallel flow with 4 baffles	22.85	206.85	87.10	435	11.10	110	0.188	1.88	18.70	40.70	1.88	23.5

5. CONCLUSIONS

In this investigation, the overall heat transfer coefficient for parallel flow for shell and tube heat exchanger were investigated numerically. the process with constant flow rate of hot and cold water as 0.028 and 0.021 respectively and inlet temperature of hot and cold water as 50°C and 32°C, for parallel and counter fluid flow, with and without baffles was studied on ANSYS software to validate the design. The results are as follows:

The overall heat transfer coefficient for parallel flow obtained numerically by LMTD and NTU method for all the readings, is maximum, for flow rate of hot and cold water of 0.022 and 0.0228 and inlet temperature of 50°C and 31°C for hot and cold water respectively, was 575.92 [W/m²K] by NTU method.

The analysis performed on ANSYS for one reading,

i.e. flow rate of 0.028 and 0.021 of hot and cold water, for parallel and counter flow without baffles, gives the overall heat transfer coefficient as

The equation for overall heat transfer coefficient is as follows:

$$\frac{1}{U} = \frac{1}{h_h} + \frac{1}{h_c} + R_f$$

Where,

h_h = hot side heat transfer coefficient [W/m²K] h_c = cold side heat transfer coefficient

[W/m²K] Rf = Fouling coefficient [W/m²K]

Fouling coefficient is neglected as it does not show any significant effect on the value.

141.673[W/m²K] and 73.121[W/m²K] for parallel and counter flow respectively.

The results obtained for parallel and counter flow with 2 baffles with same boundary conditions are 105.172[W/m²K] and 78.60[W/m²K] for parallel and counter flow respectively.

For 4 baffles configuration with same boundary conditions, the result for parallel flow was 106.51[W/m²K] and 71.93[W/m²K] for counter flow.

From the above obtained results it can be seen that the maximum overall heat transfer coefficient is for parallel flow without baffles configuration i.e. 141.673[W/m²K]

The addition of baffles in the existing model did not showed any positive results. The overall heat transfer coefficient decreased as number of baffles increased for the same boundary condition for both parallel and counter flow. comparing the results of numerical calculation and simulation result it can be seen that the value is greater for numerical calculation i.e. by NTU method.

Hence it can be concluded that if the boundary condition of parallel flow without baffles is taken for other different configuration, it will not increase the overall heat transfer coefficient and also find contours of parallel flow heat exchanger with 4 baffles, Contours of static temperature 206.85°C, Contours of turbulence kinetic energy 435 kw/ m^2K, Contours of wall function heat transfer coefficient 110 kw/ m^2K, Contours of skin friction coefficient 1.88, Contours of static pressure 40.70 M Pascal, Contours of enthalpy 23.5 KJ/kg.

REFERENCES

[1] "Plate Heat exchangers". Gold-Bar Engineering ltd. Retrieved 30 June 2015.

[2] S. M. Sohel Murshed, Manuel MatosHeat (2017) “Exchangers: Design, Experiment and Simulation”. Croatioa:National and university library in zagreb

[3] Brogan R.J.(2011), heat and mass transfer:” HEAT EXCHANGERS“, thermopedia

[4] Shah, R. K., 1981, Classification of heat exchangers, in Heat Exchangers: Thermal-Hydraulic Fundamentals and Design, S. Kakac¸, A. E. Bergles, and F. Mayinger, eds., Hemisphere Publishing, Washington, DC, pp. 9–46.

[5] Sadik Kakaç & Hongtan Liu (2002)Heat Exchangers: Selection, Rating and Thermal Design (2nd ed.). CRC Press. ISBN 0-8493-0902-6.

[6] "Plate Heat exchangers". Gold-Bar Engineering ltd. Retrieved 30 June 2015

[7] Shah Ramesh K. (2011), heat and mass transfer: “EXTENDED SURFACE HEAT TRANSFER”, University of Kentucky, Lexington. Article: thermopedia

[8] Taborek, J., Hewitt, G.F. and Afgan, N. (1983). Heat Exchangers: Theory and Practice. Hemisphere Publishing Corporation. ISBN 0-07-062806-8.

[9] Young-Seok Son, Jee-Young Shin, 2001, “Performance of a Shell and Tube Heat Exchanger with Spiral Baffle Plates” KSME International Journal, Vol 15, No. 11, pp. 1555-1562

[10] Kuashik Parmar, Kashyap Desai, 2017, “A Practical Attempt to Improve Performance of Heat Exchanger”, IJARIE , Vol-3 Issue-1

[11] Surender Kumar, 2018, “Experimental Analysis of Shell and Tube Heat Exchanger by using ABAQUS”, International Journal of Engineering Science & Research Technology
[12] Hariahchandra Thakur, Rahul Singh, 2016,“CFD Analysis of Shell and Tube Heat Exchanger”, International Journal for Scientific Research & Development, Vol 3, Issue 12
[13] Wenzhu Lin, Qianhao Wang,2018“Experimental and Numerical Investigation On the Novel Latent Heat Exchanger with Paraffin/Expanded Graphite Composite”, Applied Thermal Engineering, volume 144, pages 836-844
[14] Wenzhu Lin, Rui Huang, 2019,“Improvement of Thermal Performance of Novel Heat Exchanger with Latent Heat Storage” International Journal of Heat and Mass Transfer, Volume 140, pages 877-885
[15] Guo-yan Zhou, Ling-Yun Zhu, 2014,“Prediction of Temperature Distribution in Shell and Tube Heat Exchangers” Energy Procedia, Volume 61, pages 799- 802
[16] Sheng Yang, Teng Hu,2015,“CFD Analysis of Heat Transfer and Flow Resistance On Shell Side of the Spiral Elliptical Tube Heat Exchanger”, Proceedings of the 2015 International Forum on Energy, Environmental Science and Materials, pages 2352-5401
[17] Amir Asgari Tahery, Samad Jafarmadar,2019“Effectively Designed NTW Shell Tube Heat Exchanger with Segmental Baffles Using Flow Hydraulic Network Method”, Volume 120, pages 635-644
[18] Ali Akbar Abbasian Arani, Reza Moradi,2019,“Shell and Tube Heat Exchanger Optimization Using New Baffle and Tube Configuration”, Applied Thermal Engineering, Volume 157
[19] A. K. Srivastava, Raj Rajat Verma, 2014“Performance Analysis of Shell & Tube Type Heat Exchanger Under the Effect of Varied Operating Conditions”, IOSR Journal of Mechanical and Civil Enginering,Volume 11, pp 08-17
[20] Gurbir Singh, Hemant Kumar, 2014,“Computational Fluid Dynamics of Shell and Tube Heat Exchanger”, Journal of Civil Engineering and Environmental Technology, Vol 1, Number 3, pp 66-70
[21] Sumit S. Mukadam, Omkar v. Naik, 2018, “Temperature Analysis of Shell and Tube Heat Exchanger”, International Research Journal of Engineering and Technology, Volume-5 Issue 4
[22] Kevin M. Lunsford, 2006,“Increasing Heat Exchanger Performance”, Bryan Reearch and Engineering,Inc
[23] Pranita Bichkar, Tapobrata Dey,2018,“Study of Shell and Tube Heat Exchanger with the Effects of Types of Baffles”, Procedia Manufacturing, Volume 20, pages 195- 200
[24] Vishal H Acharya, 2017,“A review on Investigation of Shell and Tube Heat Exchanger for Different Parameter”, International Research Journal of Engineering and Technology, volume 04, Issue 02
[25] Santhosh Sivan M, Karthikeyan S, 2018, “Thermal Analysis of Shell and Tube Heat Exchanger”, International Journal of Pure and Applied Mathematics, Volume-119 No. 12, pp 14299-14306
[26] N. Prabhu Kishore, N. Alekhya,2016,”Performancce of Shell and Tube Heat Exchanger under Varied Operating Conditions”, International Journal of Innovative Research in Science Engineering and Technology, Vol. 5, Issue 6
[27] Sayali R. Bhandurge, A. M. Wankhade,2016,” Analysis and Experimentation of Shell and Tube Heat Exchanger with Different Orientation of Baffles”, International Journal for Research in Applied Science & Engineering Technology, Volume 4 Issue 6
[28] Saurabh Sharma, Dr. Ritesh Kumar Dewangan,” Performance Analysis of Shell and Tube Heat Exchanger Having Different Baffle Cut At Various Inclination Angles”, International Journal of Latest Engineering and Management Research, Volume 3 Issue 5, pp. 46-51
[29] Navaz I Shaikh, Jignesh Patel,” A REVIEW ON SHELL & TUBE HEATEXCHANGER

BY CHANGING GEOMETRICAL SHAPE OF BAFFLES", International Journal of Advance Research and Innovative Ideas in Education, Vol 3 Issue 1

[30] Sachin k. Patel, A. R. Patel, 2015," REVIEW PAPER ON INVESTIGATION OF PERFOMANCE FOR SHELL AND TUBE HEAT EXCHANGER", International Journal of Advance Engineering and Research Development, Volume 2, Issue 3

Dr. G. Sulochana,
Assistant Professor of Mechanical Engineering,
UCEN-JNTUK,
Narasaraopet,
Palnadu District – 522601.