e-ISSN: 2582-5208
International Research Journal of Modernization in Engineering Technology and Science
(Peer-Reviewed, Open Access, Fully Refereed International Journal)
Volume:06/Issue:04/April-2024 Impact Factor- 7.868 www.irjmets.com

DESIGN AND PERFORMANCE ANALYSIS OF SHELL & TUBE HEAT EXCHANGER HAVING DIFFERENT SHAPES OF TUBES

Deepak Kothari[*1], Abhishek Kumar Jain[*2]

[*1]M.Tech. Research Scholar, Dept. Of Mech. Engg., MATS School Of Engineering & IT, MATS University, Raipur, C.G., India.

[*2]Associate Professor, Dept. Of Mech. Engg., MATS School Of Engineering & IT, MATS University, Raipur, C.G., India.

ABSTRACT

All the components relevant to mechanical engineering need specific working conditions upon which they can work with maximum efficiency. In maximum processes across industries, a lot of energy in form of heat gets wasted that's why certain measures are applied so that some or the most part of this waste heat energy is recovered. In this kind of situation, a specific appliance is used which is known as heat exchanger.

These exchangers receive heat energy from one source and transfer it to another source. Lately researchers have started researching on various aspects through which this process of heat exchange is also optimized. Some of the concerned aspects are discussed in following sections.

Keywords: Heat Exchanger, CFD, LMTD, Heat Transfer.

I. INTRODUCTION

1.1. Heat Exchangers

Heat exchangers can be defined as a setup wherein heat is transferred from one fluid to another fluid. Heat exchangers can be used in both heating & cooling. fluids are separated through solid wall in order to prevent the mixing or in some cases fluids may remain direct contact of each other.

These heat exchangers are mostly used in sewage treatment, natural-gas processing, petroleum refineries, petrochemical plants, power stations, chemical plants, refrigeration, air conditioning & space heating.

Most basic examples of heat exchangers are seen in any IC engine where engine coolant (circulating fluid) flows through a radiator coil and at the same time air flows past these coils.

This way coolant is cooled and incoming air is heated. One more example of the same process can be seen in case of heat sink which is an example of passive type heat exchanger transferring heat to fluid medium normally liquid coolant or air.

- **Types of Flow Arrangements in Heat Exchangers**

On the basis of arrangement of flow, Heat exchangers can be of three types. In the parallel flow arrangement, both the fluids move along in the same direction after entering the exchanger at the same end.

In the counter flow arrangement, both the fluids enter the exchanger from opposite ends & move in opposite directions to each other. This arrangement of counter flow is the best suitable because its efficiency is maximum & the maximum amount of heat is transferred in this type of heat exchangers.

In the third type of flow arrangement which is cross-flow arrangement, both of the fluids enter along perpendicular direction to each other. In order to maximize the efficiency, maximizing the surface area is needed. Using corrugations and fins can achieve this task.

Efficiency of heat exchangers is measured using LMTD which is known as "log mean temperature difference". But sometimes NTU method is also used when LMTD has certain limitations

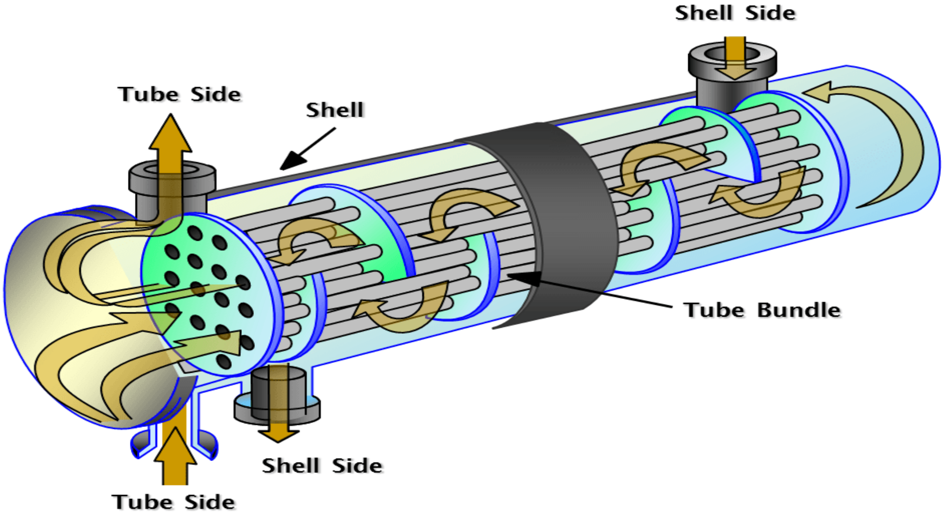

Fig. 1: A simple arrangement of Heat Exchanger.

1.2. Heat Exchanger Types

One of the simplest types of heat exchangers are of Double pipe types. These heat exchangers are cheaper (from the perspective of maintenance & design) as compared to other but because of their low efficiency, other types of heat exchangers are being used by companies now a days. Double pipe heat exchangers are basically used to for demonstration & educational purpose more.

- **Double-pipe heat exchanger**

(a) When the other fluid flows into the annular gap between two tubes, one fluid flows through the smaller pipe. The flow may be a current flow or parallel flow in a double pipe heat exchanger.

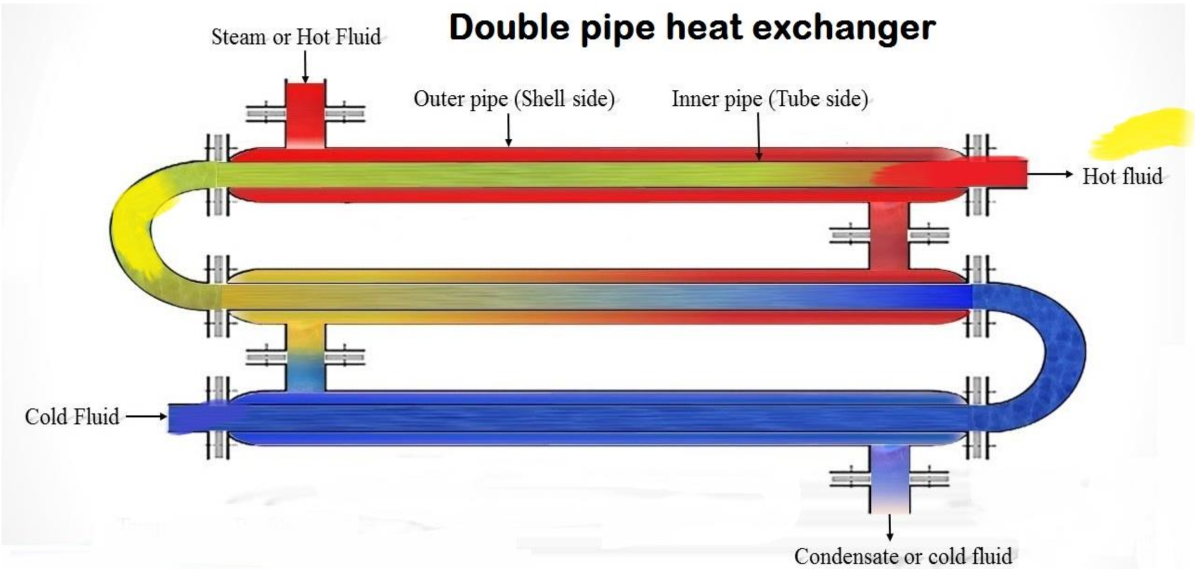

Fig. 2: Double Pipe Heat Exchanger

(b) Parallel flow, where both the fluids enter the exchangers through same corners, flow in the same direction and leave at the other corner.

(c) Counter flow, where at opposite ends, hot and cold fluids join, flow in the opposite direction and exit at opposite ends.

Comparatively, counter flow arrangement causes more heat transfer from one fluid to another. Parallel flow arrangements reveal noteworthy disadvantages because of temperature differential caused by this arrangement. Parallel flow arrangements are of disadvantage if primary need is to raise the temperature of cold fluid. Counter flow arrangement can serve better in this aspect when raising the temperature of cold fluid is of primary requirement. Counter flow has more advantages as compared with parallel flow. Counter flow

e-ISSN: 2582-5208
International Research Journal of Modernization in Engineering Technology and Science
(Peer-Reviewed, Open Access, Fully Refereed International Journal)
Volume:06/Issue:04/April-2024 Impact Factor- 7.868 www.irjmets.com

arrangement reduces the amount of thermal stresses & achieves more uniform rate of heat transfer across the tubes.

- **Shell-and-tube heat exchanger-** Basic components of this heat exchanger are tube box, baffles or fins, end headers & shell. The baffles support the tubes, as well as used to direct the flow of fluid through the tubes naturally and it is also used to maximize turbulence of the fluid contained by the shell. Several types of baffles can be used in shell & tube heat exchangers. Type of baffles, spacing between baffles & geometry of the baffles depend on force required at shell side. Numerous variations of these exchangers are available which depends on the type of flow.

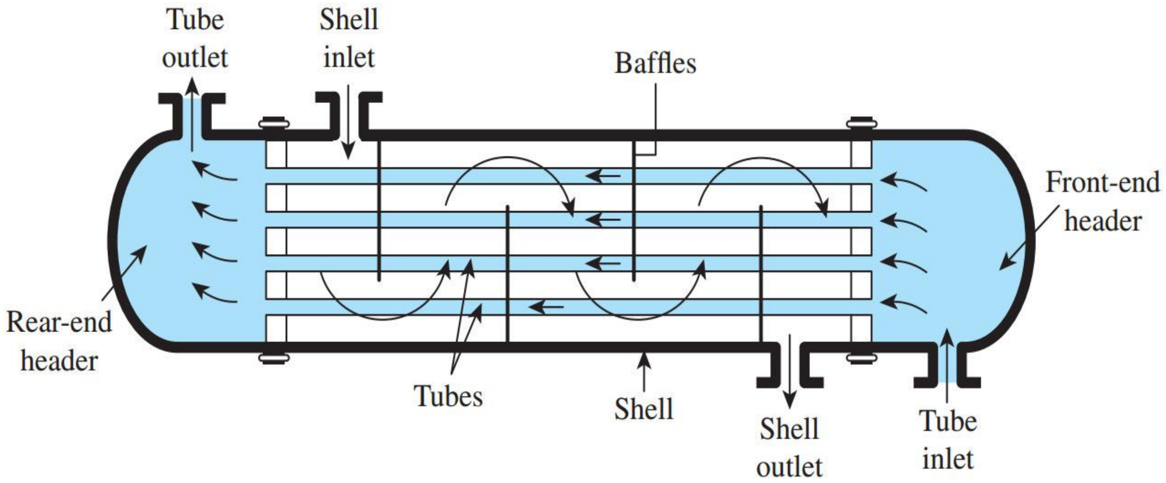

Fig. 3: Shell & Tube Heat Exchanger

- **Plate Heat Exchanger-** This type of heat exchangers includes thin plate which are bundled with one another. Every pair of plates provides separate channels and the fluid flows through these channels. These pairs are attached with each other through bolts or welding.

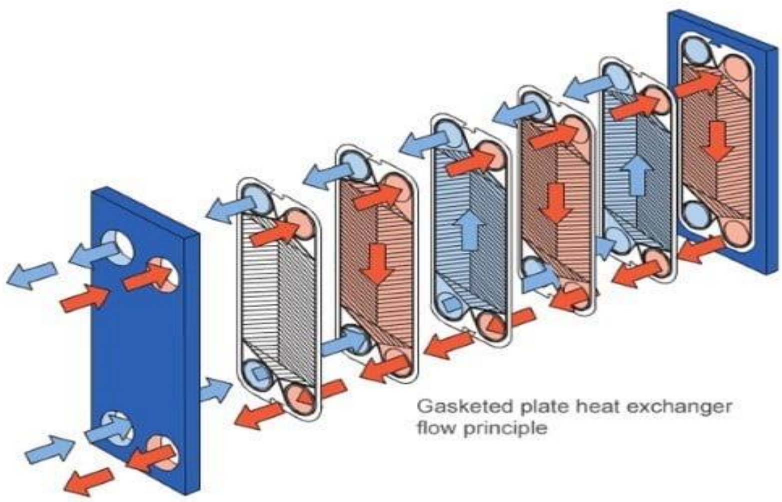

Fig. 4: Plate Heat Exchangers

1.3. Shell and tube heat exchanger

These types of heat exchangers contain a number of tubes inside a hollow shell. Tubes and shell both contain hot and cold fluid respectively & vice versa. Set of the tubes is known as Tube Bundle & their cross-sectional area can be of various types. These tubes can be longitudinally finned or plain. These heat exchangers are used for high pressure applications.

The tubes in any shell & tube heat exchangers are designed after considering numerous features of thermal design. Normally the ends of tubes are connected with water boxes which are also known as plenums. The tubes may be straight or bent in form of English letter “U” which is called U tubes.

- **Diameter of the Tube-** When small diameter tubes are used in the heat exchangers, it makes the exchanger both compact & economical. Although this small diameter tube causes fouling of the tubes quickly & because of these small diameters, tube cleaning also becomes difficult. For this reason, larger diameter tubes are generally preferred but one cannot go for larger diameters and hence several factors decide the diameter of the tube & some of them are fouling nature, cost & available space
- **Tube thickness:** thickness in any tube is provided to assure-
- corrosion
- Buckling strength
- vibration resistance
- Hoop strength
- spare parts availability
- Axial strength
- **Length of the Tube:** longer tube length & smaller shell diameter normally leads to cheaper heat exchangers. Hence practically these heat exchangers are very long in size under capabilities of production. But space availability also decides the length of heat exchanger. Hence several criteria are to be followed when developing the heat exchangers.
- **Pitch of the Tube:** Pitch of the tube may be defined as the centre-to-centre distance between two tubes. Tube pitch should not be less than 1.25 times the outside diameter of the tubes. Larger pitch of tubes causes larger overall diameter of the shell. When the shell diameter increases, overall cost of heat exchanger is increased and the exchanger becomes expensive.
- **Corrugations in the Tube:** The corrugations in the tubes cause more turbulence in the fluids & this is an important impact in the process of heat transfer which provides better overall performance.
- **Layout of the Tube:** Tube layout means how the tubes are being positioned in side shell. Basically, four types of layouts of tube positioning are followed & these are, rotated square (45°), triangular (30°), square (90°) & rotated triangular (60°). triangular pattern provides greater transfer of heat. In case of high fouling being experienced, square patterns are employed.
- **Design of Baffle:** Numerous types of baffles are employed inside the shell which primarily looks after the direction of fluid flow across the bundle of tubes. These normally run in a direction perpendicular to the shell. These baffles also stop the tubes from vibration.

1.4. Performance Parameters of Heat Exchangers

- **NTU-** Number of transfer units are a dimensionless quantity and is an indicator of size of heat transferring areas of heat exchanger. Large value of NTU means larger heat transferring area.
- **LMTD-** LMTD method is used when temperatures of both the fluids at inlet and outlet of the heat exchanger are known. Performance analysis by this method is carried out by making following assumptions: i) Overall heat transfer coefficient, U remains constant along the length of heat exchanger. ii) Specific heats and mass flow rates of both the fluids remain constant. iii) Heat exchanger is perfectly insulated and no loss of heat occurs
- **Effectiveness-** Effectiveness of a heat exchanger may be defined as the ratio of actual amount of heat transferred to the maximum possible amount of heat which can possibly be transferred.

II. LITERATURE REVIEWS

Abhishek Agarwal (2021) investigated the application of a plate heat exchanger in case of cooling of engine oil. Heat exchangers are used for various industrial application for transfer of enthalpy from hot fluid to cold. One of them is Plate Heat Exchanger which finds its application in evaporating systems. The compactness, high effectiveness and easy maintenance of Plate Heat Exchanger makes it best choice for process industries. The current research investigates the application of Plate Heat Exchanger in cooling of engine oil using techniques of Computational Fluid Dynamics for low, medium and high Reynolds number using ANSYS CFX software. The CAD model is developed using Creo design software and turbulence model used for analysis is RNG k-epsilon which gives good predictions for complex flows involving swirls. The CFD analysis is conducted for different values of Reynolds number. The temperature distribution, effectiveness and overall heat transfer coefficient is determined for different values of Reynolds number.

Ahmad Hanan et al. (2021) conducted a performance analysis of a heat exchanger. Heat exchangers have found extensive applications in the engineering sector owing to the crucial need for heat transfer and temperature regulation in the process industry. The performance analysis of a heat exchanger can either be investigated experimentally or utilizing computational fluid dynamics (CFD). The simulation of flow conditions based on CFD principles is considered as a contemporary approach for conducting performance analysis and design optimization of the heat exchanger. The paper aims to present how the performance optimization parameters of shell and tube heat exchanger effect its performance in terms of the extent of heat transfer achieved and fall in the shell outlet temperature. A standard model (Model 1) was initially defined, and seven subsequent models were created with one varying parameter as compared to Model 1 to carry out the comparative analysis. The results showed that decrease in inlet velocity, increase in thermal conductivity of the heat exchanger material, reduction in baffle spacing and use of triangular tube bundle arrangement has significant influence on the reduction of the condensate temperature. The effects of the same parameters on the temperature distribution and heat transfer rate have also been discussed.

Liliana Dumitrescu et al. (2021) conducted a CFD (Computational Fluid Dynamics) analysis on a heat exchanger of gas-liquid type which is fitted on gasification component. In order to analyze the heat exchanger, Solidworks software is employed which can be used to study the flow simulation problems. Temperature & flow patterns have been drawn with the help of this software in order to study the current research. The system transfers heat from flue gases which have been generated by gasification system to the liquid medium. This is further sent to a secondary heat exchanger which is used to heat the water from boilers.

P.S. Aswin et al. (2021) analysed the shell & tube type heat exchanger with the help of ANSYS. Here tubes are placed inside the shell. Hot water flows through tubes and cold-water flows through the shell. In order to analyze the system stream line flow, velocity & heat transfer through the fluids is being studied. Two different cases have been considered in this analysis. One case being the heat exchanger along with bafflers & the other case being the heat exchanger without bafflers. Certainly, the system where baffles are employed along with heat exchangers provides greater amount of heat transfer.

Perone C. et al. (2021) conducted a CFD analysis on a tubular type heat exchanger to be used for the conditioning of olive paste so that the process of extraction of olive oil is improved. This analysis as intended for better understanding of impact of inlet situations of olive paste on hydrodynamic and thermal behaviour. The CFD analysis is performed on Solidworks-2016. Tube in tube model of heat exchanger has been used in the current analysis here inner tube contains olive paste & jacket contains hot water. Pressure drop and heat transfer were calculated by varying mass flow rate & the temperature of inlet.

S. Elangovan et al. (2021) conducted an CFD analysis of a monolithic type heat exchanger so that heat transfer is increased by varying material & shape of air passage. Different shapes of air passage used are oval, hexagonal & circular. Materials used are $CrCO_3$, Al_2O_3 & SiC ceramic materials. ANSYS Fluent is used for analysis purpose after modelling the heat exchanger in CATIA.

Shubham Sharma et al. (2021) compared the performance of heat transfer for two STHXs (Shell & tube heat exchangers) having different arrangements of tubes. 21 & 24 tubes ere arranged in staggered grid and inline arrangement. Staggered arrangement is seen to be providing lesser amount of thermal stratification. Mass flow rate of shell side liquid is varied from 0.5 kg/s to 0.1 kg/s and tube side liquid mass flow rate is kept constant at 0.25kg/s. Pressure drop, effectiveness & heat transfer efficiency have been analysed. 0.1 kg/s of mass flow rate causes minimum pressure drop

Ahmet AYDIN et al. (2020) conducted an optimization of heat exchanger based on total cost (initial costing as well as operation costing). After optimization, flow analysis has been conducted to predict temperature pattern & flow field. CFD analysis is performed for flow analysis. Experimental as well as CFD analysis results are compared with each other. It has been confirmed that baffles significantly affect the overall performance. New baffle design creates lower pressure drop.

G. Nivedini et al. (2020) compared the experimental & CFD analysis results when nanofluids are used in heat exchangers so that transfer of thermal energy is improved across the exchanger. Silica nanoparticles are used in the current research. Only the conventional fluid is replaced by nano particles. A double pipe heat exchanger is used and CFD analysis is conducted in ANSYS. 10°C improvement in temperature is achieved when silica nano particles are used in place of conventional fluid.

Mahendra Kumar Verma et al. (2020) evaluated the thermal performance for an EAPHE (Earth-Air-Pipe-Heat-Exchanger) in transient condition when helical air pipes are used. Thermal performance of the system weakens in long term because of sub soil saturation. CFD analysis is conducted using ANSYS Fluent 19.0. Aluminium water pipe arrangement represents continuous effectiveness of 0.76 after running the heat exchanger for 36 hours.

Md Imran Pasha et al. (2020) studied about double pipe heat exchangers for different materials. Current research specifically focused on the heat exchangers used in cryogenic applications. Heat exchangers have been designed in CATIA, CFD analysis has been performed in ANSYS. Heat exchangers have been analysed for different materials like copper, aluminium & steel.

Ram Kishan et al. (2020) examined the temperature and flow fields for box and tube heat exchangers wherein different types of tube patterns have been used. ANSYS 14.0 software is used for analysis conduction. Three different types of heat exchangers are used in the current study. Impact of tube width on temperature has been discussed using CFD analysis.

Shuvam Mohanty et al. (2020) investigated a STE (Shell & Tube Heat Exchangers) for effectiveness having 50% baffle cuts & varying the number of baffles. Water has been used as the working fluid in the current research. Counterflow arrangement has been used for flow of fluids. ANSYS 14.0 has been used to conduct the CFD analysis. Various input parameters to be used are baffle spacing, baffle cut & their impact on pressure drop & heat transfer has been studied. In this investigation, a comprehensive approach is established in detail to analyse the effectiveness of the shell and tube heat exchanger (STE) with 50% baffle cuts (Bc) with varying number of baffles. CFD simulations were conducted on a single pass and single tube heat exchanger (HE) using water as working fluid. A counterflow technique is implemented for this simulation study. Modified model design yields a 44% of reduction in pressure drop & 60.66% increase in heat transfer.

P.C. Mukesh Kumar et al. (2019) investigated the heat transfer and pressure drop for a double helical coiled tube heat exchanger which handles water/MWCNT nanoparticles. For analysis purpose ANSYS 14.5 has been employed. Analysis has been conducted for a range of Dean number of 1300-2200 under the conditions of laminar flow. Heat exchanger has been modelled in Creo 2.0. nanofluids at different volume concentrations of 0.6%, 0.4% & 0.2 % have been taken into consideration. Simulation data has been compared with experimental results. At 1400 Dean number, 0.6% volume fraction has 30% larger Nusselt number than water.

Yuvraj Singh Rajput et al. (2019) investigated about temperature gradient & heat transfer in a heat exchanger having counterflow arrangement for various fin thickness. CFD analysis has been conducted on the basis of FVM (Finite Volume Method). Inlet air velocity has been taken as 4, 3, 2& 1 m/s & volume flow rate has been kept at 6, 5, 4, 3 & 2 L/min. Temperature of tube side fluid has been kept at 80, 70, 60 & 50°C. Results reveal that for a 3 mm fin heat transfer increases by 15%.

Palash Goya et al. (2018) analyzed heat transfer for a tube in tube helically coiled type heat exchanger. Fluid in both types of tubes have been varied in order to observe the effect on heat transfer of these type of heat exchangers. Helical tubes are constructed by copper & the fluid has been taken as water. Total heat transfer has been evaluated with the help of CFD analysis being conducted through ANSYS Fluent 16.0.

III. METHODOLOGY

3.1 Introduction

In the previous chapter Statement of the problem to be solved was defined. Problem identification is the most important aspect of any research study so that the research study undertaken is streamlined & the area under research is confined in the most suitable manner. After defining the problem, the current chapter focuses on analyzing the heat exchanger through CFD analysis & comparing the results for different cross sections of tubes. Following sections describe the steps of analysis.

3.2 ANSYS 18

ANSYS is considered to be a FEA software package. It consists of a pre-processor portion which is used to develop the geometry, apply the material, create a co-ordinate system, maintain contacts between the parts as and when required and to develop the mesh according to the need. After this it consists of a solution program also known as a solver of ANSYS in order to apply the loads & boundary conditions on the meshed model. At last, after solving the problem through its solver it provides the outputs in form of the results desired. These results obtained are then analyzed to evaluate the behavioral changes in the component. FEA is employed almost across all designs related to engineering.

Hence the sequence of operations to be performed by ANSYS consists of-

- Pre-Processing
- Solver
- Post-Processing

- **Pre-processing**

It is the first step of ANSYS workbench in analysis of Stirling engine with regenerator. A user can use whichever software He/She wants to in order to develop the model of the component. After the creation of model in modelling softwares, this model needs to be imported to ANSYS & for this purpose, the model is converted into. igs or .stp format. After importing the geometry, one needs to finalize the materials which will be used in the analysis and for that there is a large engineering material database provided by the ANSYS. So, the need now is to choose the suitable material. After applying the material there is need to place a coordinate system into the component although ANSYS provides a default coordinate system. So, if one needs to place another coordinate system, he or she is completely free to do so. After this contact conditions among different contacting surfaces are placed. Now one looks for the meshing of the component. Basically, to create mesh means to divide the component in several small blocks which may be one dimensional, two dimensional or three dimensional. These blocks have two components- nodes & elements. Elements are the outlines of the components & nodes are the points at which various elements meet each other. The process of mesh generation is also called grid generation.

- **Solver**

In the previous phase one needs to toil hard whereas this step is now the headache of the system to perform task at hand. One just needs to click the solve button. The software performs the necessary work internally and gives its output in form of the result desired by the user.

- **Post-processing**

Last phase in ANSYS is called as post-processing. In this phase, one needs to analyze the results obtained. ANSYS only helps the user to obtain accurate results & all the analysis work is done by the user keeping in mind the values obtained by the ANSYS analysis.

- **Assumptions**
  - Software Used is ANSYS 18
  - Mesh method to be used is default mesh of ANSYS 18.
  - We will perform Fluid Flow analysis using ANSYS Fluent.

3.3 CFD Analysis of Heat Exchanger

After creating solid model of components of heat exchanger in Solidworks-2020, proceed as follows-

- Start the analysis process of the model in ANSYS Workbench through selecting the module of CFD analysis through Fluid Flow (Fluent).
- Select the Shell & Heat Exchanger material according to the requirement. Insert the material properties as follows-

e-ISSN: 2582-5208
International Research Journal of Modernization in Engineering Technology and Science
(Peer-Reviewed, Open Access, Fully Refereed International Journal)
Volume:06/Issue:04/April-2024 Impact Factor- 7.868 www.irjmets.com

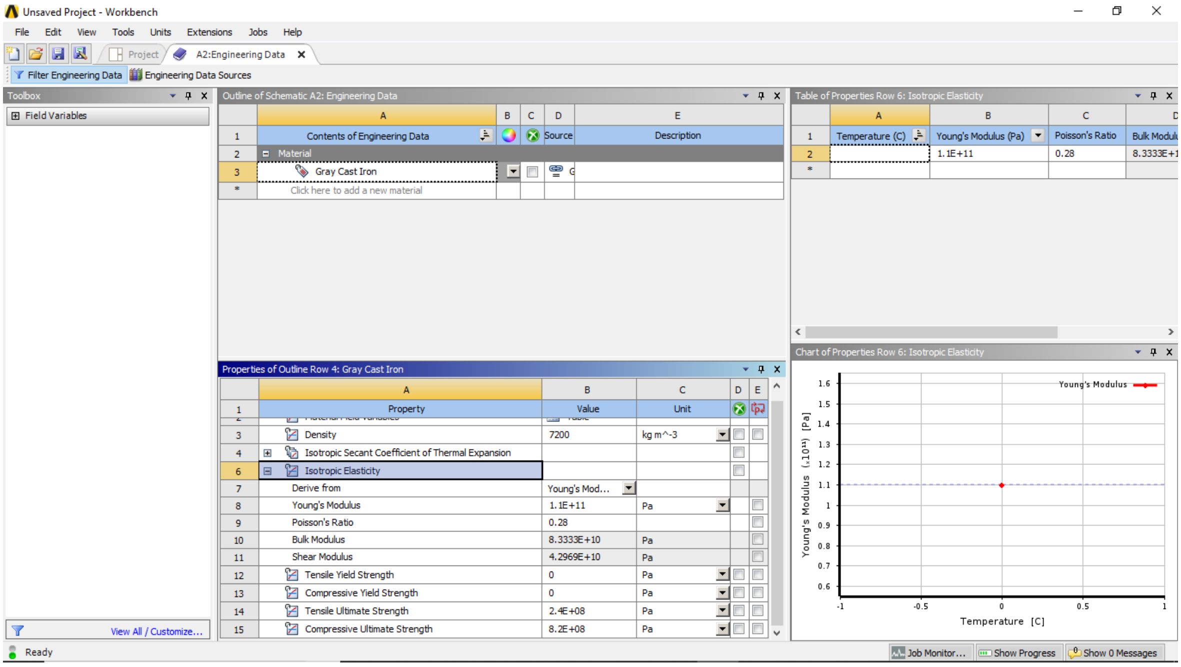

Fig. 5: Engineering data source in ANSYS Workbench

➢ Create meshing of Shell & Tube Heat Exchanger.

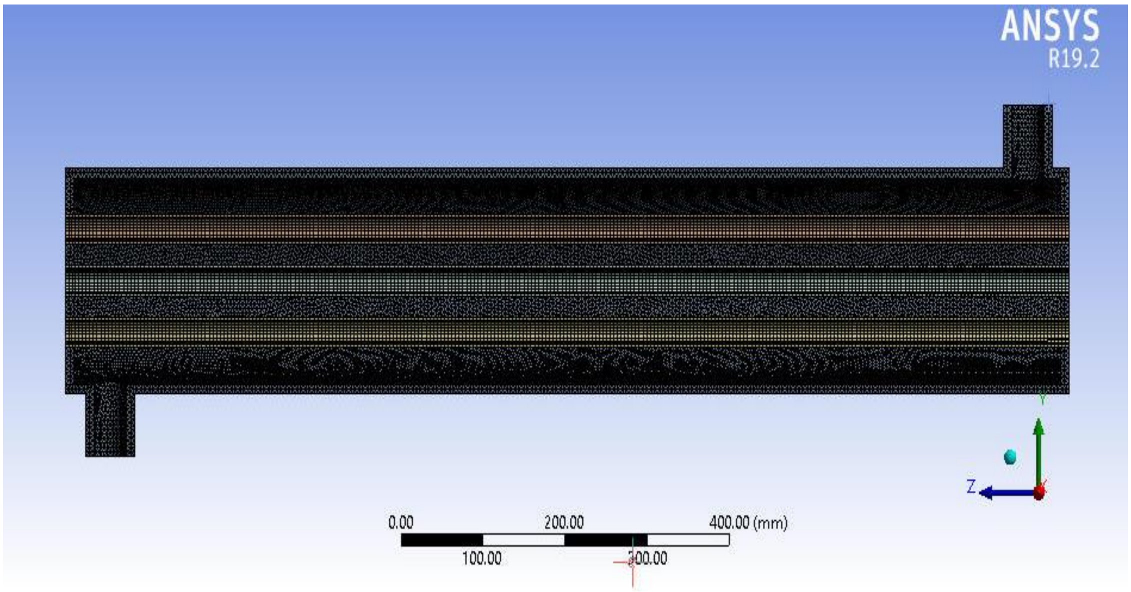

Fig. 6: Meshing of the heat exchanger

After applying the material to the part, the process of meshing is performed in order to convert the component into large number of small parts which are either one dimensional, two dimensional & three dimensional.

➢ Defining different Reports

Primary aim of the current study is to determine the temperature distribution for all the cross-sections of tubes and selecting the best feasible solution from the result set. For the purpose average temperature across all the components.

➢ Initialization

After report definition, solution initialization is performed. Standard initialization method has been selected in current analysis and reference values have been calculated from inlet.

➢ Running the Calculations

IV. RESULTS AND DISSCUSION

4.1 Data Obtained from the CFD Analysis

CFD analysis of the current Shell & Tube heat exchanger setup is conducted for all the tubes.

Various contours of Temperature distribution for different cross sections are presented here for reference purpose-

- Temperature Distribution for Circular Tube

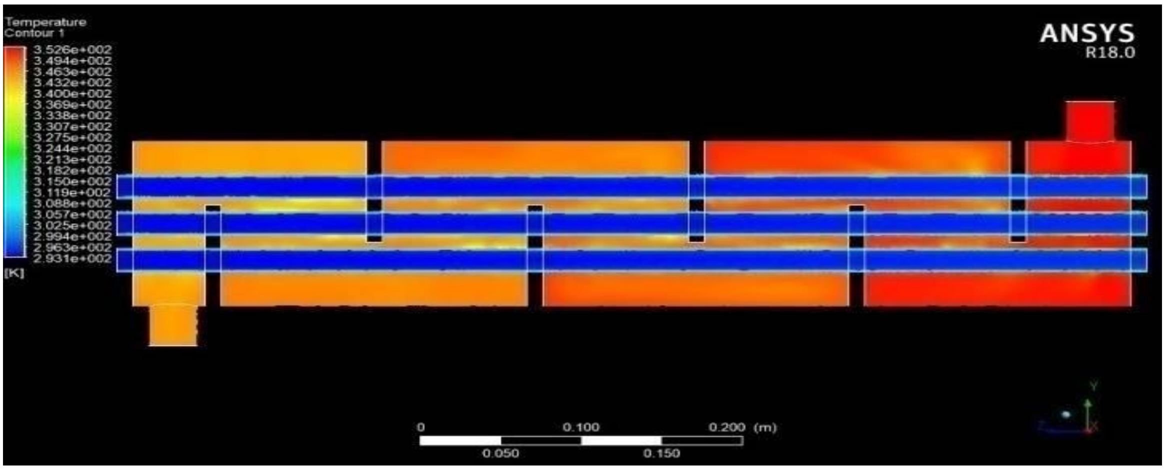

Fig. 7: Temperature Distribution for Circular Tube

- **T**emperature Distribution for Square Tube

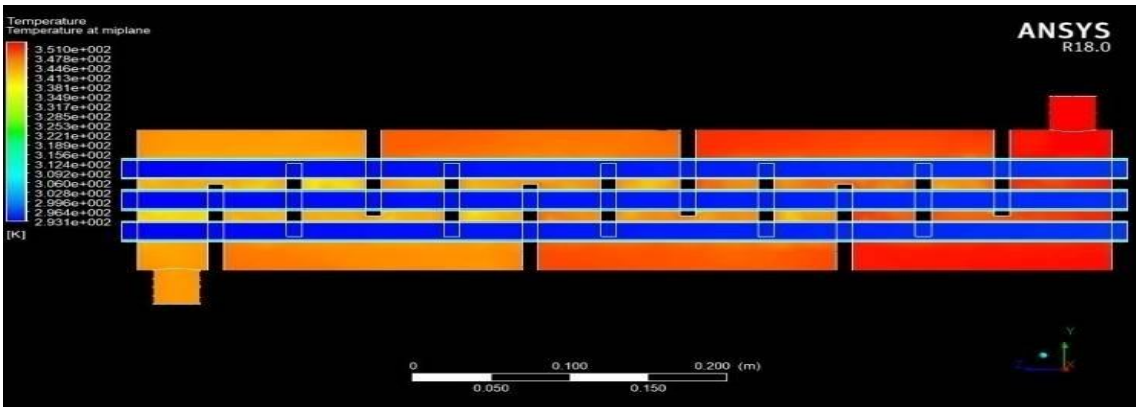

Fig. 8: Temperature Distribution for Square Tube

- Temperature Distribution for Pentagonal Tube

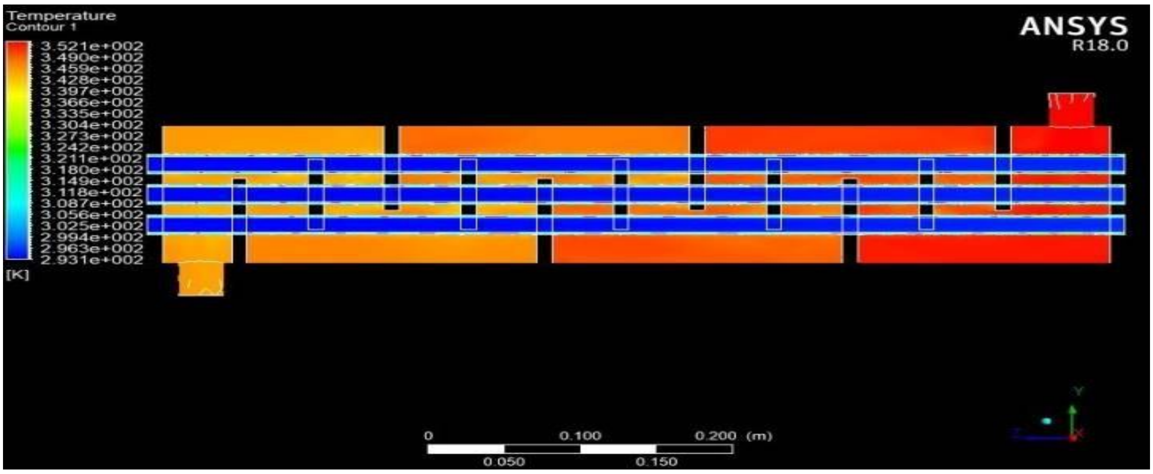

Fig. 9: Temperature Distribution for Pentagonal Tube

IRJMETS

e-ISSN: 2582-5208

International Research Journal of Modernization in Engineering Technology and Science

(Peer-Reviewed, Open Access, Fully Refereed International Journal)

Volume:06/Issue:04/April-2024 Impact Factor- 7.868 www.irjmets.com

- Temperature Distribution for Hexagonal Tube

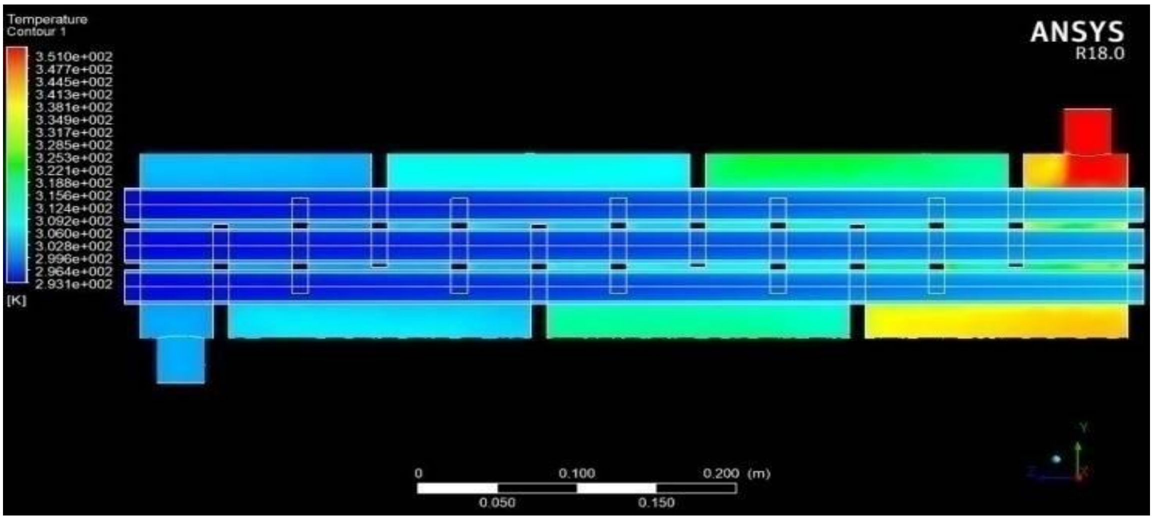

Fig. 10: Temperature Distribution for Hexagonal Tube

- Temperature Distribution for Octagonal Tube

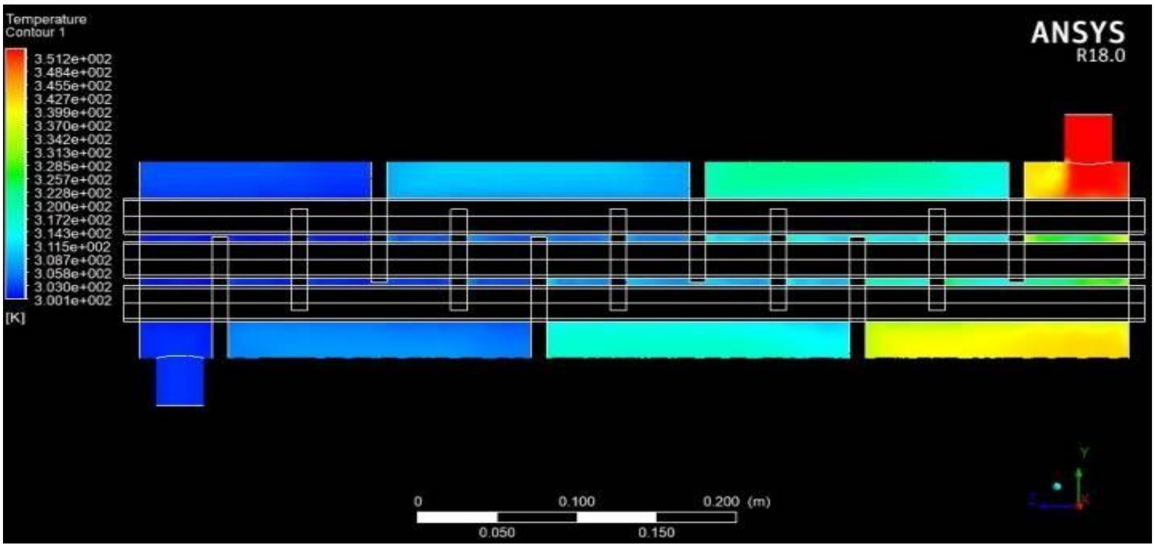

Fig. 11: Temperature Distribution for Octagonal Tube

In order to evaluate T_{CO}, Effectiveness & heat transfer is to be calculated. Formulae to be used are as follows-

- Heat Transfer - $M_H * C_{PH} * (T_{H1}-T_{H2})$
- Effectiveness- $[M_H * C_{PH} * (T_{H1}-T_{H2})] / [M_H * C_{PH} * (T_{H1}-T_{C1})]$

Results obtained through the analysis are presented here for reference-

Table 4.1: Result Set for the Heat Exchanger

Sr. No.	Tube	Hot Fluid		Cold Fluid		Heat Transfer (kJ/s)	Effectiveness
		T_{H1} (K)	T_{H2}(K)	T_{C1}(K)	T_{C2}(K)		
1.	Circular Tube	353	345	293	295.327	23.4192	0.1333
2.	Square Tube	353	345.127	293	295.753	23.0474	0.1312
3.	Pentagonal Tube	353	345.203	293	298.154	22.8249	0.1299
4.	Hexagonal Tube	353	305.374	293	302.522	139.4203	0.794
5.	Octagonal Tube	353	301.197	293	306.877	151.6481	0.8634

IRJMETS

e-ISSN: 2582-5208

International Research Journal of Modernization in Engineering Technology and Science

(Peer-Reviewed, Open Access, Fully Refereed International Journal)

Volume:06/Issue:04/April-2024 Impact Factor- 7.868 www.irjmets.com

4.2 Data Analysis & Interpretation

In order to compare the cross sections of tubes used in the current research study, the data of the above Table 5.1, is presented in form of charts. Two separate charts are prepared for the purpose which are presented as follows-

- Heat Transfer for different Cross sections

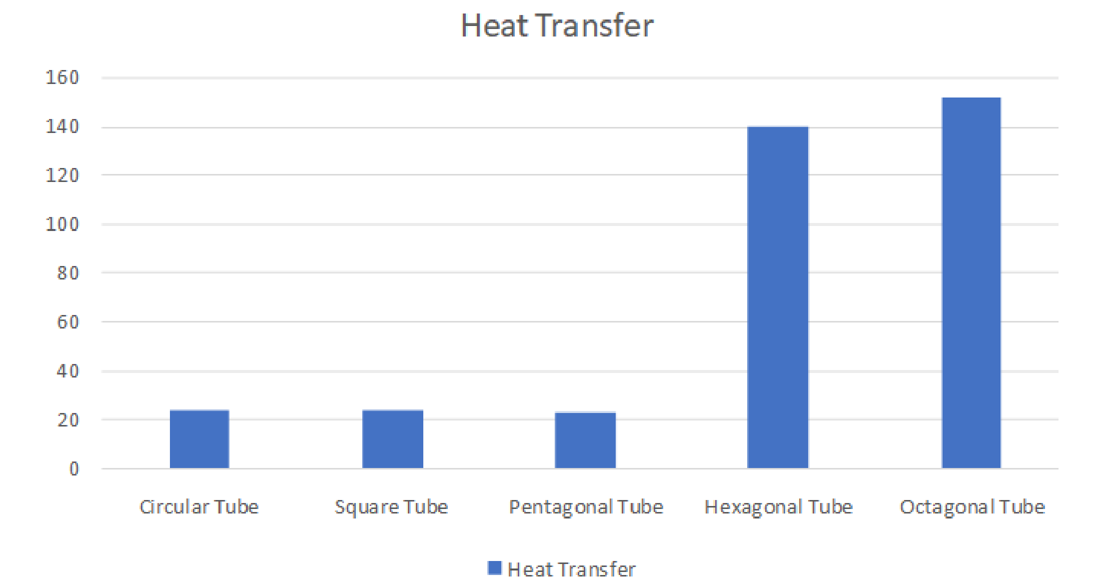

Fig. 12: Heat Transfer Distribution for different cross sections

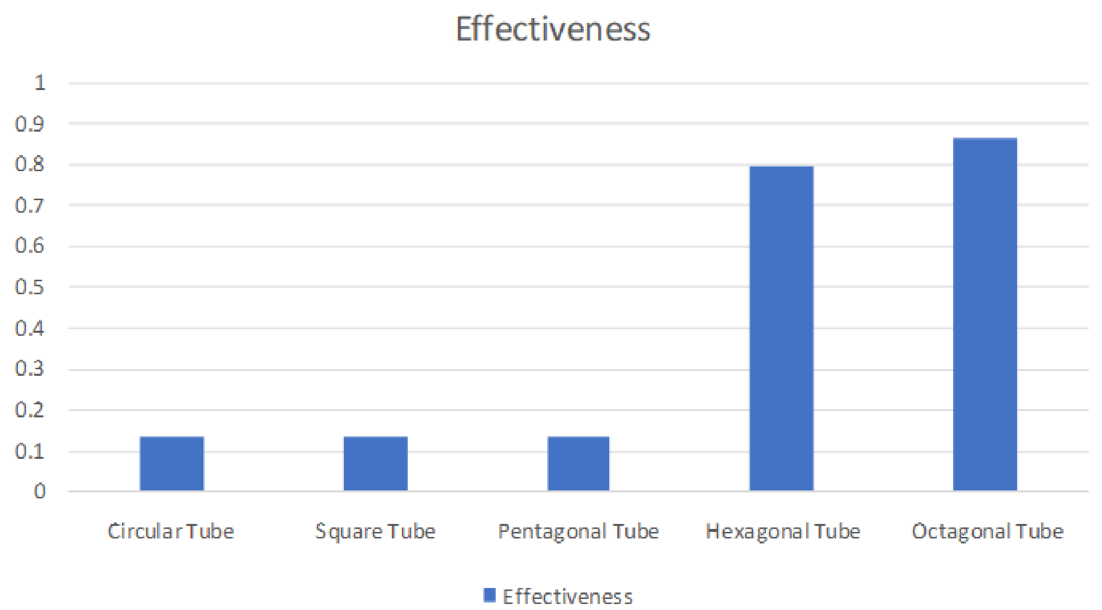

Fig. 13: Effectiveness Distribution for different cross sections

Chart for heat transfer across the heat exchanger for different cross sections is presented in the above chart. It is evident from the chart that heat transfer increases as the number of sides increases for any tube. It can be concluded by the above chart that Surface area of the tubes is directly proportional to heat transfer. Hence more the number of sides in any tube, more will be the heat transfer across the heat exchanger.

- Effectiveness for Different Cross-sections

Chart for effectiveness across the heat exchanger for different cross sections is presented in the above chart. It is evident from the chart that effectiveness is also increasing as the number of sides increases for any tube. It can be concluded by the above chart that Surface area of the tubes is directly proportional to effectiveness. Hence more the number of sides in any tube, more will be the effectiveness of the heat exchanger.

V. CONCLUSION

Current research study is focused on finding the optimum cross section of the tubes to be used in Shell & Tube heat exchangers so that the performance of the heat exchanger is optimized. Performance optimization here means maximum Heat Transfer & Effectiveness. Five different cross sections for the tubes are employed in the current research namely, Circular Tube, Square Tube, Pentagonal Tube, Hexagonal Tube & Octagonal Tube. All these tubes are compared on the basis of two performance parameters namely Heat Transfer & Effectiveness. Conclusions from the current research are as follows-

- As far as Heat Transfer is concerned, surface area is directly proportional to the amount of heat transfer across the Heat Exchanger. On the basis of this criteria, Octagonal cross section causes maximum Heat Transfer of 151.6481 kJ.
- When talking about Effectiveness of the Heat Exchanger, surface area is directly proportional to the Effectiveness of the Heat Exchanger. On the basis of this criteria, Octagonal cross section causes maximum Effectiveness of 0.8634

Both of the above criterion reveals that Tubes having octagonal cross-section optimizes the performance of Shell & Tube Heat Exchangers.

VI. REFERENCES

[1] Agarwal, A. (2021). Modelling & numerical investigation of the effectiveness of plate heat exchanger for cooling engine oil using ANSYS CFX. International Journal of Heat and Technology, Vol. 39, No. 2, pp. 653-658. https://doi.org/10.18280/ijht.390237

[2] Amol Niphade, Prof. H. A. Chavan, Swapnil.S. Kulkarni (2015). "Design & CFD Analysis of High-Volume Heat Exchanger for Dairy Application", International Journal of Advanced Engineering Research and Studies, E-ISSN2249–8974, IV/IV/July-September,2015/13-15

[3] Arpit Thakur, Aashish Sharma. "CFD Analysis of Earth-Air Heat Exchanger to Evaluate the Effect of Parameters on Its Performance", IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE), e-ISSN: 2278-1684, p-ISSN: 2320-334X, National Conference on Advances in Engineering, Technology & Management (AETM'15)" PP 14-19.

[4] Aslam Bhutta, M. M., Hayat, N., Bashir, M. H., Khan, A. R., Ahmad, K. N., & Khan, S. (2012). CFD applications in various heat exchangers design: A review. Applied Thermal Engineering, 32, 1–12. https: //doi.org/10.1016/j.applthermaleng.2011.09.001

[5] Avinash D Jadhav, Tushar A Koli (2014). "CFD Analysis of Shell and Tube Heat Exchanger to Study the Effect of Baffle Cut on the Pressure Drop", International Journal of Research in Aeronautical and Mechanical Engineering, Vol.2 Issue.7, July 2014, Pp: 1-7.

[6] Avinash P. Shinde, Mahadev L. Shinde, Sagar S. Yadav, Manasvi R. Surve, A. S. Futane, Mainak Bhaumik (2018). "Performance Analysis of Heat Exchanger using CFD", International Journal of Innovative Research in Science, Engineering and Technology, Vol. 7, Issue 3, March 2018, DOI: 10. 15680/IJIRSET. 2018. 0703072

[7] Aydin, Ahmet & Yasar, Halit & Engin, Tahsin & Büyükkaya, Ekrem. (2020). Optimization and CFD analysis of a shell-and-tube heat exchanger with a multi segmental baffle. Thermal Science. 293-293. https://doi.org/10.2298/TSCI200111293A

[8] C Rajesh Babu and Santhosh Kumar Gugulothu (2015). "CFD Analysis of Heat Transfer Enhancement by Using Passive Technique in Heat Exchanger", International Journal of Recent advances in Mechanical Engineering (IJMECH) Vol.4, No.3, August 2015.

[9] G. S. L. Swathi, V. Subrahmanyam (2014). "CFD Analysis of a Pipe in a Heat Exchanger", International Journal of Engineering Research & Technology (IJERT) IJERT, ISSN: 2278-0181, www.ijert.org, Vol. 3, Issue 7, July – 2014.

[10] Gurbir Singh, Hemant Kumar (2014). "Computational Fluid Dynamics Analysis of Shell and Tube Heat Exchanger", Journal of Civil Engineering and Environmental Technology, Print ISSN: 2349-8404; Online ISSN: 2349-879X; Volume 1, Number 3; August, 2014 pp. 66-70.

[11] Hanan, A., Zahid, U., Feroze, T., & Khan, S. Z. (2021). Analysis of the performance optimisation parameters of shell and tube heat exchanger using CFD. Australian Journal of Mechanical Engineering, 1–14. https://doi.org/10.1080/14484846.2021.1914890

[12] K. Ravikumar, Ch. Naga Raju, Meera Saheb (2017). "CFD Analysis of a Cross-flow Heat Exchanger with Different fin thickness", International Journal of Dynamics of Fluids. ISSN 0973-1784 Volume 13, Number 2 (2017), pp. 345-362.

[13] Kishan, Ram and Singh, Devendra and Sharma, Ajay Kumar, CFD Analysis of Heat Exchanger Models Design Using ANSYS Fluent (February 18, 2020). International Journal of Mechanical Engineering and Technology 11(2), 2020, pp. 1-9, Available at SSRN: https://ssrn.com/abstract=3540077

[14] Liliana Dumitrescu, Edmond Maican, Ioan Pavel, Cătălin Dumitrescu, Iuliana Găgeanu and Andrei Pătruț. "Using CFD to improve the performance of a heat exchanger from a gasifier", E3S Web Conf., 286 (2021) 01010, https://doi.org/10.1051/e3sconf/202128601010

[15] M. Satish Kumar, Mr. A. Harish, Mrs. G. Santhoshirathnam (2017). "Design and Comparative CFD Analysis of Shell and Tube Heat Exchanger", (IJITR) International Journal of Innovative Technology and Research, Volume No.5, Issue No.6, October - November 2017, 7718-7723.

[16] Mahendra Kumar Verma, Vikas Bansal (2020). "Comparative CFD Analysis of Helical Shaped Earth Air Pipe Heat Exchanger Using Water Tube as a Thermal Reservoir Using Different Pipe Materials", International Journal of Renewable Energy Research, Vol .10, No.3, September, 2020.

[17] Maradona, R., & Rajkumar, S. (2015). CFD Analysis of Heat Transfer Characteristics of Helical Coil Heat Exchangers. In Applied Mechanics and Materials (Vol. 787, pp. 172–176). Trans Tech Publications, Ltd. https://doi.org/10.4028/www.scientific.net/amm.787.172

[18] Md Imran Pasha, Mohd Abdul Moyeed (2020). "Design and Thermal Analysis of Multi-Pipe Heat Exchanger by Changing the Materials and Adding Augmentation using CFD", International Journal of All Research Education and Scientific Methods (IJARESM), ISSN: 2455-6211, Volume 8, Issue 12, December-2020.

[19] Mohammed Irshad, Mohammed Kaushar, G. Rajmohan (2017). "Design and CFD Analysis of Shell and Tube Heat Exchanger", International Journal of Engineering Science and Computing, April 2017, Volume 7, Issue No.4, 6453-6457.

[20] Mohanty, S., & Arora, R. (2020). CFD Analysis of a Shell and Tube Heat Exchanger with Single Segmental Baffles. International Journal of Automotive and Mechanical Engineering, 17(2), 7890–7901. https://doi.org/10.15282/ijame.17.2.2020.08.0589

[21] Mukesh Kumar, P. C., & Chandrasekar, M. (2019). CFD analysis on heat and flow characteristics of double helically coiled tube heat exchanger handling MWCNT/water nanofluids. Heliyon, 5(7), e02030. https://doi.org/10.1016/j.heliyon.2019.e02030

[22] Mukkera Hemanth, Sandeep Mulabagal (2017). "CFD Analysis of Shell and Tube Heat Exchanger with and Without Baffles by Using Nano Fluids", Journal of Emerging Technologies and Innovative Research (JETIR), www.jetir.org, December 2017, Volume 4, Issue 12, JETIR (ISSN-2349-5162).

[23] Nivedini, G., Prasad, K., Sandeep, C. et al. Empirical and CFD analysis of silica nanofluid using a double pipe heat exchanger. SN Appl. Sci. **2,** 2145 (2020). https://doi.org/10.1007/s42452-020-03916-3

[24] P.S. Aswin, A. Mohan (2021). "CFD Analysis of Shell Tube Heat Exchanger with and Without Baffler", International Research Journal of Engineering and Technology (IRJET), e-ISSN: 2395-0056, Volume: 08, Issue: 01, Jan 2021, www.irjet.net, p-ISSN: 2395-0072.

[25] Palash Goyal, Nitin Kushwah, Rahul Dabi. "CFD Analysis of Helical Coil Heat Exchanger", International Journal of Science & Engineering Development Research (www.ijrti.org), ISSN:2455-2631, Vol.3, Issue 7, page no.70 - 74, July-2018, http://www.ijrti.org/papers/IJRTI1807012.pdf

[26] Perone C, Romaniello R, Leone A, Catalano P, Tamborrino A (2021). CFD Analysis of a Tubular Heat Exchanger for the Conditioning of Olive Paste. Applied Sciences. 2021; 11(4):1858. https: //doi. org/10. 3390/app11041858

[27] Petrik M., Szepesi G.L., (2018), "Shell side CFD analysis of a model shell-and-tube heat exchanger", Chemical Engineering Transactions, 70, 313-318, https://doi.org/10.3303/CET1870053

[28] Pramod Deshmukh, Vikram D Patil, Prof. Baviskar Devakant (2016). "CFD Analysis of Heat Transfer in Helical Coil Tube in Tube Heat Exchanger", Novateur Publications, International Journal of Innovations in Engineering Research and Technology [IJIERT], ISSN: 2394-3696, Volume 3, Issue 1, Jan.-2016.

[29] Rahul Singh, Divyank Dubey, Harishchandra Thakur (2016). "CFD Analysis of Shell and Tube Heat Exchanger", IJSRD - International Journal for Scientific Research & Development, Vol. 3, Issue 12, 2016, ISSN (online): 2321-0613

[30] Ramkrishna Gondane, Y. M. Jibhakate (2018). "Design And CFD Analysis of Shell and Tube Heat Exchanger Using Plain Tube and Corrugated Tube", IJARIIE, ISSN(O)-2395-4396, Vol-4 Issue-3 2018.

[31] S. Elangovan, M. Sundararaj, E. Mahavishnu (2021). "CFD Analysis of Monolithic Heat Exchanger by Using Various Ceramic Materials", Turkish Journal of Computer and Mathematics Education, Vol.12, No. 1S (2021), 498 – 501, https://doi.org/10.17762/turcomat.v12i1S.1913

[32] Sachin k. Patel, A. R. Patel (2015). "Review Paper on Investigation of Performance for Shell and Tube Heat Exchanger", International Journal of Advance Engineering and Research Development, Volume 2, Issue 3, March -2015.

[33] Shubham Sharma, Shalab Sharma, Mandeep Singh, Parampreet Singh, Rasmeet Singh, Sthitapragyan Maharana, Nima Khalilpoor, Alibek Issakhov, (2021) "Computational Fluid Dynamics Analysis of Flow Patterns, Pressure Drop, and Heat Transfer Coefficient in Staggered and Inline Shell-Tube Heat Exchangers", Mathematical Problems in Engineering, Article ID-6645128, 10 pages, https: //doi. org/10.1155/2021/6645128

[34] Sugin Elankavi R, Uma Shankar T (2018). "Study of Flow and Heat Transfer Analysis in Shell and Tube Heat Exchanger using CFD", International Research Journal of Engineering and Technology (IRJET), e-ISSN: 2395-0056, Volume: 05, Issue: 10, Oct 2018, www.irjet.net, p-ISSN: 2395-0072

[35] Yamini Y. Pawar, Ashish N. Sarode, Rajesh V. Dahibhate, "An Experimental and CFD Analysis on Helical Coil Heat Exchanger with Different Geometry", International Journal of Analytical, Experimental and Finite Element Analysis, RAME Publishers, vol. 4, issue 2, pp. 21-26, June 2017. https: //doi. org/10. 26706/IJAEFEA.2017.2.4.21-26

[36] Yuvraj Singh Rajput, Prof. Abhishek Arya (2019). "CFD Analysis of Cross Flow Heat Exchanger with Different Fin Thickness", July 2019, IJIRT, Volume 6, Issue 2, ISSN: 2349-6002.