Design and Analysis of Shell and Tube Heat Exchanger using different tubes materials

Ravi Kumar Gupta[1], Deenoo Pawar[2]

[1] Research Scholar, Department of Mechanical engineering, LNCT, Bhopal
[2] Assistant Professor, Department of Mechanical engineering, LNCT, Bhopal

ABSTRACT

In this work, a brief research background is provided over the tube and shell heat exchangers. Heat exchanger are very extensively brought in use for applications like heat transfer in the industries. Tube and shell heat exchanger are among such heat exchangers and provides more areas for transferring heat among the two fluids in comparison with other kinds of heat exchangers. Tube heat exchanger and shell heat exchanger are very extensively brought in use for applications like liquid to liquid heat transfer constituting of high-density working flute. The focus of this study is over the utilization of tube and shell heat exchangers with different materials: copper, steel and aluminium. Further with the help of ANSYS, a computational model was implemented for the same heat exchanger. This study has conducted investigations over the tube and shell heat exchanger with assessment off exchanger’s effectiveness and temperature. Steel, copper and aluminum showed effectiveness of 0.738, 0.748 and 0.747 respectively. This result showed that copper has the maximum effectiveness and steel has the lowest effectiveness.

Keywords: Shell and tube heat exchanger, baffles, pressure drop, heat transfer coefficient, Reynolds Number, Nusselt Number, baffle spacing and baffle angle.

INTRODUCTION

Heat exchanger can be defined as a device that transfers heat from one fluid to another. The fluids can be single-phase or two-phase, and they are separable or are in direct contact, depending over exchanger’s type. (Hemanth and Mulabagal, 2017) Heat exchangers are not generally thought of as devices that use energy sources like nuclear-fuel pins or fired heaters, despite the fact that many of the principals involved in their design are the same. In many industrial applications, it is beneficial to raise one fluid's temperature while reducing another’s temperature. This double action is economically accomplished by a heat exchanger. Chilling one petroleum fraction while warming another, cooling air or other gases with water between compression stages, including preheating combustion air fed to a boiler furnace utilising hot flue gas as the heating medium are just a few of its applications. Other applications include transferring heat through metals to water within the atomic power plants and recovering heat from a gas turbine's exhaust by transferring energy to the compressed air on its way.

Shell and tube heat exchangers in their various construction modifications are probably the most widespread and commonly used basic heat exchanger configuration in industry. Shell-and-tube heat exchangers are further classified according to the number of shell and tube passes involved. (Pathak, Tembhurne and Gangwar, 2017) Shell and tube heat exchangers are typically used for high-pressure applications (with pressures greater than 30 bar and temperatures greater than 260 °C). This is because the shell and tube heat exchangers can withstand high pressures due to their shape. In this type of heat exchanger, a number of small bore pipes are fitted between two tube plates and primary fluid flows through these tubes. (Dew and Shrivastava, 2018) The tube bundle is placed inside a shell and the secondary fluid flows through the shell and over the surface of the tubes. In nuclear engineering, this design of heat exchangers is widely used as in case of steam generator, which are used to convert feed water into steam from heat produced in a nuclear reactor core. To increase the amount of heat transferred and the power generated, the heat exchange surface must be maximized. This is obtained by using tubes. Each steam generator can contain anywhere from 3,000 to 16,000 tubes, each about 19mm diameter. (Pavani and Kumar, 2018)

A shell and tube exchanger consists of a number of tubes mounted inside a cylindrical shell. Two fluids can exchange heat, one fluid flows over the outside of the tubes while the second fluid flows through the tubes. (Santhakumar, B.Meganathan and M.Sanjeevkumar, 2018) The fluids can be single or two phase and can flow in a parallel or a cross/counter flow arrangement.

The shell and tube exchanger consists of four major parts:

- Front Header—this is where the fluid enters the tubeside of the exchanger. It is sometimes referred to as the Stationary Header.
- Rear Header—this is where the tubeside fluid leaves the exchanger or where it is returned to the front header in exchangers with multiple tubeside passes.
- Tube bundle—this comprises of the tubes, tube sheets, baffles and tie rods etc. to hold the bundle together.
- Shell—this contains the tube bundle.

The remainder of this section concentrates on exchangers that are covered by the TEMA Standard. (Kumar *et al.*, 2018)

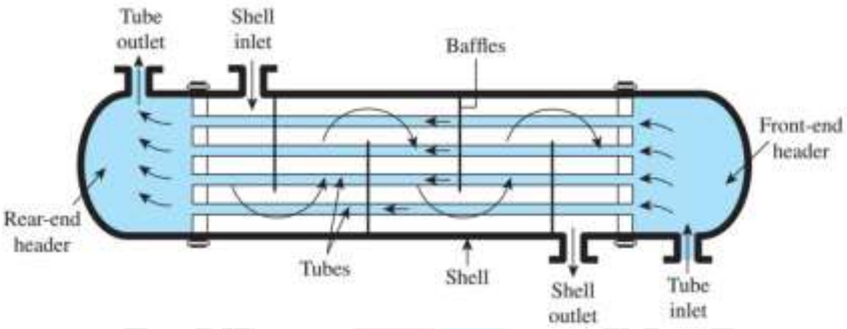

Figure 1: Shell and Tube Heat Exchangers

Applications of shell and tube heat exchanger:
These are highly used to make heat transfer possible between two fluids or mediums. These are used in industrial sectors for heating or cooling purpose. (Dhobale and Hole, 2018) The main applications are:

- Space heating
- Refrigeration
- Air conditioning
- Power plants
- Chemical plants
- HVAC
- Air processing

Advantages: The main advantage of STHX is that they are easy to service mainly when installed with floating baffles. Floating baffles are the baffles that are not welded to shell. (Pathak and Sahu, 2018)

LITERATURE REVIEW

(Abeykoon, 2020) A counter flow heat exchanger was evaluated for design needs, and its length was theoretically estimated using the LMTD technique, as well as the pressure drop and energy consumption using the Kern method. After that, using ANSYS, a computational model of the same heat exchanger was created, and this model was then extended to six other models by changing its essential design parameter for the optimization purposes. These models were eventually utilised to investigate heat transfer behaviour, mass flow rates, pressure drops, flow velocities, and vortices of shell and tube flows within the heat exchanger. The cooling performance of the hot fluid was just 1.05 percent different between theoretical and CFD values. Both the total heat transfer coefficient and pump power requirement exhibited good correlations with axial pressure decreases. Overall, the findings of this study show that CFD modelling can be useful for the design and optimization of heat exchangers, since it allows for the testing of several design choices without the need for physical prototypes.

(SANDEEP, Chowdary and Babu, 2020) The report's central idea is to investigate the LMTD (logarithmic mean temperature difference), Heat transfer coefficient, & Effectiveness (ε) of a combination of heat exchanger employing an acetoacetate/water combination as a function of various mass flow rates. This work investigates the effects of acetoacetate/water mixture on heat transfer coefficient, LMTD, productivity, and total heat transfer coefficient in 3 distinct heat exchangers: tubes in tube, shell and tube, and combination heat exchangers. The exploratory examination of the forced convective heat transfer and flow properties of a 25% acetoacetate containing 75% water is summarised in these conducting tests. Assuming laminar flow conditions, the acetoacetate/water mixture flows in a parallel, counter-clockwise direction in the tube in tube, shell and tube heat exchanger, and combination of heat exchanger. The largest increase in the coefficient of convective heat transfer was 56.3 percent, with 49.6 percent effectiveness. Based on a multi-pass flow of Acetoacetate/water, integrated heat exchangers give greater heat transfer characteristics than parallel & counter flow tubular & shell and tube heat exchangers.

(KISHORKUMAR, MEHTA and SHAH, 2019) Shell and Tube Heat Exchanger in which It is Fixed Tube-sheet and Counter Flow Type Heat Exchanger. In this research contain the analysis and comparison between plain tubes used Shell and tube heat exchanger and corrugated tubed used shell and tube heat exchanger system with different thickness. In this research, thermal analysis of the shell and tube heat exchanger. In this solid works 2014 is used for geometrical modelling. And also, TEMA standards are used for starting the experimental structure. Which are validate in ansys. Create a sample experiment model of a shell and tube heat exchanger inside which plain & corrugated tubes are used sequentially, as well as taking readings for thermal analysis calculations, and comparing the results of the experiment with the results of the Cfd simulation for affirmation.

(B, Lakshmi and Krishna, 2019) From the current situation, the Heat Exchangers uses extreme commonly are tube and Shell heat exchangers. The most usual uses of Shell and tube heat exchangers are electricity creation, cooling system of hydraulic fluid, oil in motors, transmissions, and hydraulic power packs. Shell and tube heat exchangers are made of the casing using a bunch of tubes with inside. The desirable outcome of the paper is to figure out the speed of heat transport using hot water as the hot liquid. The target of this paper is to mimic a tube and shell heat exchanger and also to assess blood flow and temperatures from the tubes and shell by employing applications tool Ansys. The simulation is composed of modelling and meshing cross section of tube and shell heat exchanger utilizing computational fluid dynamics (CFD). ANSYS Meshing is a general-purpose, intelligent, automated high-performance product. It produces the most appropriate mesh for accurate, efficient multi physics solutions.

(Pavani and Kumar, 2018) In the realm of energy conservation, conversion, as well as recovery, heat exchangers play a critical role. For the transfer of thermal energy, shell and tube heat exchangers are among the most extensively utilised in many technical applications. They're used in a variety of industries because of their capacity to transport vast volumes of heat in very low-cost, maintainable designs without combining hot and old fluids. They could provide a large number of functional tube surfaces while reducing floor space, liquid volume, and weight requirements. The heat transfer coefficient must be enhanced in order to compress the size of the heat exchangers. Nano fluids can be brought in use for increasing the heat exchanger performance. The heat transfer of a shell and tube heat exchanger operating with Nano fluids was evaluated analytically at various volume concentrations and evaluated to water as the base fluid in this research. The study takes into account turbulent flow conditions.

(Sankararao *et al.*, 2017) A heat exchanger is a device used throughout the power generation, refrigeration, as well as chemical process sectors that permits heat from one fluid to flow to another without the fluids mixing or coming into direct contact. For several years, forced convection has been used in shell and tube heat exchangers to lower the hot fluid temperature while raising the cold fluid temperature. Work aimed for using the ANSYS software as well as experimental simulations to investigate temperature drops utilising camphor-water as a hot fluid and water as a cold fluid with varying inlet parameters such as temperature and velocity. The cross flow heat exchanger model was strengthened and studied for this purpose. The experimental results were compared to the CFD values.

(Singh and Kumar, 2014) The tube and shell heat exchanger are among such kinds of heat exchanger in which hot water flows inside one tube and cold water flows over the other. The heat exchanger is simulated using a CFD technique, which is a computer-based simulation that involves fluid movement and heat transmission. For determining the temperature gradients, pressure distribution, as well as velocity vectors, CFD resolves the entire heat exchanger in discrete elements. For correct CFD results, the turbulence model k- is being used. Temperature differences are estimated for parallel and counter flow by adjusting the mass flow rate of fluid of 2L/min and 3L/min, which is regulated by a rota metre, and temperature variations are detected by sensors mounted at the inlet and exit of tube. The solid geometry is created in SOLID WORKS and then loaded into GAMBIT, which is the ANSYS 13.0 pre-processor for meshing the geometric shapes. The simulated results, such as temperature contours, pressure contours, and velocity vectors, are generated using the post processor FLUENT.

METHODOLOGY

Step of working

Expected Procedure to be followed during the complete study:

- Modeling of Shell and Tube type heat exchanger according to the selected base paper.
- Further converting the file for compatibility in simulation process.
- Importing the file in Ansys Fluent for performing the simulation in Ansys.
- Assigning the name selection to the different parts of Shell and Tube type heat exchanger.
- Perform the meshing of cross flow heat exchanger compatible for CFD analysis.
- Providing the suitable boundary conditions according to the selected base paper.
- Assigning the material properties.
- Setting the proper setup for CFD analysis procedure.
- Evaluating the results after the finish of simulation work.

The prime motive is to estimate the thermal behavior as well as distribution of velocities produced by the gradient of temperature and hence contrast the thermal properties.CFD is an established method of solving fluid dynamics problems by using numerical methods such as: Navier-Stokes Equations, Euler-Equations or Partial-Differential Equations.

Navier-Stokes Equation

The Navier-Stokes equations govern the motion of fluids and can be seen as Newton's second law of motion for fluids.

$$\rho \frac{D\vec{V}}{Dt} = \rho\vec{g} - \nabla P + \mu\nabla^2\vec{V}$$

where, $\frac{D}{Dt}(\vec{V}) = \frac{\partial}{\partial t}(\vec{V}) + u\frac{\partial}{\partial x}(\vec{V}) + v\frac{\partial}{\partial y}(\vec{V})$

$$\nabla^2(\vec{V}) = \frac{\partial^2}{\partial^2 x}(\vec{V}) + \frac{\partial^2}{\partial^2 y}(\vec{V})$$

Continuity Equation

According to the Continuity Equation, if no fluid is added or removed from the pipe in any length then the mass passing across different sections shall be the same. This is in accordance with the principle of conservation of mass which states that matter can neither be created nor be destroyed.

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$$

Momentum Equation

Linear momentum equation for fluids can be developed using Newton's 2nd Law which states that sum of all forces must equal the time rate of change of the momentum, **ΣF** = d(m**V**)/dt. This easy to apply in particle mechanics, but for fluids, it gets more complex due to the control volume.

- x – momentum: $\rho\frac{D}{Dt}(u) = \rho g_x - \frac{\partial P}{\partial x} + \mu\nabla^2(u)$

- y – momentum: $$\rho \frac{D}{Dt}(v) = \rho g_y - \frac{\partial P}{\partial y} + \mu \nabla^2 (v)$$
- z – momentum: $$\rho \frac{D}{Dt}(w) = \rho g_y - \frac{\partial P}{\partial y} + \mu \nabla^2 (w)$$

Meshing

Meshing is an integral part of the engineering simulation process where complex geometries are divided into simple elements that can be used as discrete local approximations of the larger domain. The mesh influences the accuracy, convergence and speed of the simulation.

Number of Nodes	242206
Number of element	720686

- Nodes and element of heat exchanger

It is determined as number of nodes, 242206 and number of element, 720686 for meshing.

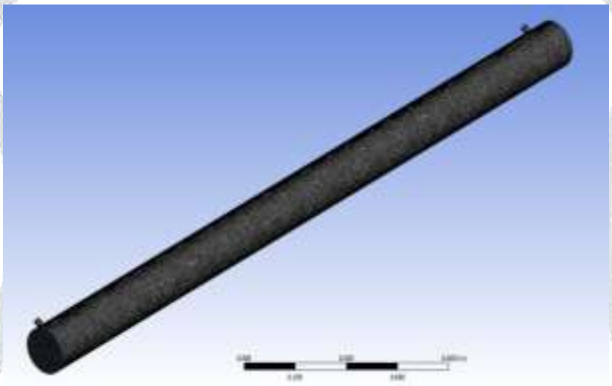

Figure 2: mesh size along with number of nodes and elements

Boundary condition

- Water liquid is selected as fluid flow through both cold and hot fluid.
- For 3 different cases, different material is used for pipe:-
    - ➢ Steel
    - ➢ Copper
    - ➢ Aluminium
- 2.465 kg/s mass flow inlet is selected for cold inlet with inlet temperature of 20°C.
- 1.219 kg/s mass flow inlet is selected for hot inlet with inlet temperature of 100°C.

RESULTS AND DISCUSSION

The result obtained with three different cases of mixture compound for hot outlet and the temperature variation throughout the system and surrounding.

Case-1 Steel

- Temperature contour at hot outlet and cold outlet

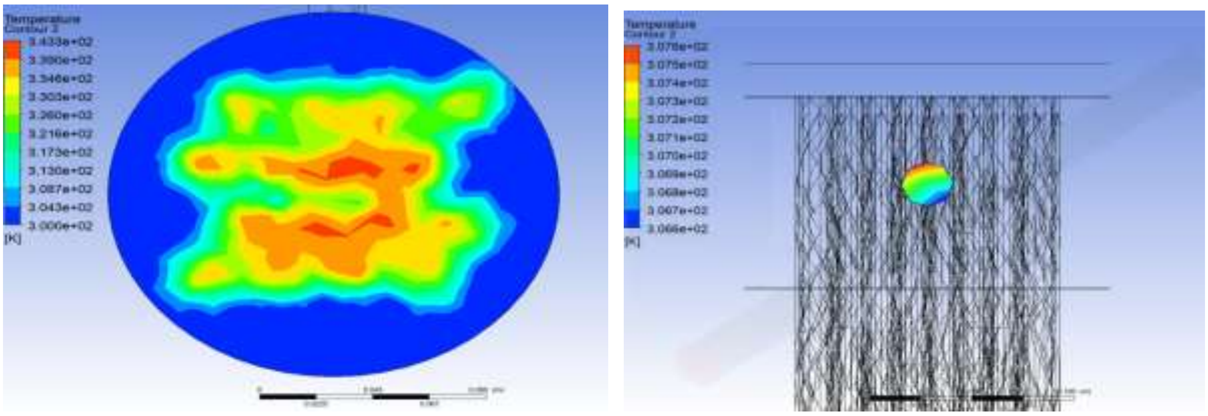

Figure 3: Temperature contours of steel

- Temperature Volume Rendering

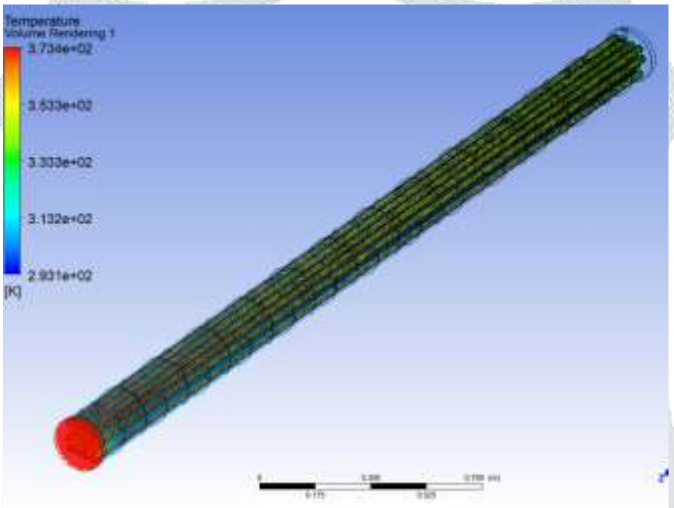

Figure 4: Temperature volume rendering of steel

Case-2 Copper

- Temperature contour at hot outlet and cold outlet

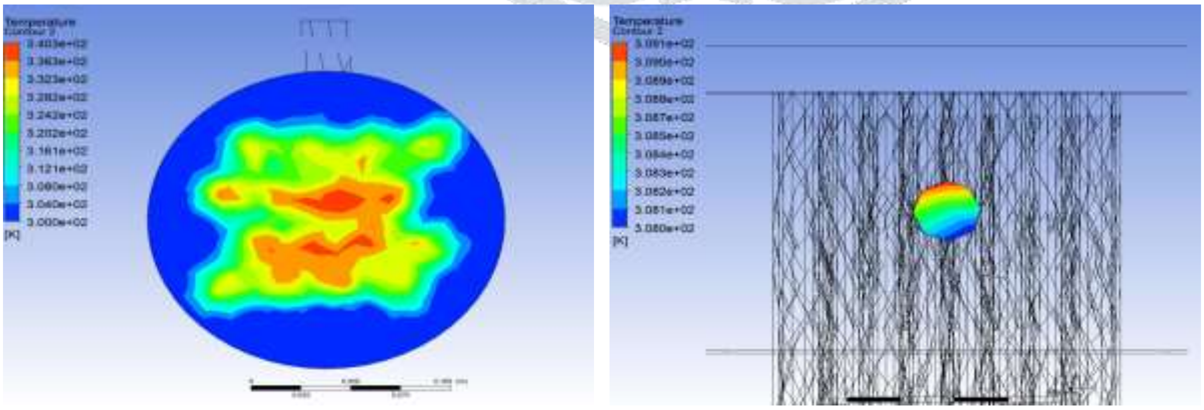

Figure 5: Temperature contour of copper

- Temperature Volume Rendering

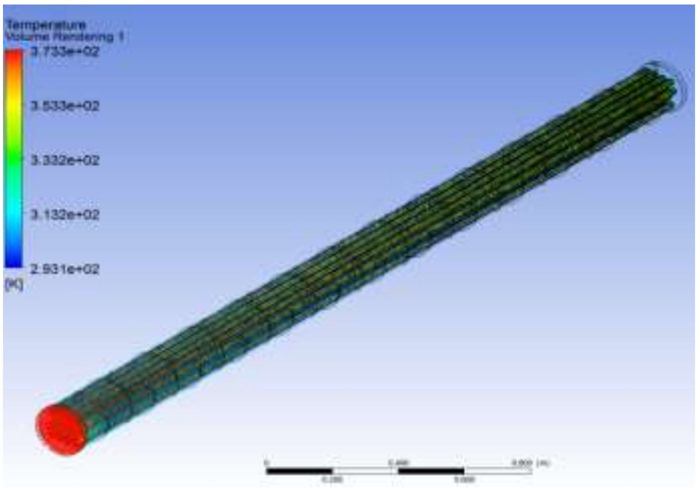

Figure 6: Temperature volume rendering of copper

Case-3 Aluminium

- Temperature contour at hot outlet and cold outlet

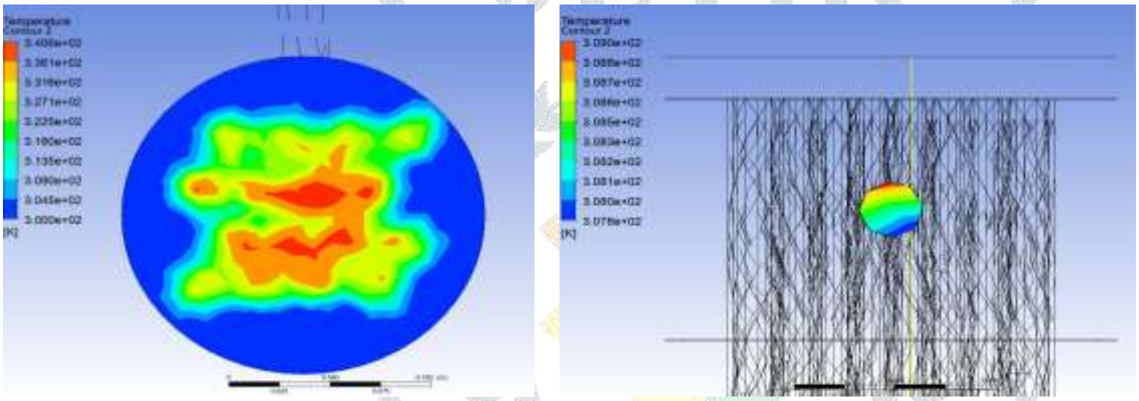

Figure 7: Temperature contour of aluminium

- Temperature Volume Rendering

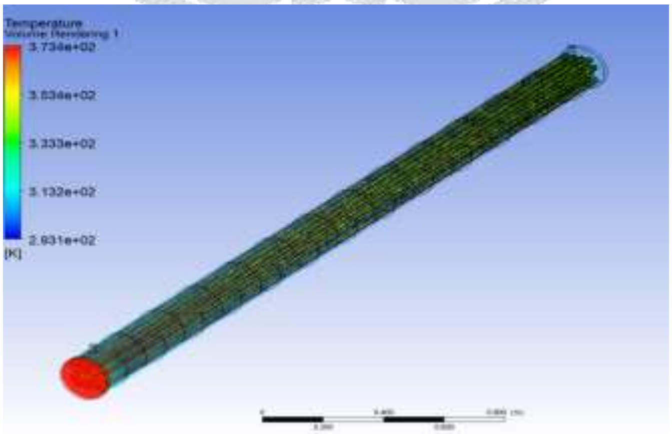

Figure 8: Temperature volume rendering of aluminium

Comparative study of steel, copper and aluminium

The graph shown below represents the length v/s temperature graph of steel, copper and aluminium. Steel is indicated by blue, copper by red and aluminium by green color.

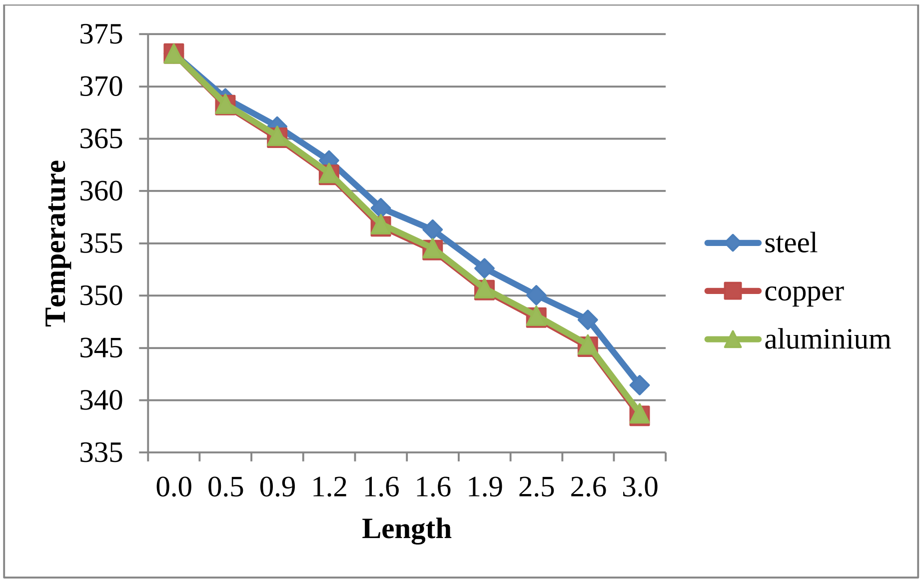

Figure 9: Graphical representation of steel, copper and aluminium

Table 1: Hot and cold temperature at inlet, steel, copper and aluminium

Condition	Inlet	steel	copper	aluminium
Cold Temperature(k)	293.15	307.041	308.415	308.276
Hot Temperature(k)	373.15	314.072	313.246	313.32

Table 2: Effectiveness of all three cases is mentioned below

Case	**Effectiveness**
Steel	0.738
Copper	0.748
Aluminium	0.747

From the results, it can be concluded that copper is highly effective, and steel is least effective after comparing all three cases such as: steel, copper and aluminum. Overall, The findings obtained from this study exhibits a positive agreement among the CFD and the theoretical outcomes. In addition, it is observed that CFD is a very promising tool for designing compact heat exchangers through optimising the performances When brought in use with use an appropriate theoretical validation.

CONCLUSION

Extensive applications of tube and shell heat exchangers in the industrial sector were found and taken as the key topic by numerous researchers to research on. For increasing performances of these heat exchangers, simulations were conducted with the help of ANSYS through using three different materials.

The finding of the study are as follows:

- Copper attained the cold and hot temperature of 308.415K and 313.246K respectively. Copper provided the best results among all the 3 materials on the basis of temperature.

Steel, copper and aluminum showed effectiveness of 0.738, 0.748 and 0.747 respectively. This result showed that copper has the maximum effectiveness and steel has the lowest effectiveness.

REFERENCES

1. Abeykoon, C. (2020) ‘Compact heat exchangers – Design and optimization with CFD’, *International Journal of Heat and Mass Transfer*. Elsevier Ltd, 146, p. 118766. doi: 10.1016/j.ijheatmasstransfer.2019.118766.
2. B, K. P., Lakshmi, B. U. and Krishna, A. H. (2019) ‘Numerical Simulation of Shell and Tube Heat Exchanger using Ansys Fluent’, *International Journal of Innovative Technology and Exploring Engineering*, 8(11S2), pp. 543–545. doi: 10.35940/ijitee.k1093.09811s219.
3. Dew, K. K. and Shrivastava, P. (2018) ‘Cfd Analysis of Double Tube Heat Exchanger Using Different Nanofluid’, *International Journal for Research Trends and Innovation*, 3(7), pp. 146–155.
4. Dhobale, A. A. and Hole, J. A. (2018) ‘Review on Heat Exchanger Using Different Nano Fluids’, *IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)*, pp. 1–4.
5. Hemanth, M. and Mulabagal, S. (2017) ‘Cfd Analysis of Shell and Tube Heat Exchanger With and Without Baffles By Using Nano Fluids’, *Journal of Emerging technologies and innovative research*, 4(12), pp. 25–31.
6. KISHORKUMAR, G. H., MEHTA, D. M. and SHAH, A. P. J. (2019) ‘EXPERIMENTAL AND NUMERICAL STUDY OF SHELL AND TUBE HEAT EXCHANGER BY VARYING THE INNER RADII OF CORRUGATED TUBE’, *International Journal of Technical Innovation in Modern Engineering & Science (IJTIMES)*, 5(04), pp. 629–640.
7. Kumar, J. V. *et al.* (2018) ‘MODIFIED DESIG OF SHELL AND TUBE HEAT EXCHANGER AND CFD ANALYSIS’, *INTERNATIONAL JOURNAL OF CURRENT ENGINEERING AND SCIENTIFIC RESEARCH (IJCESR)*, 5(4), pp. 39–44.
8. Pathak, A. K., Tembhurne, Y. K. and Gangwar, M. (2017) ‘A Review on Comparison of Computational Fluid Dynamics Analysis of Natural Fluid & Nanofluid in Helical Coil Heat Exchanger’, *International Journal of Current Research and Review*, 2(8), pp. 83–90. doi: 10.7324/ijcrr.2017.9197.
9. Pathak, S. and Sahu, H. S. (2018) ‘CFD Analysis of Heat Transfer Enhancement in Shell and Tube Type Heat Exchanger creating Triangular Fin on the Tubes’, *International Journal of Trend in Scientific Research and Development*, Volume-2(Issue-4), pp. 1161–1170. doi: 10.31142/ijtsrd14259.
10. Pavani, K. and Kumar, V. R. (2018) ‘NUMERICAL INVESTIGATIONS FOR PERFORMANCE IMPROVEMENT IN SHELL AND TUBE HEAT EXCHANGERS USING NANO FLUIDS’, *International Journal of Engineering Sciences & Research Technology*, 7(7), pp. 1–16.
11. SANDEEP, M. S., Chowdary, D. V. L. and Babu, K. madh. (2020) ‘Comparison On Cfd Analysis Of Acetoacetate / Water & Investigational Analysis Of Heat Transfer Characteristics Of Acetoacetate / Water By Using Tube In Tube And Shell And Tube Heat Exchanger’, *European Journal of Molecular & Clinical Medicine*, 07(08).
12. Sankararao, U. *et al.* (2017) ‘Enhancement of heat transfer in shell and tube heat exchanger by using nano fluid’, *International Journal of Mechanical and Production Engineering Research and Development*, 7(5), pp. 191–198. doi: 10.24247/ijmperdoct201720.
13. Santhakumar, S. T., B.Meganathan and M.Sanjeevkumar (2018) ‘Comparsion and Cfd Analysis of Heatexchanger With Nanofluid for Increasing Effectiveness With and With Out Baffles’, *International Journal of Pure and Applied Mathematics*, 118(24), pp. 1–11.
14. Singh, G. and Kumar, H. (2014) ‘Computational Fluid Dynamics Analysis of Shell and Tube Heat Exchanger’, *Journal of Civil Engineering and Environmental Technology*, 1(3), pp. 66–70.