Academic Guru Publication House

Technological Advancements : Research & Reviews
Volume 1
Year: 2021

AG PH Books

To study the thermal performance of Shell and Tube Heat Exchanger by Using Different Tubes Materials

Ravi Kumar Gupta[1*]

[1]*Research Scholar, Department of Mechanical Engineering, Lakshmi Narayan College of Technology, Bhopal*

Abstract

In this work, a brief research background is provided over the tube and shell heat exchangers. Heat exchangers 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: Heat Exchanger; Tube and shell heat exchanger; ANSYS; CFD.

[*] ISBN No. 978-81-953278-8-1

1. INTRODUCTION

A heat exchanger is a device that transfers heat between two fluids. Different types of exchangers call for different types of fluids, which might be either separate or in direct touch. Even though many of the principals involved in their construction are similar to those used in nuclear-fuel pins or fired heaters, heat exchangers are not typically considered to be devices that utilise energy sources. Temperature differences between two fluids may be advantageous in many industrial applications. To do this double duty, a heat exchanger saves money.

Some examples of its uses include chilling one petroleum fraction while warming another, preheating combustion air supplied to a boiler furnace with hot flue gas, and cooling air or other gases with water between compression stages. Metals may be used to transmit heat to water inside atomic reactors, while compressed air from a gas turbine can be used to recover heat from exhaust. Heat exchangers are used widely in power plants, gas turbines, heating and air conditioning, refrigeration, and the chemical industry, among other uses.

1.1. Computational Fluid Dynamics

If you'd want to better understand the numerous components of a CFD simulation, here is an overview of how it's done. The technique is broken down into the following steps:

- “Flow Problem is formulated”
- “Geometry & Flow Domain are modelled”
- “Establishment of the Boundary and Initial Conditions”
- “Generation of the Grid”
- “Establishment of the Simulation Strategy”
- “Establishment of the Input Parameters and Files”
- “Perform the Simulations”
- “Monitoring the Simulations for Completion”
- “Post-process the Simulation to get the Results”
- “Make Comparisons of the Results”
- “Repeat the Process to Examine Sensitivities”
- “Document”

1.2. Principles of Heat Exchanger

Heat flows from higher to lower temperatures on their own, which is how heat exchangers function. To put it another way, heat may be transferred from hot to cold fluids by use of heat conducting surfaces.

The rate of heat flow (kW/m2 of transfer surface) at any given position relies on the following factors:

- The heat transfer coefficient (U) is a function of the fluid characteristics, fluid velocity, building materials, geometry, as well as hygiene of the heat exchanger.
- Differences among cold and Hot streams temperature

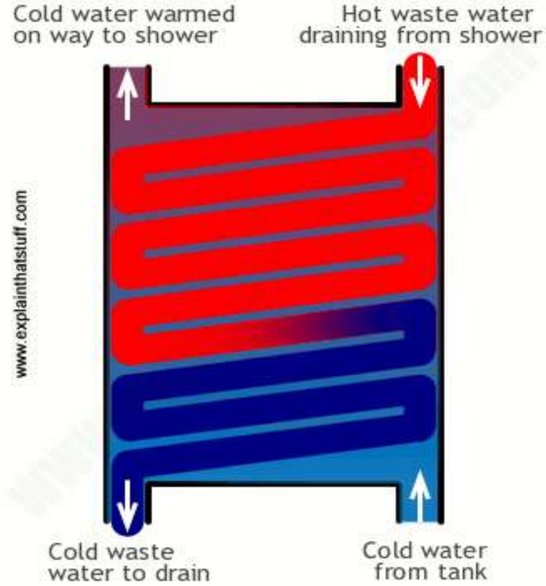

Figure 1: Simplified Heat Exchangers

1.3. Objects of Research

The main aim of this research work is to ascertain the following changes in relation to the response of a Shell and Tube Heat Exchanger pipe materials on Steel, Copper and Aluminium.

1. To performed the CFD analysis of "shell and tube heat exchanger" using ANSYS software.
2. To check the Hot and Cold water outlet temperature using different pipe material.
3. To calculate the effectiveness of "shell and tube heat exchanger" and compare all three materials result.
4. To suggest the best material for manufacturing "shell and tube heat exchanger" by using CFD investigation.

2. LITERATURE REVIEW

(Du, Jiang and Wang, 2020) [1] Among the most energy-efficient but also good for the environment construction of support services is the geothermal heat exchanger system. This study used a CuO/water nanofluid as the heat transfer fluid to improve the energy efficiency of "geothermal heat exchangers". A 3D model, well-validated against actual data on nanofluids in geothermal heat exchangers, was used to study the effect of nanoparticle diameter and circularity on the thermal performance of the "geothermal heat exchanger".

(Sunil and Kumar, 2019) [2] Laminar and turbulent regimes of alumina-water nanofluid convection and pressure drop were studied using viscous laminar models and the conventional k-model. There is a micro-channel heat exchanger (0.1 m length and 0.5-mm inner diameter) with an Al2O3 water nanofluid flow of 0 to 5 percent and cold water pouring out, with constant wall temperatures, within the spherical micro-channel. The experiment revealed that when the quantity of nanoparticles rose, convection heat transfer dramatically increased at different Reynolds numbers. The heat transfer coefficient improved 15% in the laminar regime and 12% in the turbulent zone as compared to pure water at 5% volume concentration (Nusselt number decreased about 2.5 percent).

(Krishnakanth, Aravind and Ashok, 2019) [3] Heat exchangers with two tubes or concentric tubes have been used in power plants, refrigeration, and chemical facilities to enhance heat transfer rates and efficiency. Throughout this research, an examination of a twin tube heat exchanger with variable flow rates as well as nanoparticle concentrations will be carried out. The heat transfer rate in a heat exchanger was increased by using Al2O3 nanofluid as a base fluid. The assessment will be carried out using ANSYS Fluent 14.0 and CFD technique. For the counter current flow pattern, the heat transfer rate is changed. The heat transmission rate is increased by using nanofluids as a base fluid.

(KISHORKUMAR, MEHTA and SHAH, 2019) [4] 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 validating 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. Utilize Solid Works for geometry & design, ICEM CFD (ANSYS) for meshing, as well as Fluent for analysis in Computational Fluid Dynamics (CFD) Analysis Software.

(B, Lakshmi and Krishna, 2019) [5] **"**Tube and Shell heat exchangers" are the most prevalent types of heat exchangers now in use. These heat exchangers are often used to generate energy, cool hydraulic fluid or oil in motors, gearboxes or hydraulic power packs and to cool hydraulic fluid or oil used in motors and transmissions. These heat exchangers are constructed of a casing and a number of tubes with an inner core. In this study, the goal is to determine out how fast heat can be transferred using hot water. A tube and shell heat exchanger is the focus of this research, which uses the Ansys programme to simulate the heat exchanger and measure blood flow and temperature from the tubes and the shell. "Computational fluid dynamics (CFD)" is used to model and mesh the cross section of a tube and shell heat exchanger for the simulation.

(Pavani and Kumar, 2018) [6] Heat exchangers serve an important role in energy conservation, conversion, and recovery. Shell-and-tube heat exchangers are among the most widely used in numerous technological applications for the transmission of thermal energy. A number of industries employ them because to their ability to convey large amounts of heat in relatively low-cost, easily maintained designs without mingling hot and old fluids. Nanofluid physical characteristics and numerical simulation volume fractions are used to analyse the heat exchanger's thermal and CFD properties. CATIA is used to create a 3-Dimensional model of the proposed heat exchanger, as well as Ansys fluent is used to analyse it. There are 2 materials used for making the tubes of the heat exchanger: copper and aluminium.

(Santhakumar, B.Meganathan and M.Sanjeevkumar, 2018) [7] Development of nanofluids is aimed at improving heat transfer coefficient and reducing the size of thermal fluids used in heat exchangers. Thermal conductivity, viscosity, specific heat, as well as density are some of the major elements that determine the heat transfer characteristics of nanofluids. Nanofluids' thermo-physical characteristics are also influenced by their operating temperature. As a result, precise temperature-dependent property measurements of nanofluids are critical. The scope of this study is to review recent advances in nanofluid research and to do cfd analysis on nanofluids.

(Kumar et al., 2018) [8] Additionally, it is important to note that in this study, the researchers used the Ansys software tool for a heat exchanger with segmental baffles to explore the flow and temperature within the shell and tubes and to calculate total heat transfer for each design. The heat exchanger with 6 baffles is positioned along the shell and tube heat exchanger angles & orientations for establishing flow routes between tubes in this research, which analyzed both water & water and TiO2. The geometric model is compared by varying baffle inclination i.e. 450 and 900. The fundamental geometry of the heat exchanger is modeled by using CFD package ansys 15.0 in the modelling process. The complete heat transport is depicted in this diagram.

3. METHODOLOGY

3.1. Step of working

During the course of the study, the following procedures should be followed:[9]

1. Modeling of a Shell and Tube heat exchanger using the chosen base paper as a starting point.
2. For compatibility in the simulation process, the file has to be further converted.
3. Ansys Fluent is used to do the simulation.
4. Identifying the various sections of the Shell and Tube type heat exchanger by their names.
5. For CFD study, mesh the cross flow heat exchanger.
6. Providing the appropriate boundary conditions based on the chosen base paper for the experiment.
7. Defining the material's attributes.
8. Setting up the CFD analysis technique in the correct manner.
9. After the simulation has been completed, evaluate the findings.

3.2. Calculating the required length of the heat exchanger

The determination of the rate of mass flow of the hot fluid stream m_h) should be made with properly known Temperatures at outlet and inlet from equation: [10]

$$m_h = \rho_h \times v_{tube} \times A_{tube}$$

where the density of the hot fluid is signified by ρ_h, velocity of the fluid inside the tubes is signified byv_{tube} and cross sectional area of the tubes is signified by A_{tube}. Equation can then be brought in use for calculating the heat load (q) extracted from the hot fluid or acquired by the cold fluid.

$$q = mc_{ph}(T_{h,i} - T_{h,o}) = mc_{pc}(T_{c,i} - T_{c,o})$$

where T is the temperature and I and o are the intake and exit conditions, respectively. Following that, the shell design and tube bank layout were examined.

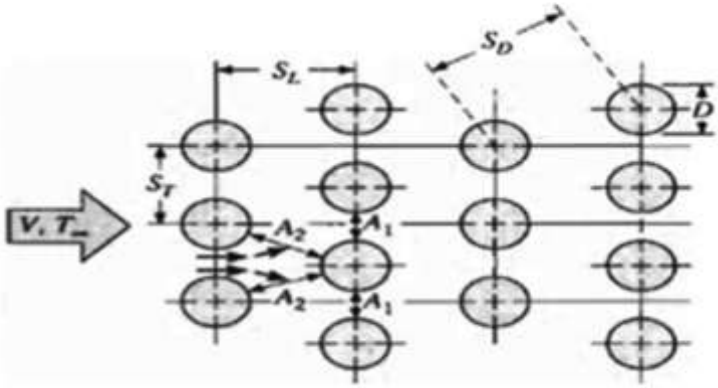

Figure 2: Staggered tube bank arrangement

3.3. Meshing

Meshing is a step in the engineering analysis method that uses breaking down complex geometries into simple parts which can be used as discrete local estimates to the wider domain. The mesh has an impact on the simulation's accuracy, resolution, and efficiency. [11] [12]

Table 1: Nodes and Element

Number of Nodes	242206
Number of element	720686

3.4. 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.

4. RESULTS AND DISCUSSION

4.1. A Case-1 Steel

The temperature at the hot outlet of the heat exchanger is shown in the below mentioned figure, that is 343.3K. In case 1, steel is used as the material for the pipe. There are two temperature ranges: the blue colour indicates the lowest and red colour indicates the highest.

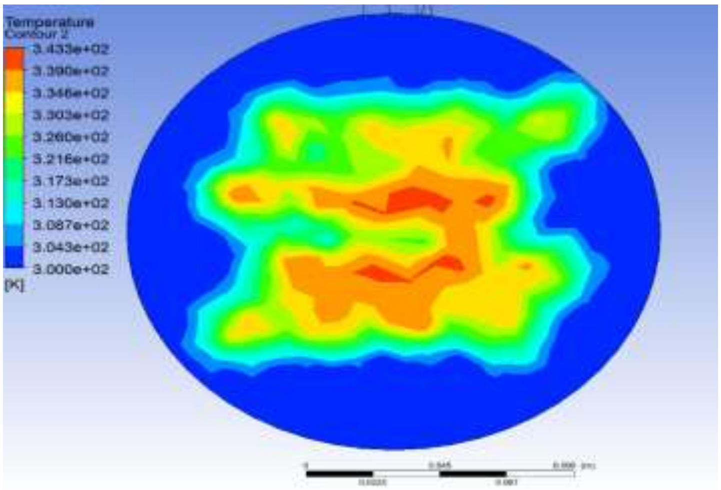

Figure 3: Case 1 hot outlet temperature

The temperature at the cold outlet of the heat exchanger is shown in the below mentioned figure, that is 307.6K. In case 1, steel is used as the material for the pipe. Blue represents the lowest temperature and red represents the highest temperature.

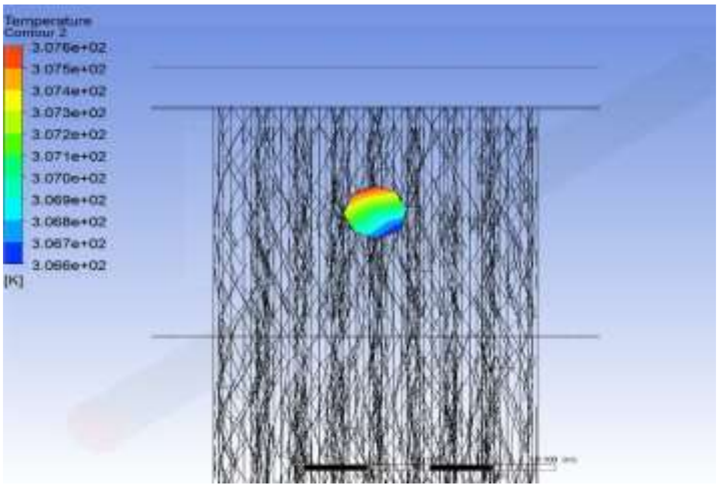

Figure 4: Case 1 cold outlet temperature

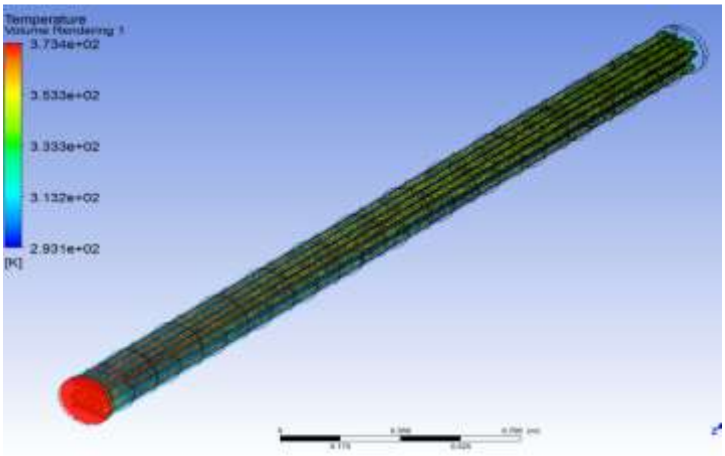

Figure 5: Case 1 temperature volume rendering

4.2. Case-2 Copper

The temperature at the hot outlet of the heat exchanger is shown in the below mentioned figure, that is 340.3K. In case 2, copper is used as the material for the pipe. Blue represents the lowest temperature and red represents the highest temperature possible.

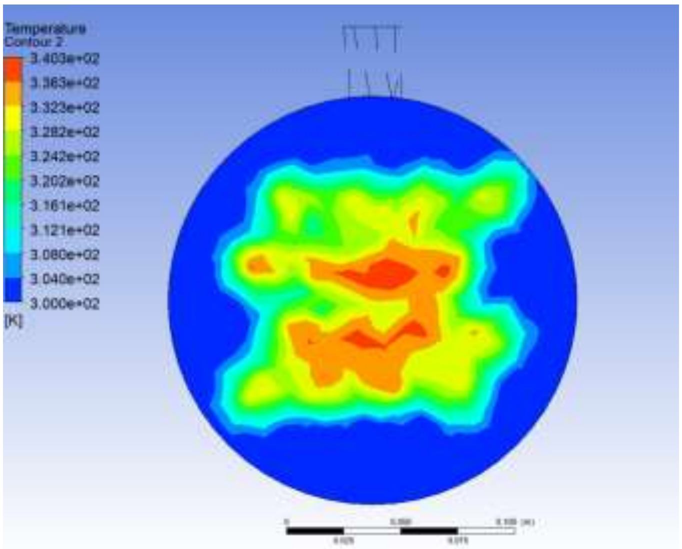

Figure 6: Case 2 hot outlet temperature

The temperature at the cold outlet of the heat exchanger is shown in the below mentioned figure, that is 309.1K. In case 2, copper is used as the material for the pipe. The blue colour represents the lowest temperature range, while the red colour represents the highest temperature range.

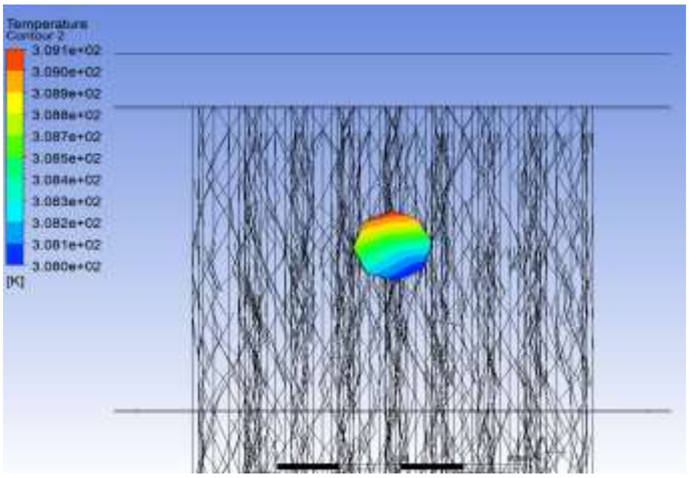

Figure 7: Case 2 cold outlet temperature

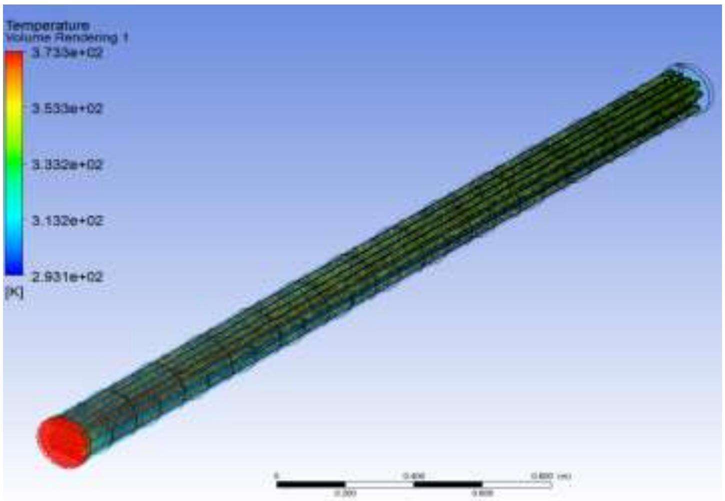

Figure 8: Case 2 temperature volume rendering

4.3. Case-3 Aluminium

The temperature at the hot outlet of the heat exchanger is shown in the below mentioned figure, that is 340.6K. In case 3, aluminium is used as the material for the pipe. There are two temperature ranges: the blue colour indicates the lowest and red colour indicates the highest.

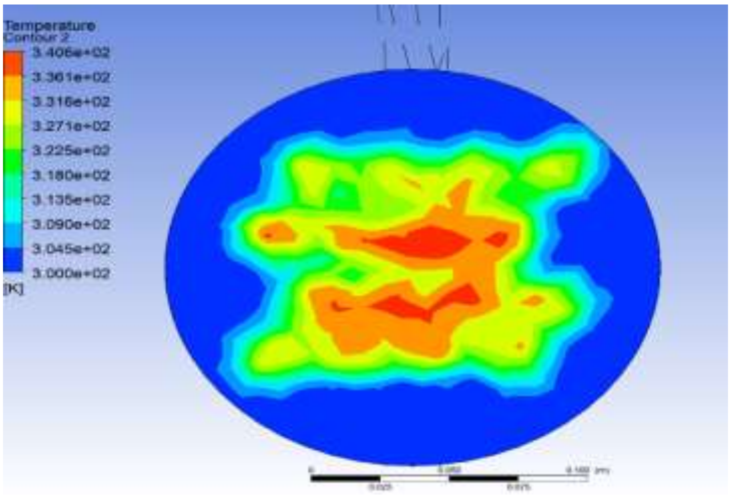

Figure 9: Case 3 hot outlet temperature

The temperature at the cold outlet of the heat exchanger is shown in the below mentioned figure, that is 309.0K. In case 3, aluminium is used as the material for the pipe. Blue represents the lowest temperature and red represents the highest temperature.

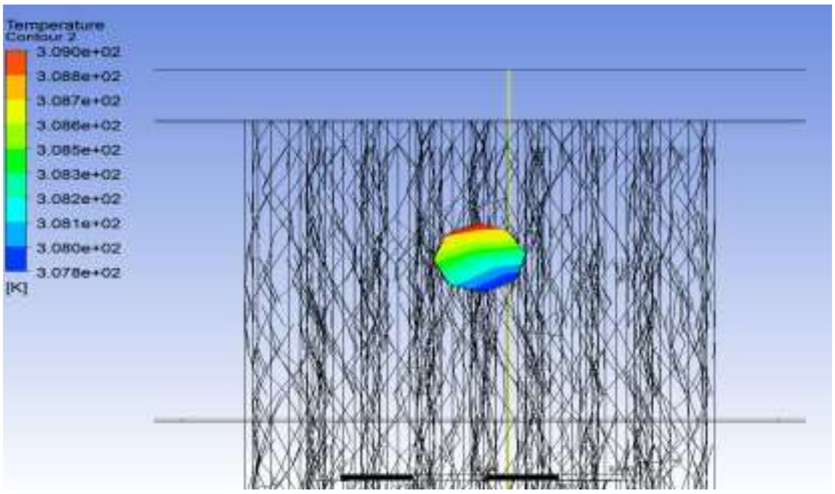

Figure 10: Case 3 cold outlet temperature

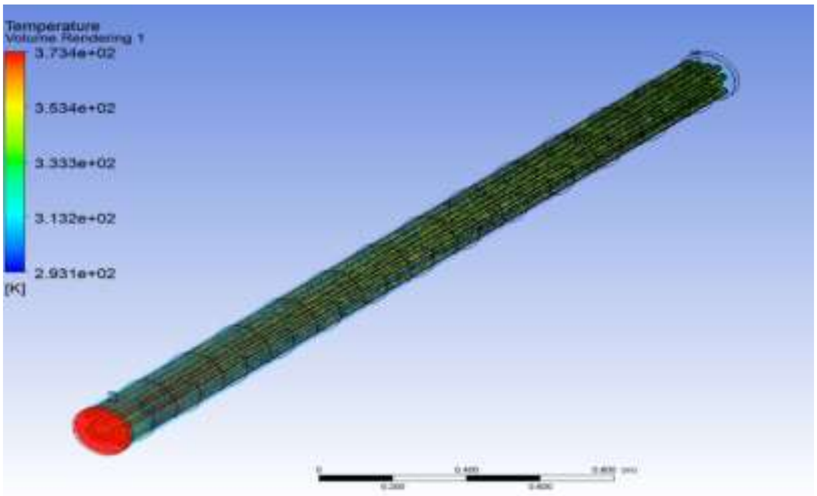

Figure 11: Case 3 temperature volume rendering

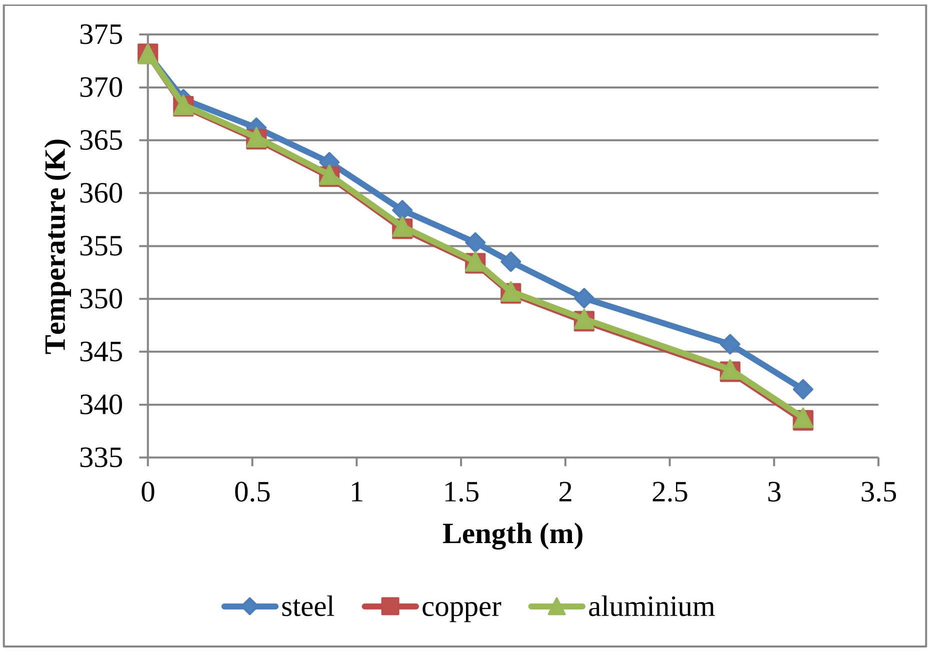

Figure 12: Hot tube temperature comparision graph

Table 2: Hot and cold outlet temperature

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 3: Effectiveness

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 optimizing the performances When brought in use with use an appropriate theoretical validation.

5. **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 findings 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.

6. References

[1] R. Du, D. Jiang, and Y. Wang, “Numerical investigation of the effect of nanoparticle diameter and sphericity on the thermal performance of geothermal heat exchanger using nanofluid as heat transfer fluid,” *Energies*, vol. 13, no. 7, 2020, doi: 10.3390/en13071653.

[2] A. K. Sunil and R. Kumar, “CFD analysis of thermal performance of microchannel nanofluid flow at different reynolds numbers,” *Songklanakarin J. Sci. Technol.*, vol. 41, no. 1, pp. 109–116, 2019, doi: 10.14456/sjst-psu.2019.13.

[3] M. S. Krishnakanth, C. T. A. Aravind, and M. A. Ashok, “Numerical Investigation of Concentric Tube Heat Exchanger using a Nanofluid,” *Int. J. Eng. Res. Technol.*, vol. 8, no. 06, pp. 1339–1344, 2019.

[4] G. H. KISHORKUMAR, D. M. MEHTA, and A. P. J. SHAH, “EXPERIMENTAL AND NUMERICAL STUDY OF SHELL AND TUBE HEAT EXCHANGER BY VARYING THE INNER RADII OF CORRUGATED TUBE,” *Int. J. Tech. Innov. Mod. Eng. Sci.*, vol. 5, no. 04, pp. 629–640, 2019.

[5] K. P. B, B. U. Lakshmi, and A. H. Krishna, “Numerical Simulation of Shell and Tube Heat Exchanger using Ansys Fluent,” *Int. J. Innov. Technol. Explor. Eng.*, vol. 8, no. 11S2, pp. 543–545, 2019, doi: 10.35940/ijitee.k1093.09811s219.

[6] K. Pavani and V. R. Kumar, “NUMERICAL INVESTIGATIONS FOR PERFORMANCE IMPROVEMENT IN SHELL AND TUBE HEAT EXCHANGERS USING NANO FLUIDS,” *Int. J. Eng. Sci. Res. Technol.*, vol. 7, no. 7, pp. 1–16, 2018.

[7] S. T. Santhakumar, B.Meganathan, and M.Sanjeevkumar, “Comparsion and Cfd Analysis of Heatexchanger With Nanofluid for Increasing Effectiveness With and With Out Baffles,” *Int. J. Pure Appl. Math.*, vol. 118, no. 24, pp. 1–11, 2018.

[8] J. V. Kumar, M. V. S. P. Kumar, J. S. Kumar, and T. N. V. A. Kumar, “MODIFIED DESIG OF SHELL AND TUBE HEAT EXCHANGER AND CFD ANALYSIS,” *Int. J. Curr. Eng. Sci. Res.*, vol. 5, no. 4, pp. 39–44, 2018.

[9] H. V. Rao, J. Bala, and B. Rao, “Fluent Analysis of Shell and Tube Heat Exchanger with Different Materials and Fluids,” *Int. J. Mag. Eng. Technol. Manag. Res.*, vol. 5, no. 2, pp. 2–3, 2018.

[10] K. K. Dew and P. Shrivastava, “Cfd Analysis of Double Tube Heat Exchanger Using Different Nanofluid,” *Int. J. Res. Trends Innov.*, vol. 3, no. 7, pp. 146–155, 2018.

[11] U. Sankararao, N. Ramesh, P. P. D. Rao, and D. B. Rao, “Enhancement of heat transfer in shell and tube heat exchanger by using nano fluid,” *Int. J. Mech. Prod. Eng. Res. Dev.*, vol. 7, no. 5, pp. 191–198, 2017, doi: 10.24247/ijmperdoct201720.

[12] H. Yerrennagoudaru, B. V. Prasad, and S. V. Kumar, “Nano Fluids for Heat Exchanger,” *Int. J. Eng. Sci. Innov. Technol.*, vol. 5, no. 4, pp. 82–89, 2016. Design And Analysis of Shell and Tube Heat Exchanger Using Different Tubes Materials