CFD ANALYSIS OF HEAT TRANSFER USING COUNTER FLOW SINGLE-PASS SHELL AND TUBE HEAT EXCHANGER

A thesis

by

ABUL HASNAT HIMEL: BME-1903019127
ARMAN HOSSAIN: BME-1903019126
MD. HASIBUR RAHMAN: BME-1903019492
ABDULLAH AL MARUF: BME-1903019450

Department Of Mechanical Engineering

SONARGAON UNIVERSITY (SU)

MAY 2023

CFD ANALYSIS OF HEAT TRANSFER USING COUNTER FLOW SINGLE-PASS SHELL AND TUBE HEAT EXCHANGER

A thesis by

Abul Hasnat Himel: BME-1903019127
Arman Hossain: BME-1903019126
MD.Hasibur Rahman: BME-1903019492
Abdullah Al Maruf: BME-1903019450

Supervised by

MD. MAHEDY HASAN

Assistant Professor

Department of Mechanical Engineering

Submitted to the
DEPARTMENT OF MECHANICAL ENGINEERING
SONARGAON UNIVERSITY (SU)
In partial fulfillment of the requirements for the award of the degree
of
Bachelor of Science in Mechanical Engineering

MAY 2023

Acknowledgement

Almighty God is to be praised and thanked for allowing this study to continue. The assistant professor at Sonargaon University's (SU) Department of Mechanical Engineering, **Md. Mahedy Hasan** , sir has our sincere gratitude for his insightful and helpful advice throughout the design and development of the research activities. His desire to voluntarily donate his personal time has been really encouraging. Special gratitude should be extended to each and every group member for their collaboration and assistance; without their support, the thesis work could not have been finished within the allotted time. Finally, the writers would like to express their gratitude to all those who have helped and encouraged us throughout this task.

Abstract

This study uses the CFD analysis program ANSYS fluent to investigate the heat transfer of a counter flow single pass Shell and Tube Heat Exchanger (problem statement). With the aid of ANSYS fluent, a simulation of conjugate heat transfer is performed. SolidWorks CAD software is utilized for the 3D design of the issue statement, and the geometry is added to ANSYS fluent CFD Solver to construct the boundary conditions to solve the problem. When the inlet temperatures are 15 °C and 90°C, and the fluid is flowing in the opposite direction, water is employed as the working fluid in the Tube and Shell fluid domain. To determine whether or not the heat transfer is improved, a simplified model of a counter-flow plain tube heat exchanger (Body of copper) is contrasted with the desired model of a heat exchanger, which has a (Body of steel) placed evenly along the length of the tube. Fluid outlet temperature is measured using the experimental data all during the CFD simulation. To support the increase in heat transfer, LMTD is also calculated and compared with a standard case or model. It is discovered from the experiment that the LMTDs of plain tubes with copper and steel bodies are 54.89°C and 55.25°C, respectively.
Since, LMTD of the **Copper** and **Steel** body tube counter-flow heat exchanger is approximately same, So the finding is said that there is tiny difference between the body of those.

Table of Contents

Acknowledgement		ii
Abstract		**iii**
Table of contents		**iv-vi**
List of Tables		**vi**
List of Figures		**vii**
Chapter 1	Introduction and Objectives.	01
	1.1 Objectives	
	--	
	--	
Chapter 2	Literature Review.	02-08
	2.1 Background and Recent Approaches.	02-03
	2.2 C.F.D (Computational Fluid Dynamics)	04
	2.3 ANSYS fluent as a CFD solver	04
	2.4 Different Types of Fluid Flow	05-06
	2.5 Dimensionless Numbers	07
	2.6 Heat Exchanger	08
	2.7 Applications of Heat Exchangers	08
	2.8 Heat Transfer Enhancement	08
	2.9 Use of CFD in Heat Transfer Enhancement of Shell and Tube Heat Exchanger	08
	--	
	--	

Chapter 3	Problem Statement	09-14
	3.1 Physical Modeling of the Problem	09
	3.2 Geometry	10
	3.3 Required calculation for boundary condition	11
	3.4 Mesh	13
	3.5 Setup	14
	--	
	--	
Chapter 4	Result and Discussion	15-21
	4.1 Temperature Contour	15-16
	4.2 Fluid stream line temperature of hot fluid and cold fluid domain.	17
	4.3 Velocity contour and Streamline	18-19
	4.4 Node based Temperature	20
	4.5 LMTD Calculation	21
	4.6 Discussion	21
	--	
	--	

List of Contents

Chapter 5	Conclusion	22

List of Tables

	Page no.
Table no.1 Dimensions of the model	10
Table no.2 Boundary conditions	12
Table no.3 Nodes base temperature	20
Table no.4 Fluid inlet and outlet temperature	21

List of Figures

Figure No.3: Schematic diagram of problem statement 09

Figure No.4: Cross- Section view 09

Figure No.5: 3D view of Heat exchanger 10

Figure No.6: Shell wall 13

Figure No.7: Tube wall 13

Figure No.8: Temperature contour of Plain tube heat exchanger 15

Figure No.9: Temperature contour of Plain tube heat exchanger along the length.(Copper body) 16

Figure No.10: Temperature contour of Plain tube heat exchanger along the length(Steel body) 16

Figure No.11: Fluid Streamline Temperature of Plain tube Counter Flow Heat exchanger.(Copper body) 17

Figure No.12: Fluid Streamline Temperature of Plain tube Counter Flow Heat exchanger.(Steel body) 17

Figure No.13: Velocity contour (Copper Body) 18

Figure No.14: Velocity contour (Steel Body) 18

Figure No.15: Velocity Streamline of fluid (copper body) 19

Figure No.16: Velocity Streamline of fluid (Steel body) 19

Figure No.17: Single point node 20

Figure No.18: Pointed nodes 20

Chapter 1

INTRODUCTION

We are living in an era of technological advancement. Due to technological advancement in our lifestyle and increment of population resulting soaring of energy crisis. To save energy and making devices more efficient, devices are getting compact in size. Heat exchangers are the most common type of device that is used in different engineering and industrial applications. Among various types of heat exchangers **Shell and Tube Heat Exchanger** have wide range of applications ranging from Power, Gas, Automotive and Aviation to various industrial sectors. Convective heat transfer plays a pivotal role in Shell and Tube heat exchangers. To make the heat exchangers most efficient in convective heat transfer different heat transfer augmentation or enhancement techniques are employed. That could be Active, Passive or Compound technique of heat transfer enhancement. To investigate effectiveness of heat transfer enhancement technique in the heat exchanger different experimental and CFD simulations are done around the globe by different thermo-fluid researchers. The augmentation of heat transfer can be achieved by increasing the surface area, comparing different body metal, roughness and by inducing disturbance in boundary layer, which are the methods of passive techniques. However, active techniques of heat transfer need external source of power to accelerate the heat transfer rate on the other hand, passive methods of enhancing the heat transfer doesn't require external power source to accelerate the heat transfer.

1.1 Objectives

In our experiment a CFD simulation of Conjugate heat transfer is done on a counter flow single pass heat exchanger. Objectives of this experiment are:-

- To investigate the heat transfer through heat exchanger.
- To evaluate the performance of the heat exchanger.

Chapter 2

LITERATURE REVIEW

Heat is defined as the amount of energy a substance has. A heat exchanger is a heat transfer device whose purpose is the transfer of energy from one moving fluid stream to another moving fluid stream [1]. The transfer of heat is done with a separation of a wall to avoid mixing of the two fluids of different properties.

Transfer of heat happens by three principles means which are Radiation, Conduction and Convection. Conduction occurs as the heat from the higher temperature fluid passes through the solid wall. To maximize the heat transfer, the wall should be thin and made of a very conductive material. Forced convection in the other hand allows for the transfer of heat of one moving stream to another moving stream [2].The convection and conduction each has its own thermal resistance and can be calculated by the formula after finding the enthalpy and the cross sectional area of the tube .The way to improve heat transfer performance is referred to as heat transfer enhancement. Nowadays a significant number of thermal engineering researchers are seeking for new methods of enhancing heat transfer between surface and the surrounding fluid [3].

There are so many arrangements are used in experimental approaches to get better heat transfer rate such as different tube shape i.e., plain tube, tube with fins, corrugated tube etc.[4]. The most important part of Heat Exchanger is condenser (or) evaporator so changing the material of shell and tube giving best transfer rate and good thermal efficiency. Even though material properties develops essential once conversation a huge quantity of hotness within minimum specified timing in order to give manufacture standards and secure period budget[5]. Generally CFD is one the simulation software which analysis the flow of fluid passing through exchangers and pipes and modeling of CFD to the examination of flow and transfer rate in crenelated plate heat exchanger [6].

2.1 Background and Recent Approaches

Heat transfer enhancement is the process of increasing the effectiveness of heatexchangers. Heat transfer enhancement concerns the improvement of thermal performance of any heat transport process, heat exchanging medium, component, device or equipment [7].

Methods of heat transfer enhancement techniques:-

1. Active
2. Passive
3. Compound

Active Technique: Active techniques requires external forces, i.e., electric field, acoustic or surface vibration etc.. Yang xia experimental studied of active technique in which the electro hydro dynamics (EHD) effect is used[8].A few external power sources were needed for enhancing the heat transfer for instance, fluid's vibration, fluid's stirring using magnetic field for disturbing particles related to flowing fluid, introduction pulsating regarding cam as well as reciprocation plunger[9].The active method involves external power input for the enhancement in heat transfer. For examples, it includes mechanical aids and the use of magnetic field to disturb the light seeded particles in a following stream etc[10].

Passive Technique: The passive heat transfer augmentation methods does not need any external power input. By using this technique causes the swirl in the bulk of the fluids and disturbs the actual boundary layers which increase effective surface area, residence time and simultaneously heat transfer co-efficient increases in an existing system[11].Twisted tapes, wire coils, ribs, fins, dimples, etc., are the most commonly used passive heat transfer augmentation tools. In recent studies, emphasis is given to works dealing with twisted tapes and wire coils because, according to recent studies, these are known to be economic heat transfer augmentation tools.

Compound Techniques: A compound technique consists of the combination of more than one heat transfer enhancement method (active and/or passive) to increase the thermos hydraulic performance of heat exchangers. It can be employed simultaneously to generate an augmentation that promotes the performance of the system either of the techniques operating independently. Preliminary studies on compound passive augmentation technique of this kind are quite encouraging [12].

2.2 C.F.D (Computational Fluid Dynamics)

Computational fluid dynamics is the field of study devoted to solution of the equations of fluid flow through use of computer. CFD is then employed to shorten the design cycle through carefully controlled parametric studies thereby reducing the required amount of experimental testing [13].

A qualitative and quantitative prediction can be put together with the help of CFD which uses mathematical modeling tools; numerical computation and software tools to device understand contract and therefore, predict the required scenarios. For complex modeling geometry, CFD provides quick results when associated with physical modeling techniques as they would take more time space and costs.

CFD allows for the study of in-depth queries in a more structured and visualized manner CFD also used for controlled simulation that are not created in real life situations. CFD gives the possibility to analyze different problems which are very difficult.

CFD can be used in different domains of science which include maximizing yield of equipment, modeling, ventilation and air flow for cooling purpose, optimizing strategies for oil discovery and extraction. CFD simulation is frequently used are Aerospace/Aeronautics, Manufacturing process Engineering, oil and gas industry and Turbo Machinery etc. CFD Modeling is used for deriving most accurate options for designing and enhancing the efficiency level of HVACs system. Application of CFD is not limited to the aforementioned areas therefore, extensively applied in various field of study [14].

2.3 ANSYS Fluent as a CFD solver

Ansys fluent is the industry leading fluid simulation software known for its advance physics modeling capabilities and industry leading accuracy. Ansys fluent is known for its power; simplicity and speed which has helped make it a world leader in CFD software both in academic and industry. Ansys fluent gives us more time to innovate and optimize product performance. With Ansys fluent, we can create advanced physics models and analyze a variety of fluid phenomena all in a customizable and intuitive space. To better understand the mathematical models being applied, well validate the result from Ansys fluent with numerical solutions calculated using mathematical.

1. Defining the modeling goals.

2. Setting up the solver and physical model.

3. Computing and monitoring the solution.

4. Examining and saving the results.

2.4 Different types of fluid flows

1. Steady and unsteady flows.

2. Laminar and Turbulent flows.

3. Compressible and incompressible flows.

Steady and Unsteady Flows

1.Steady flow: The type of flow in which the fluid characteristics like velocity, pressure, density, etc. at a point do not change with time is called steady flow.

Example - Flow through a prismatic or non-prismatic conduit at a constant flow rate Q m3/s is steady.

2.Unsteady flow: It is that type of flow in which the velocity, pressure or density at a point change with respect to time.

Example-The flow in a pipe whose valve is being opened or closed gradually.

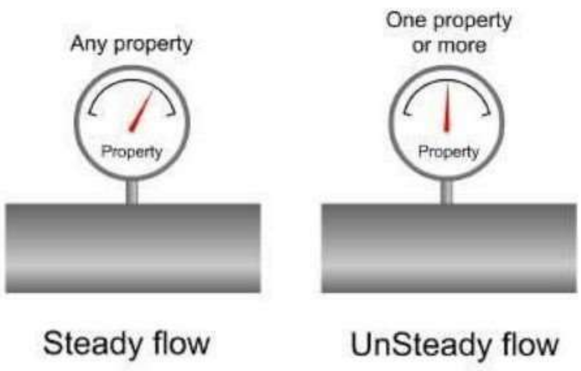

Figure No.1: Steady and unsteady flow

Laminar and Turbulent Flows

3.Laminar flow: A laminar flow is one which paths take by the individual particles do not cross one another and move along well-defined paths This type of flow is also called stream-line flow or viscous flow.

Examples.

(i) Flow through a capillary tube.

(ii) Flow of blood in veins and arteries. (iii) Ground water flow.

4.Turbulent flow: A turbulent flow is that flow in which fluid particles move in a zig-zag way.

Example. High velocity flow in a conduit of large size. Nearly all fluid flow problems encountered in engineering practice have a turbulent character.

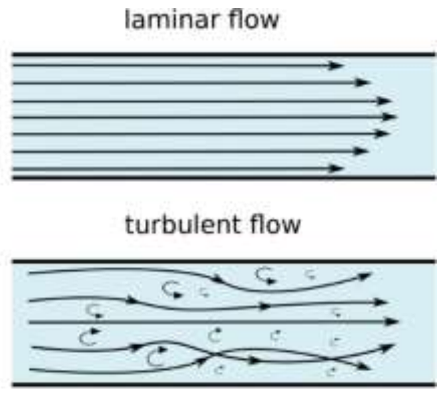

Figure No. 2: Laminar and turbulent flow

Laminar and turbulent flows are characterized on the basis of Reynolds number.

For Reynolds number (Re) <2000 flow in pipes is laminar flow.

For Reynolds number (Re) >4000 flow in pipes is turbulent flow.

For Reynolds number between 2000 and 4000 flow in pipes transitional flow.

2.5 DIMENSIONLESS NUMBERS

1. Reynolds Number

2. Prandtl Number

Reynolds Number**:-** It is defined as the ratio of the inertia force to the viscous force.

$$RE = \frac{\text{inertia force}}{\text{viscous force}} = \frac{\rho u L}{\mu} = \frac{uL}{V}$$

Where:

- ρ is the density of the fluid (SI units: kg/m^3)
- u is the velocity of the fluid with respect to the object (m/s)
- L is a characteristic linear dimension (m)
- μ is the dynamic viscosity of the fluid
- v is the kinematic viscosity of the fluid (m^2/s)

Newtonian Fluid Reynolds Number (RE) Formula

$$RE = \frac{\rho v D}{\mu}$$

- μ- Fluid dynamic viscosity in kg/ (m.s)
- ρ - Fluid density in kg/m^3
- V - Fluid velocity in m/s
- D - pipe diameter in m.

Prandtl Number : It is the ratio of kinematic viscosity (V) to thermal diffusivity(α).

$$Pr = \frac{\mu C_p}{k} = \frac{\rho v C_p}{k} = \frac{v}{k/\rho C_p} = \frac{v}{\alpha}$$

- μ - is the dynamic viscosity
- k - is the thermal conductivity
- C_p- is the specific heat

2.6 Heat Exchanger

A heat exchanger is a system used to transfer heat between a source and a working fluid. Heat exchangers are used in both cooling and heating processes. The fluids may be separated by a solid wall to prevent mixing or they may be in direct contact.

Shell and Tube Heat Exchanger: A shell and tube heat exchanger is a class of heat exchanger designs. It is the most common type of heat exchanger in oil refineries and other large chemical processes, and is suited for higher-pressure applications. As its name implies, this type of heat exchanger consists of a shell (a large pressure vessel) with a bundle of tubes inside it. One fluid runs through the tubes, and another fluid flows over the tubes (through the shell) to transfer heat between the two fluids. The set of tubes is called a tube bundle, and may be composed of several types of tubes: plain, longitudinally finned, etc. The shell inlet is at the top rear and outlet in the foreground at the bottom.

2.7 Application of Heat Exchangers

These heat exchanger tubes are mostly used in power plants, chemical factories, ship buildings, fertilizer plants, refineries, steel plants, etc. Their application is usually in high pressure equipment, air pre heaters, air coolers, pressure vessels, heat exchanger tube sheets, boilers, cryogenic pressure vessels, finned tubes, etc.

2.8 Heat Transfer Enhancement

Nowadays a significant number of thermal engineering researchers are seeking for new methods of enhancing heat transfer. The way to improve heat transfer performance is referred to as between surface, changing body metal and the surrounding fluid.

2.9 Use of CFD for analysis the shell and tube heat exchanger

A complex computationally based design and analysis method is CFD. Through computer modeling, CFD software enables you to simulate fluid-structure interaction, acoustics, multiphase physics, chemical reaction, heat and mass transfer, moving bodies, and flows of gases and liquids. Before applying real-world physics and chemistry to the model, this program can also create a virtual prototype of the system or device. It then outputs visuals and data that forecast how well the design will function. CFD tool is used to study performance and how heat exchanger acts in convective heat transfer manner as a result of design changes for heat exchanger optimization. This led to the selection of an efficient heat exchanger with a modified design for comparison of heat transfer.

Chapter 3

PROBLEM STATEMENT

A single pass counter flow heat exchanger is investigated with the help of CFD solver ANSYS fluent. The physical configuration has a single tube and a shell or outer jacket of the heat exchanger filled with cold and hot fluid domain (water) respectively. Here, for heat transfer comparison we have taken copper and steel as body of shell and tube wall.

3.1 Physical modeling of the problem.

The length of the tube and shell are 5m and 4m respectively. The inner and outer diameter of the tube is 0.2m and 0.3m. In terms of shell, the inner and outer diameter 0.9m and 1m respectively. The height of the shell inlet pipe is 0.5m and the inner and outer diameter is 0.2m and 0.3m respectively.

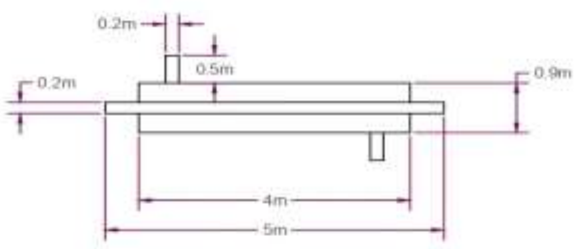

Figure No.3: Schematic diagram of problem statement (side view)

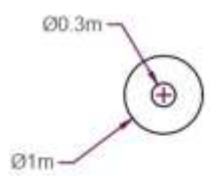

Figure No.4: Cross-Section View

3.2 Geometry

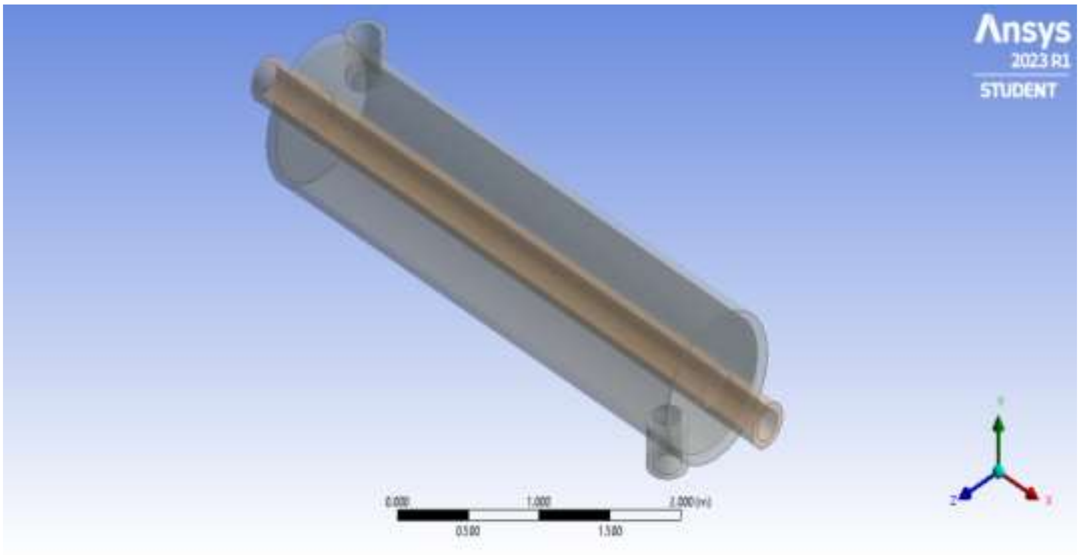

Figure No.5: 3D view of Heat Exchanger

Table no.1 Dimensions of the model:

Parameters	Shell	Tube	Inlet Outlet Parts
Internal Dia (D_i)	0.9m	0.2m	0.2m
Externa Dia (D_o)	1m	0.3m	0.3m
Thickness (ε)	0.1m	0.1m	0.1m
Length (L)	4m	5m	0.5m

3.3 Required calculation for boundary condition

Mass flow rate (cold fluid, water)

Inlet port dia, D_p = 0.2m

Cross section area of inlet port, $A_p = \frac{\pi Dp2}{4} = \frac{\pi\times(0.2)2}{4} = 3.142\times10^{-2} m^2$

$m_c = \boldsymbol{\rho} v A_p$

$= 998.2\times1\times3.142\times10^{-2}$

$= 31.36$ kg/s

Mass flow rate (Hot fluid, water)

Internal Dia of Tube, D_t = 0.2m

Area of the internal tube, $A_t = \frac{\pi Dt2}{4} = \frac{\pi\times(0.2)2}{4} = 3.142\times10^{-2} m^2$

$m_h = \boldsymbol{\rho} v A_p$

$= 998.2\times1\times3.142\times10^{-2}$

$= 31.36$ kg/s

Operating condition → 26°

Hydraulic diameter for hot fluid,

D_h = (D_i) Shell – (D_o)Tube

=(0.9-0.3)m

=0.6m

<u>Reynolds Number</u>

Hot fluid, $Re = \frac{\rho vDh}{\mu} = \frac{998.2\times1\times0.6}{0.00315} = 190133$

Cold fluid, $Re = \frac{\rho vDt}{\mu} = \frac{998.2\times1\times0.2}{0.00113} = 175507$

Table no. 2 Boundary conditions:

Parameters	Hot fluid (water)	Cold (water)
Initial temperature	$T_h = 90$ ℃	$T_c = 15$ ℃
Mass flow rate (**m**)	$m_h =$ 31.36 kg/s	$m_c =$ 31.36 kg/s
Velocity(**v**)	1m/s	1m/s
Default Value of density(ρ)	998.2 kg/m^3	998.2 kg/m^3
Viscosity(**μ**)	0.00315 kg/m.s	0.00113 kg/m.s
Specific heat (C_p)	4182 j/kg K	4182 j/kg K
Thermal conductivity of water(**K**)	0.6 w/m K	0.6 w/m K
Reynolds number (**Re**)	190133	175507
Thermal conductivity of Cu (**K**)	387.6 w/m K	

3.4 Mesh

Ansys meshing capabilities help reduce the amount of time and effort spent to get to accurate results. Since meshing typically consumes a significant portion of the time it takes to get simulation results , Ansys helps by making better and more automated meshing tools.

Figure No. 6: Shell wall

Figure No. 7: Tube wall

Details about mesh:

- Types : Quadrilateral
- Element Size : 30mm
- Number of nodes : 87938

3.5 Setup

Materials :-

Fluid - Water liquid

Solid - Copper and Steel

Cell zone Condition :-

Fluid — a. Shell Fluid

b. Tube Fluid

Solid — a. Shell Wall

b. Tube Wall

Boundary Condition:-

Inlet:

Hot inlet (90^{o}) For both Solid

Cold Inlet (15^{o}) (Copper and Steel)

Inlet Type: Velocity Inlet

Outlet Type: Pressure Outlet

Methods: K-epsilon Turbulence model.

K-epsilon (k-ε) turbulence model is the most common used in computational fluid dynamics (CFD) to simulate mean flow characteristics for turbulent flow conditions. Since we get result from Reynolds equation where we know that our flow is Turbulence flow that's why we can use the K-epsilon method.

Function: Realizable and Scalable function.

Chapter 4

RESULTS AND DISCUSSION

4.1 Temperature contour

Using Ansys fluent, CFD simulation was done both on plain tube single pass counter flow heat exchanger (copper body and steel body). In figure temperature counter is illustrated through three cross sections at inlet of cold fluied, mid origin of the tube and at the exit of the cold fluid. Warmer section indicates that hot fluid domain on the other hand cold fluid domain is indicated by Blue and Indigo color.

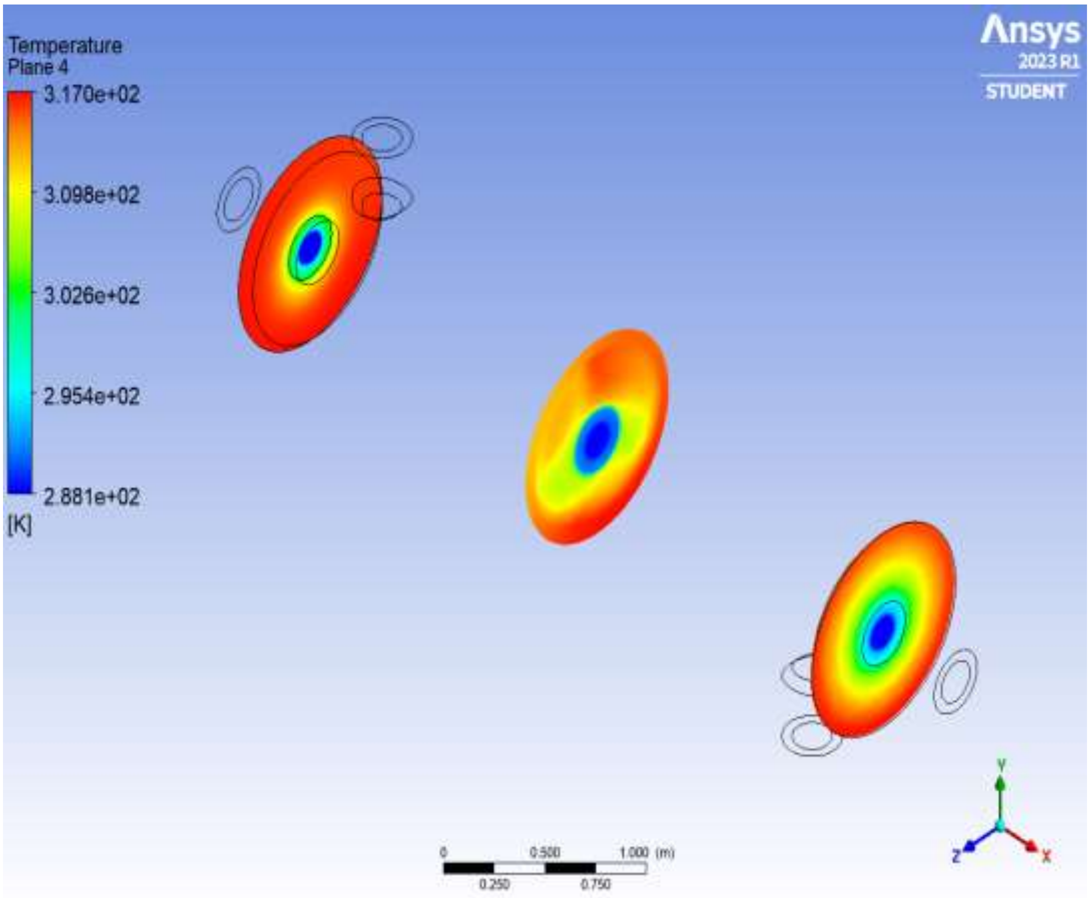

Figure No. 8: Temperature contour of plain tube heat exchanger.

In second picture of temperature contour along the longitudinal section is illustrated. That depicts gradual cool down of fluid at the exit on the contrary, cold fluids get warmed while absorbing the heat from the hot fluid flowing inside the tube. Figure 9 is for plain tube (Copper body) and figure 10 is for plain tube (Steel body) heat exchanger.

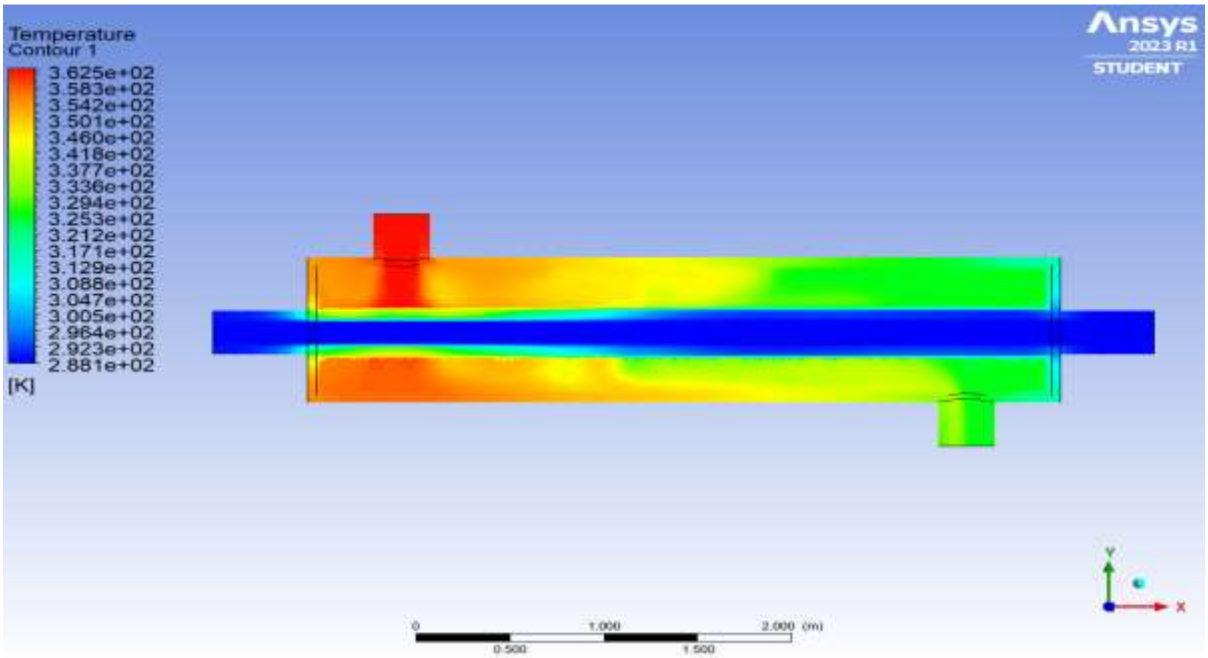

Figure No. 9: Temperature contour plain tube heat exchanger along the length (Copper body)

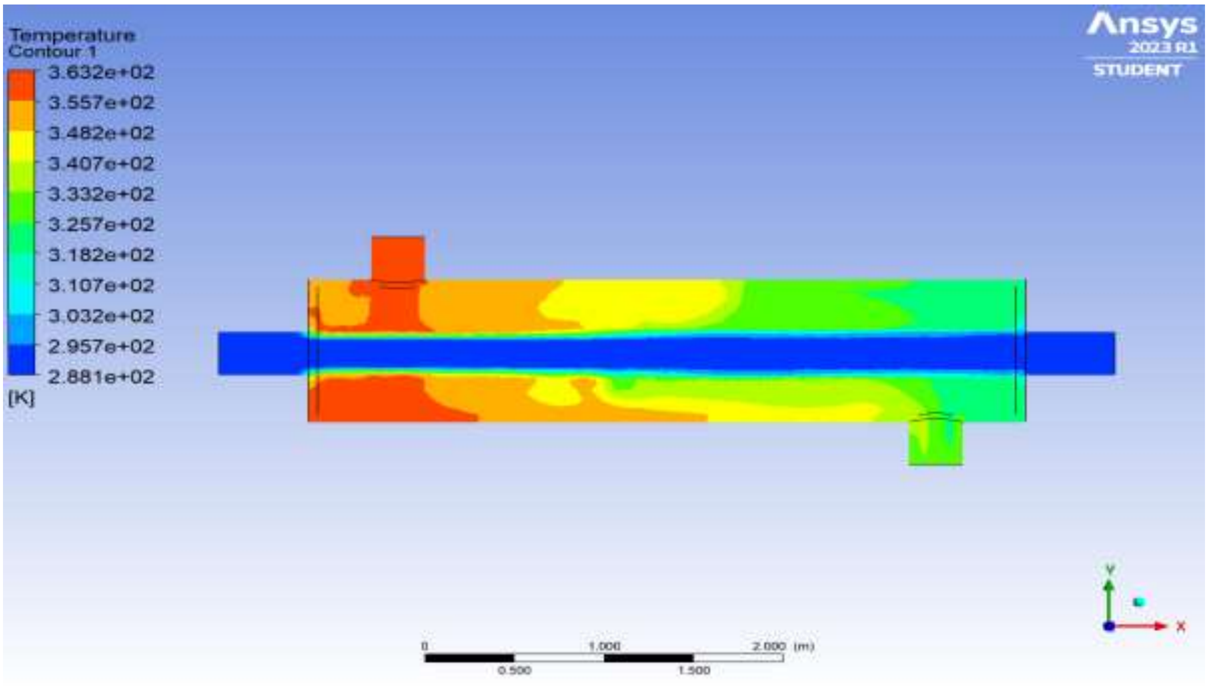

Figure No10:Temperature contour of plain tube heat exchanger along the length (Steel body)

4.2 Fluid stream line temperature of hot fluid and cold fluid domain.

Fluid stream line temperature is illustrated in the following picture. Fluid steam lines are changing their color from inlet to the outlet for both hot and cold fluid stream lines tube and shell zone respectively due to convective heat transfer. Hot fluid and cold fluid enter the heat exchanger at 90º C (363 K) and 15º C(288K) respectively. Figure 11 represents stream lines of plain tube counter flow heat exchanger(Copper body) and 12 represents plain tube counter flow heat exchanger(Steel body).

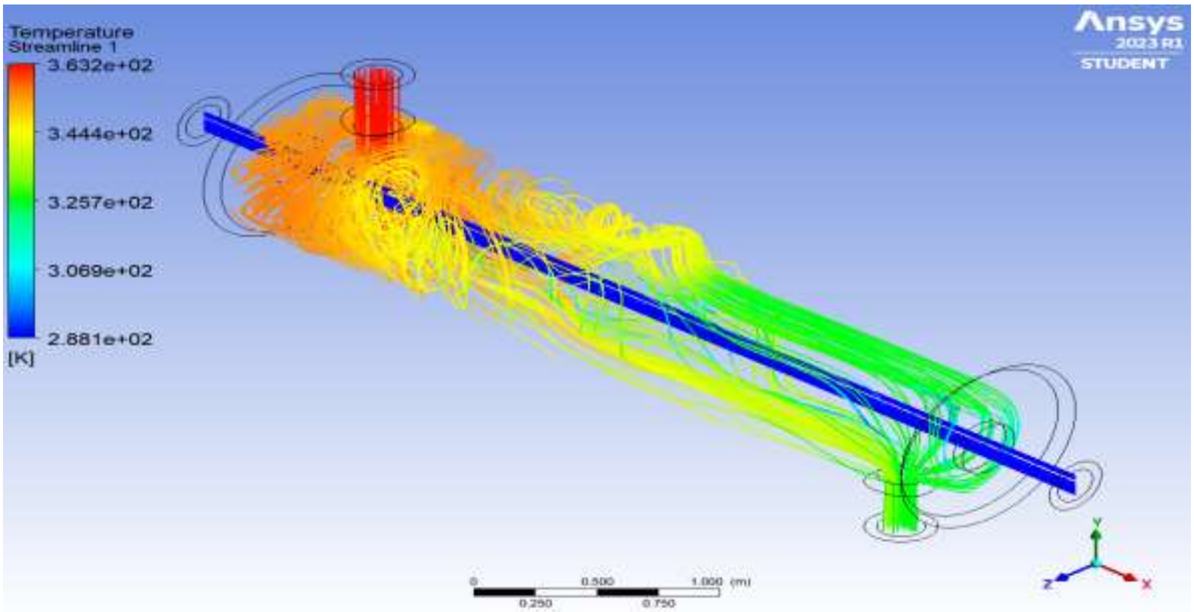

Figure No. 11: Fluid Streamline Temperature of plain tube Counter Flow Heat exchanger (Copper Body)

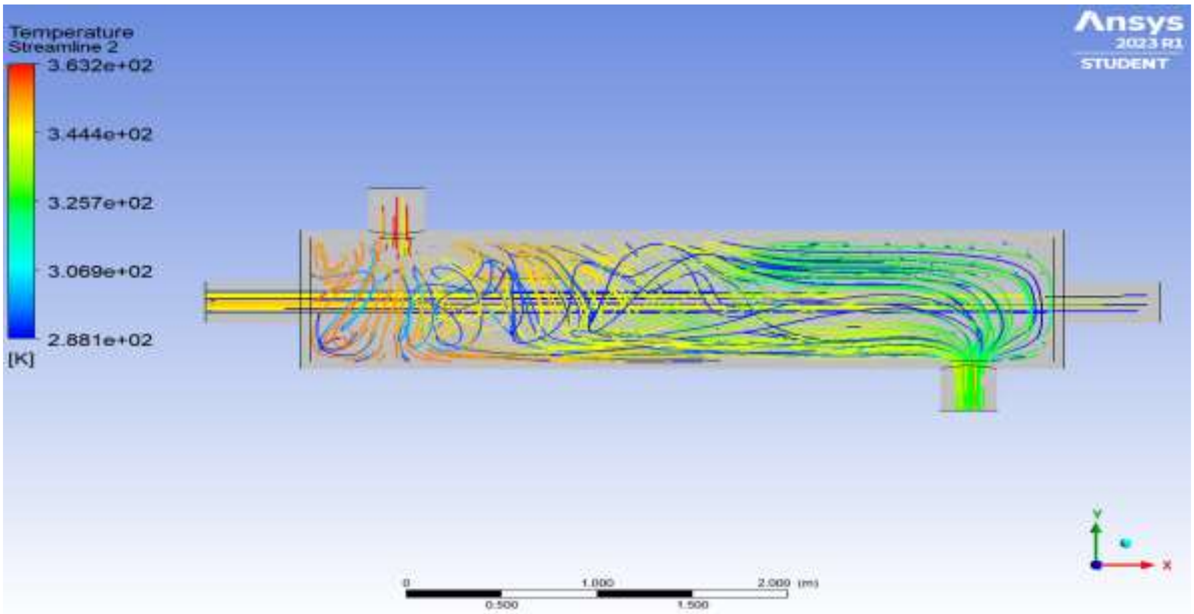

Figure No. 12: Fluid Streamline Temperature of plain tube Counter Flow Heat exchanger(Steel body)

4.3 Velocity Contour and Streamline

The velocity contour is the velocity field for that iteration in steady state. Data sampling can also be done on a steady case where we may see some slight changes in the flow which can then produce an averaged velocity field.

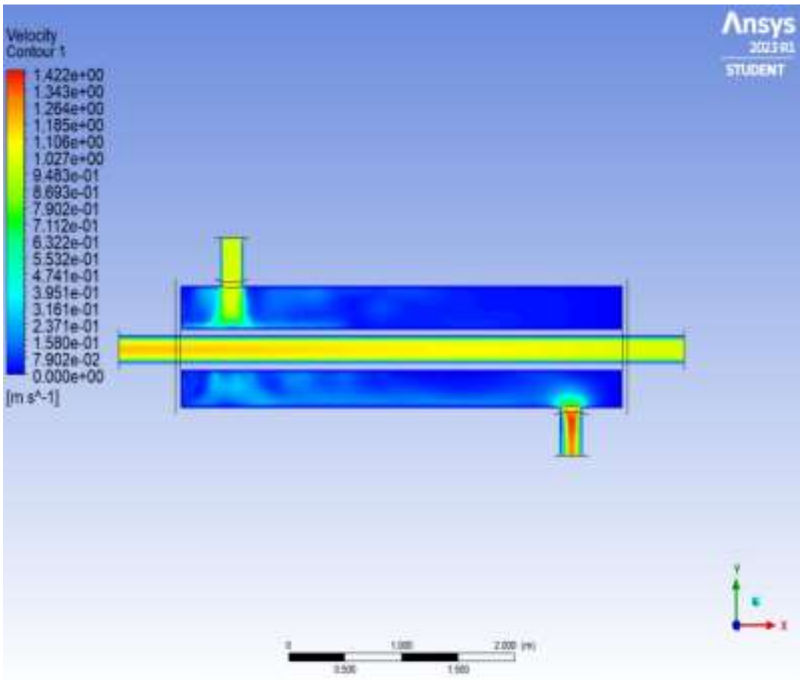

Figure No.13: velocity contour (Copper body)

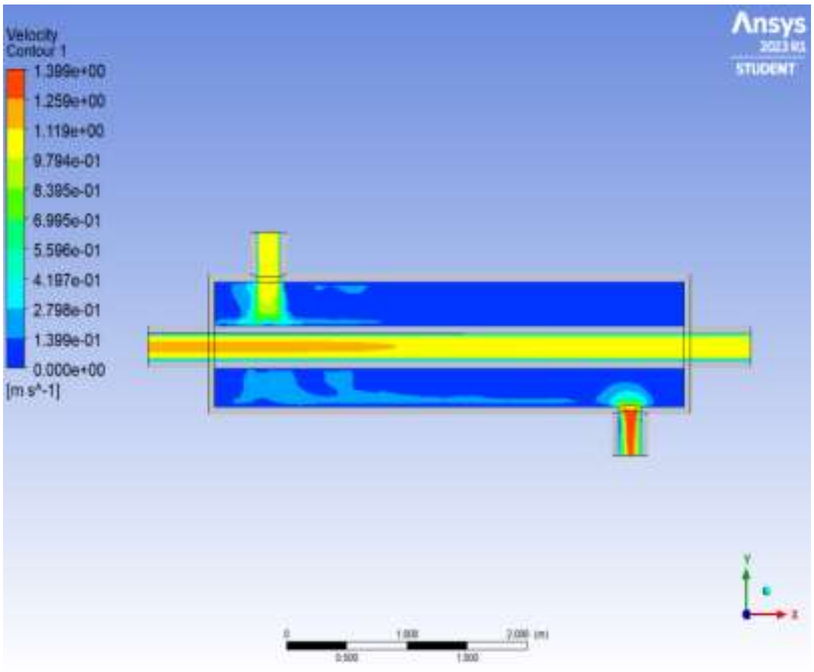

Figure No.14: Velocity contour (Steel body)

Velocity Streamline :

A streamline is a line that is tangential to the instantaneous velocity direction .To visualize this in here, we could imagine the the motion of streamline marked element of fluid.

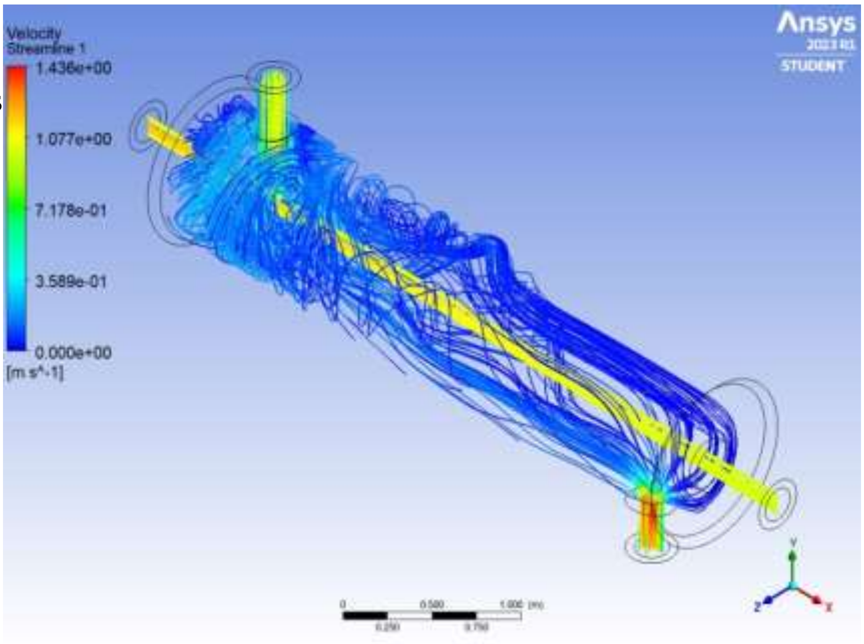

Figure No. 15: Velocity streamline of fluid (Copper body)

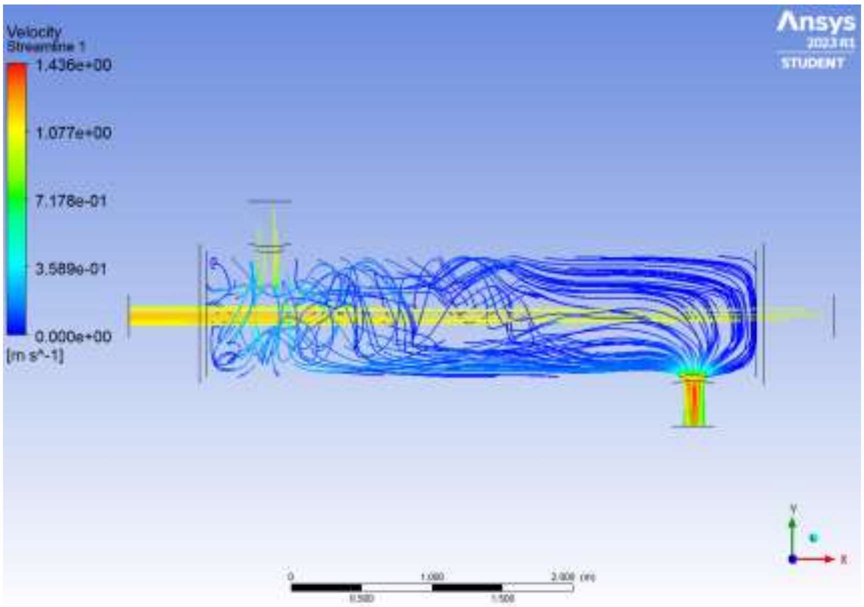

Figure No. 16: Velocity streamline of fluid (Steel body)

4.4 Node based temperature

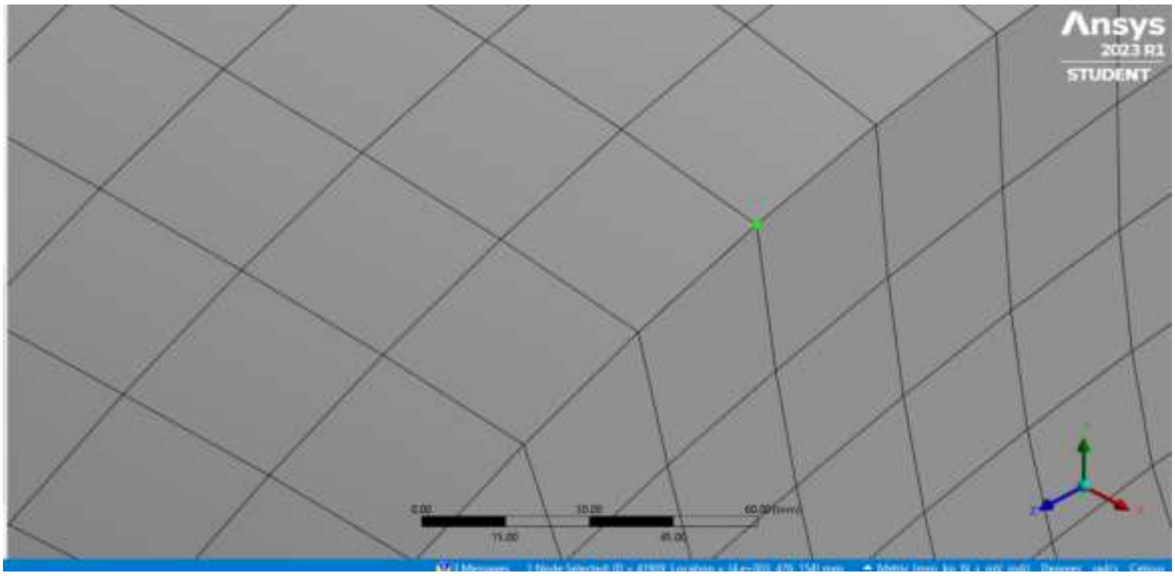

Figure No.17: Single point node

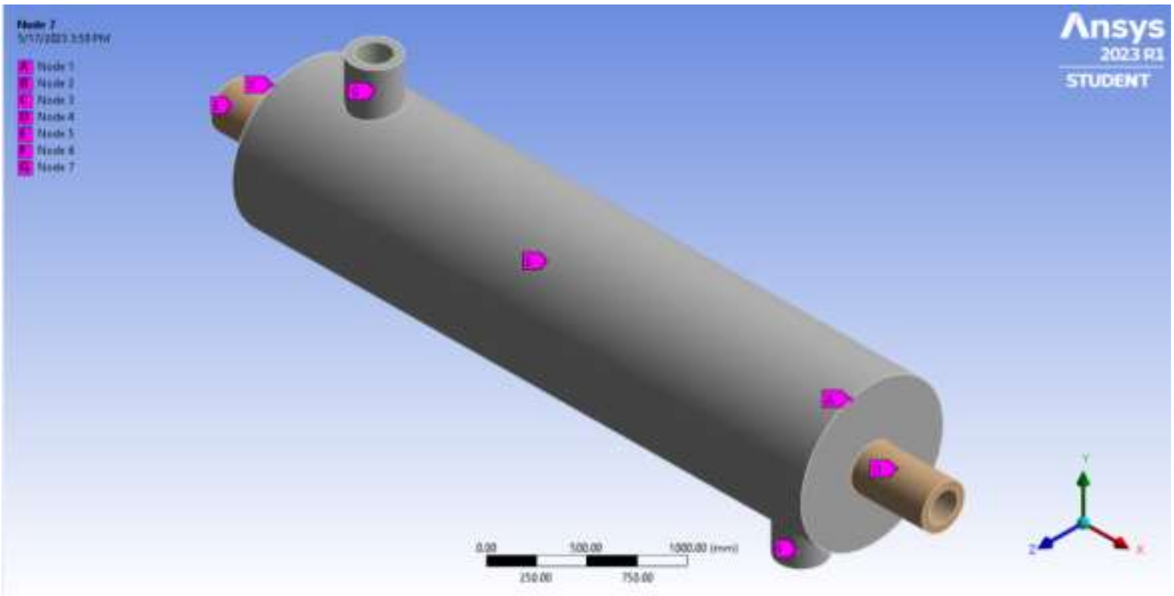

Figure No.18: Pointed Nodes

<u>Table no.3 Node based temperature:</u>

Nodes	Temperature (℃)
Node 1	57
Node 2	72
Node 3	79
Node 4	15
Node 5	15
Node 6	57
Node 7	90

4.5 Logarithmic mean temperature difference (LMTD) Calculation

$$\textbf{LMTD} = \frac{\{T_h in - T_c out\} - \{T_h out - T_c in\}}{\ln\frac{\{T_h in - T_c out\}}{\{T_h out - T_c in\}}}$$

<u>Table no.4 Fluid inlet and outlet temperature.</u>

Plain tube single pass counter flow heat exchanger(copper body)		
(Hot fluid)	Inlet temperature ℃	Outlet temperature ℃
	90	54.41
(Cold fluid)	15	16
Plain Tube single pass counter flow heat exchanger(steel body)		
(Hot fluid)	Inlet temperature ℃	Outlet temperature ℃
	90	54.47
(Cold fluid)	15	15.24

$$\textbf{LMTD} = \frac{\{T_h in - T_c out\} - \{T_h out - T_c in\}}{\ln\frac{\{T_h in - T_c out\}}{\{T_h out - T_c in\}}}$$

$$\text{LMTD of Plain Tube copper body} = \frac{(90-16)-(54.41-15)}{\ln\left(\frac{90-16}{54.41-15}\right)}$$

$$= \mathbf{54.89^0\ C}$$

$$\text{LMTD of Plain Tube Steel body} = \frac{(90-15.24)-(54.47-15)}{\ln\left(\frac{90-15.24}{54.47-15}\right)}$$

$$= \mathbf{55.25^0\ C}$$

4.6 Discussion

As comparing the value of LMTD between two different model we can conclude that, heat transfer of the both model is approximately same.

Chapter 5

CONCLUSION

Enhancing the rate of heat transfer among different Industrial and Consumers application is a mast. Effective working of the devices requires cooling as being heated up it declines the working efficiency of the device. Active, passive and compound techniques are employed for the augmentation of heat transfer. In our research approach we used (copper and steel) two different body and in conjunction with CFD solver we investigate the effectiveness of the model. However, it seems same as the result.

REFERENCES

[1] Y. Xue, Z. Ge, X. Du, and L. Yang, On the heat transfer enhancement of plate fin heat exchanger, 2018. doi:10.20944/preprints201804.0304.v1

[2] D. D. Ganji and A. Malvandi, “Introduction to heat transfer enhancement,” Heat Transfer Enhancement Using Nano fluid Flow in Micro channels, pp. 1–12, 2016. doi:10.1016/b978-0-323-43139-2.00001-0

[3] H. Abu-Mulaweh, “Experimental comparison between heat transfer enhancement methods in heat exchangers,” 2001 Annual Conference Proceedings, 2001. doi:10.18260/1-2--9259

[4] L. J. Yang, J. Ren, X. Z. Du, D. Y. Liu, and Y. P. Yang, “Convection heat transfer active enhancement by magnetically induced longitudinal vortices,” Heat Transfer Enhancement, 2006. doi:10.1615/ihtc13.p17.30

[5] A. Sharma, “Introduction to CFD: Development, application, and analysis,” Introduction to Computational Fluid Dynamics, pp. 19–33, 2021. doi:10.1007/978-3-030-72884-7_2

[6] H. Santos, W. Li, and D. Kukulka, “CFD study of the thermal performance improvement of a counter-flow heat exchanger using enhanced surface tubes,” ASME 2021 Heat Transfer Summer Conference, 2021. doi:10.1115/ht2021-63577

[7] J. C. Tannehill, D. A. Anderson, and R. H. Pletcher, Computational Fluid Mechanics and Heat Transfer. Washington, DC, Columbia: Taylor & Francis, 1997.

[8] R. S. Khurmi and J. K. Gupta, A Text Book of Thermal Engineering: S.I. Units. New Delhi, India, Uttar pradesh: S. Chand & Co., 2001.

[9] P. A. Longwell, Mechanics of Fluid Flow. New York, Northeastern: McGraw-Hill, 1966.

[10] C. Rey and J.-M. Rosant, “Reynolds number and Prandtl number influence on the determination of isotropic velocity and temperature turbulent length scales,” Advances in Turbulence, pp. 209–214, 1987. doi:10.1007/978-3-642-83045-7_25

[11] Y. A. Cengel and J. M. Cimbala, Fluid Mechanics: Fundamentals and Applications. New York, NY, north eastern: McGraw-Hill Education, 2018.