TRIBHUVAN UNIVERSITY

INSTITUTE OF ENGINEERING

PULCHOWK CAMPUS

THESIS NO.: M-132-MSTIM-2020-2023

CFD Analysis of Shell and Tube Heat Exchanger with Different Number of Baffles and Baffle Cut

by

Shaurav Suwal

A THESIS

SUBMITTED TO THE DEPARTMENT OF MECHANICAL AND AEROSPACE ENGINEERING IN PARTIAL FULFILLMENT OF THE REQUIREMENTS FOR THE DEGREE OF MASTER OF SCIENCE IN ENGINEERING IN TECHNOLOGY AND INNOVATION MANAGEMENT

DEPARTMENT OF MECHANICAL AND AEROSPACE ENGINEERING

LALITPUR, NEPAL

OCTOBER, 2023

COPYRIGHT

TRIBHUVAN UNIVERSITY

INSTITUTE OF ENGINEERING

PULCHOWK CAMPUS

DEPARTMENT OF MECHANICAL AND AEROSPACE ENGINEERING

The undersigned certify that they have read, and recommended to the Institute of Engineering for acceptance, a thesis entitled " **CFD Analysis of Shell and Tube Heat Exchanger with Different Number of Baffles and Baffle Cut**" submitted by Shaurav Suwal in partial fulfillment of the requirements for the degree of Master in Technology and Innovation Management

____

Supervisor, Dr. Sanjeev Maharjan
MSTIM Program Coordinator
Department of Aerospace and Mechanical Engineering

____

Under Secretary, Er. Asbina Baral
External Examiner
Ministry of Education, Science and Technology

____

Committee Chairperson, Dr. Sudip Bhattrai
Head of Department
Department of Aerospace and Mechanical Engineering

Date: October 12, 2023

ABSTRACT

This thesis emphasizes on the effects of number of baffles and baffle cut for a simple shell and tube heat exchanger. Heat transfer between separate medium can result in heat loss to the environment and inefficiency in heat transfer. So, in order to minimize losses and improve heat transfer properties, corrective action is needed. This is where baffle spacing and baffle cut plays its role and improves the heat transfer phenomena.

The objective of this thesis is to design a shell and tube heat exchanger with different number of baffles and baffle cut inside a shell and tube heat exchanger and analyze the flow, temperatures and pressure drop inside the shell using ANSYS software. The process in analyzing consists of modelling, meshing and simulating the basic geometry of shell and tube heat exchanger using computational Fluid Dynamics (CFD) package ANSYS. The geometry consists of seven tubes within the shell where water is considered as the working fluid and only shell side flow characteristics is observed. Three different number of baffles (8, 6 and 4) with three different baffle cuts (25%, 35% and 45%) are used for the analysis. According to the results of the CFD, the number of baffles and baffle cut plays an important role in the heat transfer and pressure drop. The highest heat transfer was achieved for the shell and tube heat exchanger with 25% baffle cut and 8 number of baffles and the lowest heat transfer was achieved with the 35% baffle cut and 4 number of baffles. The heat transfer is 12.22% more in case of 8 number baffles with 25% baffle cut compared to 4 number baffles with 35% baffle cut.

ACKNOWLEDGEMENT

I am indebted to the Department of Mechanical and Aerospace Engineering, Pulchowk Campus, Institute of Engineering for accepting the thesis proposal on "CFD Analysis of Shell and Tube Heat Exchanger with Different Number of Baffles and Baffle Cut" and giving me an opportunity to undertake the thesis.

I am grateful to Prof. Dr. Sanjeev Maharjan, for his guidance and constant supervision as well as for providing necessary information regarding the project and also for his support in completing the project.

I appreciate supports from our colleagues and all the staffs of Department of Mechanical and Aerospace Engineering in developing the thesis.

Lastly, I would like to thank all the teachers, seniors and friends who helped me directly or indirectly during my thesis completion. It wouldn't have come to fruitful end without their kind cooperation.

TABLE OF CONTENTS

COPYRIGHT ii
APPROVAL ii
ABSTRACT iv
ACKNOWLEDGEMENT v
TABLE OF CONTENTS vi
LIST OF TABLES viii
LIST OF FIGURES ix
LIST OF SYMBOLS xi
CHAPTER ONE: INTRODUCTION 1
1.1. Background 1
1.2. Problem Statement 2
1.3. Objectives 3
1.3.1. Main Objective: 3
1.3.2. Specific Objective: 3
CHAPTER TWO: LITERATURE REVIEW 4
2.1 Types of Heat Exchanger 4
2.1.1. Nature of heat exchange process 4
2.1.2 Relative direction of fluid motion 5
2.1.3 Design and constructional features: 6
2.1.4 Physical state of fluids 6
2.2 Segmental Baffles 7
2.2.1. Single Segmental Baffle: 7
2.2.2. Double Segmental Baffle: 8
2.2.3. Triple Segmental Baffle: 8
2.3 Disc and Ring Baffle: 9
2.4 Orifice Baffle: 9
2.5 Baffle Cut 9
2.6 Baffle Spacing 10

CHAPTER THREE: RESEARCH METHODOLOGY 16
3.1 Pre-processing 17
3.1.1 Geometry Development 17
3.1.2 Mesh Generation.. 19
3.1.3 Boundary Conditions 20
3.2 Solving or Processing 20
3.3 Post-processing... 20
CHAPTER FOUR: RESULTS AND DISCUSSION 22
4.1 Design of shell and tube heat exchanger 22
4. 2 Simulation of shell and tube heat exchanger 22
4.2.1 Simulation with 8 number of baffles 22
4.2.2 Simulation with 6 number of baffles 28
4.2.3 Simulation with 4 number of baffles 34
4.3 Validation of Results 40
4.4 Effect of number of baffles and baffle cut on heat transfer 41
4.5 Effect of number of baffles and baffle cut on shell side pressure drop.................. 42
CHAPTER 5: CONCLUSIONS AND RECOMMENDATIONS 44
5.1 Conclusions 44
5.2 Recommendations 45
REFERENCES 46
APPENDIX... 48

LIST OF TABLES

Table 3. 1 Design Parameters 17
Table 3. 2 Properties of water 17
Table 3. 3 Properties of aluminum.. 17

LIST OF FIGURES

Figure 3. 1 Flow Chart 16
Figure 3. 2 Shell and tube with 8 number of baffles 18
Figure 3. 3 Shell and tube with 6 number of baffles 18
Figure 3. 4 Shell and tube with 4 number of baffles 19
Figure 3. 5 Overall Meshing 19
Figure 3. 6 Magnified view of meshing 20
Figure 4. 1 Velocity contour at 25% baffle cut 23
Figure 4. 2 Streamline at 25% baffle cut 23
Figure 4. 3 Velocity contours at 35% baffle cut 24
Figure 4. 4 Streamline at 35% baffle cut 24
Figure 4. 5 Velocity contour at 45% baffle cut 25
Figure 4. 6 Streamline at 45% baffle cut 25
Figure 4. 7 Temperature contour at 25% baffle cut 26
Figure 4. 8 Temperature contour at 35% baffle cut 26
Figure 4. 9 Temperature contour at 45% baffle cut 27
Figure 4. 10 Pressure contour at 25% baffle cut 27
Figure 4. 11 Pressure contour at 35% baffle cut 28
Figure 4. 12 Pressure contour at 45% baffle cut 28
Figure 4. 13 Velocity contour at 25% baffle cut 29
Figure 4. 14 Streamline at 25% baffle cut 29
Figure 4. 15 Velocity contour at 35% baffle cut 30
Figure 4. 16 Streamline at 35% baffle cut 30
Figure 4. 17 Velocity contour at 45% baffle cut 31
Figure 4. 18 Streamline at 45% baffle cut 31
Figure 4. 19 Temperature contour at 25% baffle cut 32
Figure 4. 20 Temperature contour at 35% baffle cut 32
Figure 4. 21 Temperature contour at 45% baffle cut 33
Figure 4. 22 Pressure contour at 25% baffle cut 33
Figure 4. 23 Pressure contour at 35% baffle cut 34
Figure 4. 24 Pressure contour at 45% baffle cut 34
Figure 4. 25 Velocity contour at 25% baffle cut 35
Figure 4. 26 Streamline at 25% baffle cut 35
Figure 4. 27 Velocity contour at 35% baffle cut 36
Figure 4. 28 Streamline at 35% baffle cut 36
Figure 4. 29 Velocity contour at 45% baffle cut 37
Figure 4. 30 Streamline at 45% baffle cut 37
Figure 4. 31 Temperature contour at 25% baffle cut 38
Figure 4. 32 Temperature contour at 35% baffle cut 38

Figure 4. 33 Temperature contour at 45% baffle cut 39
Figure 4. 34 Pressure contour at 25% baffle cut 39
Figure 4. 35 Pressure contour at 35% baffle cut 40
Figure 4. 36 Pressure contour at 45% baffle cut 40
Figure 4. 37 Heat transfer variation with no. of baffles in current research 41
Figure 4. 38 Heat transfer variation with no. of baffles in existing research.................... 41
Figure 4. 39 Pressure drop variation with no. of baffles in current research 41
Figure 4. 40 Pressure drop variation with no. of baffles in existing research................... 41
Figure 4. 41 Heat transfer variation with different number of baffles 42
Figure 4. 42 Heat transfer variation with different baffle cut 42
Figure 4. 43 Pressure drop variation with different number of baffles 43
Figure 4. 44 Pressure drop variation with different baffle cut 43

LIST OF SYMBOLS

Ds	Shell diameter
d	Tube diameter
d_s	Shell inlet diameter
N_t	Number of tubes
L	Heat Exchanger Length
T	Shell side inlet Temperature
t	Tube side inlet temperature
B_c	Baffle cut
N_b	Number of baffles
Q	Heat transferred per unit time
U	Overall heat transfer coefficient
A	Heat transfer area
ΔT_m	Log mean temperature difference
ρ	Density
$\vec{v}$	Velocity
∇	Gradient operator
$\mathscr{p}$	Static pressure
$\bar{\bar{\tau}}$	Viscous stress tensor
E_t	Total Energy
W	Work done

CHAPTER ONE: INTRODUCTION

1.1. Background

Heat exchangers are used to transfer heat between different media. These media may consist of gases, liquids, or a mixture of both. The mediums involved may either be in direct contact or separated from each other by solid walls to avoid mixing. Heat exchangers transfer heat from systems that don't need it to other systems where it can be used more effectively, increasing a system's energy efficiency. The fluid in a heat exchanger usually moves rapidly through forced convection to improve heat transfer. The fluid's pressure drops as a result of the rapid flow. The efficiency of a heat exchanger refers to how well it transfers heat in relation to the resulting pressure drop. In addition to meeting other design objectives like resistance to high fluid pressures, resistance to contamination and corrosion, and ability to be cleaned and repaired, the most recent heat exchanger technology minimizes pressure drop while optimizing heat transfer (Ipieca, 2022).

The most common and extensively utilized type of heat exchanger is one with a shell and tube design. One fluid passes through the tubes in these exchangers, while the other fluid passes through the shell that covers the tubes. It is possible to arrange tubes inside the shell to permit cross-flow, counter-flow, parallel flow, or both. Large pressures can be handled by this type of exchanger because of its tubular construction. In shell and tube heat exchangers, flow diverters are commonly used to increase the fluid's capacity to transfer heat by increasing the turbulent flow of the shell-side fluid. Common flow diverters include rod, disk, and donut baffles, helical baffles, segmental baffles, and orifice baffles etc. Baffles enhance the transfer of heat by directing the fluid flow through shell in a desired pattern which increases the turbulence and reduces the stagnant pockets. Baffles are often considered a central element in the shell and tube heat exchanger’s design. In this exchanger, baffles are crucial for directing flow and generating turbulence which results in heat exchanger operation more efficiently (Webbusterz Engineering, 2023).

Computational Fluid Dynamics (CFD) is visualization of gas and liquid flow and its effect on objects through the use of computer software, physics, and applied mathematics. The basis of computational fluid dynamics is the Navier-Stokes equations. The relationship between a moving fluid's temperature, pressure, density, and velocity is expressed by these equations. CFD proves valuable in examining various aspects such as fluid flow, temperature distribution, and heat transfer. To derive numerical solutions for pressure distributions and temperature gradients, With CFD, the system is divided into small cells, and governing equations are applied to each of these distinct components (Othman, 2009). The software is able to build a virtual prototype of a device or system before adding the real physics into the model. It also offers data and images that forecast how well this design will work.

This thesis focuses on analyzing the performance of a shell-and-tube heat exchanger by varying baffles number and changing the baffle cut size using computational Fluid Dynamics. The term "baffle cut" refers to the height of the segment that is cut out of each baffle to allow fluid from the shell to pass through it. It is usually expressed as a percentage of the inside shell diameter that the baffle does not cover. A standard recommended value of 20% to 25% cut of the shell diameter is typically applied in the design of shell and tube heat exchangers. (Raju, 2011) The heat exchanger's performance is significantly influenced by the baffle cut size. A larger cut tends to result in a lower pressure drop. The larger cut leads to a poorly distributed flow characterized by large eddies, creating dead spaces or stagnant areas behind the shell's baffles. Consequently, the heat transfer coefficient is also lowered. A smaller cut, however, raises the heat transfer coefficient and pressure drop. (Webbusterz Engineering, 2023) . The baffle cut orientation can differ from horizontal cut and vertical cut and varies from 15% to 45% of the shell inside diameter. This research is be based on studying the pressure, temperature and flow characteristics inside the shell using horizontal baffle with varying number of baffles and baffle cut 25%, 35% and 45%.

1.2. Problem Statement

Heat is transferred between different media using heat exchangers. This could lead to some heat loss to the environment as heat is transferred between different media, which would be ineffective heat transfer. In order to reduce the loss and enhance the properties of heat transfer, the heat exchangers should be designed properly. For improving the heat transfer phenomenon, the baffle number and baffle cut plays an important role. Optimizing the heat exchanger is crucial for achieving higher effectiveness and efficient heat transfer at the most cost-effective rates. Understanding the effects of baffle cut and number on pressure drop and heat transfer will help achieve this optimization. By carefully considering these factors, we can select a right balance that enhances performance while keeping costs to a minimum. Thus, this study helps in studying the impact of baffles number and baffle cut in a shell and tube using Computational Fluid Dynamics.

1.3. Objectives

1.3.1. Main Objective:

i. To analyze the shell-and-tube heat exchanger with different baffles number and different baffle cut using CFD

1.3.2. Specific Objective:

i. To design the shell-and-tube heat exchanger with different baffles number and different baffle cut.

ii. To simulate the shell-and-tube heat exchanger with CFD for analyzing the fluid flow patterns, temperature distributions, and pressure drops within the shell.

iii. To study the impact of baffles number and baffle cut on heat transfer and pressure drop.

CHAPTER TWO: LITERATURE REVIEW

Heat exchangers are widely employed devices in various process industries. Heat is transferred between two process streams using heat exchangers. Heat exchangers are required for processes involving cooling, heating, condensation, boiling, and evaporation. There is a temperature variation between the fluids along the length of a heat exchanger due to the variation in temperature of each fluid as it passes through the device. According to their application, various heat exchangers are named accordingly. For example, a heat exchanger used for condensing is called a condenser, and similarly a heat exchanger used for boiling purposes is called a boiler.

2.1 Types of Heat Exchanger

Numerous heat exchanger types have been developed and classified based on various factors, such as the nature of the heat exchange process, the relative direction of fluid movement, design and construction, and the physical state of the fluid. This extensive classification system allows for the accommodation of a wide range of applications (Rajput, 2012).

2.1.1. Nature of heat exchange process

Heat exchangers can be divided into two primary categories based on the type of heat exchange process they involve: direct (open) contact and indirect contact heat exchangers.

i. Direct contact heat exchanger

In direct heat exchangers, also known as open heat exchangers, the heat exchange takes place through the direct mixing of hot and cold fluids, facilitating the simultaneous transfer of heat and mass. These units are selectively employed in situations where the mixing of liquids is either harmless or desirable. Examples of direct contact heat exchangers include cooling towers, jet condensers, direct contact feed heaters.

ii. Indirect contact heat exchanger

In this, heat transfer between two fluids occurs by passing through the wall separating them. This type encompasses the following subtypes:

a. Regenerators

b. Recuperators or surface exchangers

In a regenerative heat exchanger, a hot liquid is directed through a specific medium known as a matrix. Heat is transferred to the solid matrix and stored there. The heat stored in the matrix is then transferred to the cold liquid by flowing over the heated matrix. Regenerators are often used in conjunction with engines and gas turbines.

In a recuperator, liquid flows simultaneously on both sides of the separating walls. Without mixing or making direct physical contact, heat transfer takes place between fluid streams. These heat exchangers are used when mixing two liquids is not possible or desirable. Examples: car radiators, oil coolers, intercoolers, economizers. Flow through direct heat exchangers and heat exchangers is considered steady state, whereas flow through regenerators is transient in nature.

2.1.2 Relative direction of fluid motion

On the basis of relative direction of two fluid streams, the heat exchangers are divided into three groups:

i. Parallel flow heat exchanger

Two liquid streams (hot and cold) flow parallelly in the same direction through the device in this type of heat exchanger. Both liquids enter at one side and exit at the other end. To prevent mixing, the two liquids can be separated by a solid wall or pipe, or they can be in direct contact with each other. Throughout the process, the temperature difference between the hot and cold liquids consistently decreases from the inlet to the outlet. This type of heat exchanger necessitates a substantial heat transfer area and is not frequently employed in practical applications.

ii. Counter-flow heat exchanger

A counter-flow heat exchanger has two fluids that flow in opposite directions. The hot and cold fluids enter the exchanger at different ends, creating a counter-flow configuration. This design maximizes the heat transfer rate over a specified surface area. Therefore, counter-flow heat exchangers find predominant use in applications involving the heating and cooling of liquids.

iii. Cross-flow heat exchanger

This heat exchanger is designed to allow the two fluids to flow perpendicular to one another. In this type of heat exchanger, the two fluids, hot and cold, intersect within a space, typically at right angles. Hot fluids flow through separate tubes and the fluid streams never mix. The cold fluids are thoroughly mixed as they flow through the heat exchanger. This is typically used where one side is liquid and the other gas, like a car radiator, where hot water flowing from side to side is cooled by air flowing up and down.

2.1.3 Design and constructional features:

Based on this, heat exchangers are categorized as follows:

i. Concentric tubes heat exchanger

This design makes use of two concentric tubes, each containing one of the fluids. The flow direction can be either parallel or opposite, and the utilization of vortex flow enhances the heat exchanger’s efficiency.

ii. Shell and tube heat exchanger

This kind of exchanger has a single fluid moving through a group of tubes that are surrounded in a shell. The other fluid flows over the tubes exterior surface and is directed through the shell. Such an arrangement is preferred when reliability and the effectiveness of heat transfer are paramount. The use of multiple tubes increases the surface area, significantly improving the heat transfer rate.

iii. Multiple shell and tube passes heat exchanger

This employs multiple passes for both the shell and the tubes, enhancing overall heat transfer efficiency. Multiple shell passes are achievable by rerouting the fluid through the shell, forcing it to pass through and back across the tubes with the aid of baffles. In heat exchangers with multiple tube passages, the liquid is directed in opposite directions through the tubes.

iv. Compact heat exchangers

These heat exchangers are designed for specific applications, offering a generous transfer area per unit volume. They find typical use in where there is a significant difference in the convective heat transfer coefficients between two liquids. Examples include flattened fin, plate-fin in tube exchange etc.

2.1.4 Physical state of fluids

Heat exchangers are classified according to the physical state of the liquid as follows:

i. Condenser

In condenser, the cold fluid's temperature rises gradually from the inlet to the outlet while the condensed fluid stays at a constant temperature throughout the heat exchanger. In this process, the hot fluid releases a portion of its latent heat, which is absorbed by the cold fluid.

ii. Evaporators

In this, the temperature of the hot fluid progressively drops from the inlet to the outlet, while the boiling fluid, or cold fluid, is continuously maintained at a constant temperature.

Shell and tube heat exchangers are without a doubt the most well-known and extensively utilized kind of heat exchanger among all of the other varieties. By guiding the fluid through the shell in a desired pattern, baffles improve heat transfer by increasing turbulence and decreasing stagnant pockets inside the heat exchanger (Webbusterz Engineering, 2023). There are several reasons to use baffles in shell and tube heat exchangers. Some reasons for using baffles are:

- To provide inner tubes structural support.
- To lessen the tubes mechanical vibration. Additionally, this lessens failures and leaks close to the tube sheet.
- To move the fluid inside shell over the tubes in the desired direction.
- To accelerate heat transfer. The convective heat transfer coefficient rises as a result of greater turbulence on the shell side and heat exchanger operates more efficiently.

In heat exchangers, various types of baffles are used.

2.2 Segmental Baffles

Segmented baffles include single segment, double segment, or triple segment baffles. It consists of partitions or baffles that divide the shell into multiple chambers and allow liquid to flow in a specific direction. Single-segment and double-segment baffles are the most commonly used because they most effectively direct flow onto the pipe. The types of segment baffles are explained below (Webbusterz Engineering, 2023).

2.2.1. Single Segmental Baffle: A single segment baffle separates the tank into compartments with a single continuous section. This baffle has rotating, vertical, and horizontal baffle cuts.

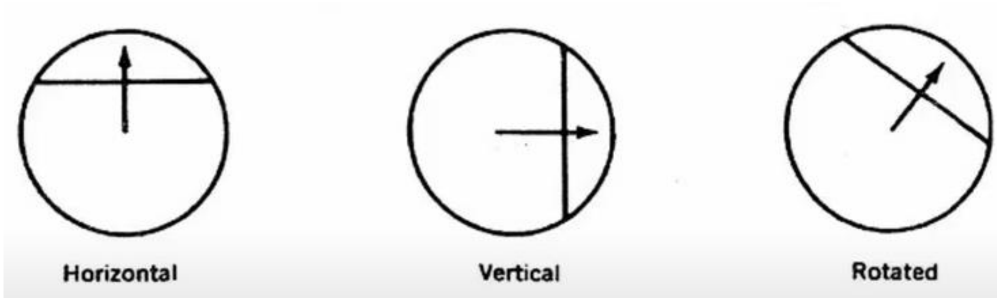

Figure 2. 1 Single Segmental Baffles

2.2.2. Double Segmental Baffle: The baffle plate is cut in this type of baffle so that two parts result: a split baffle and a center baffle. The split baffle and the center will slightly overlap. At least one row of tubes accommodates this overlap. With every additional baffle, these baffles divide the shell side flow into two flows. This results in a decrease in pressure drop and fluid velocity. These baffles are very useful for shell and tube heat exchangers with large tube bundles. Inside the shell, double segment baffles are positioned to guarantee more efficient flow distribution (Webbusterz Engineering, 2023).

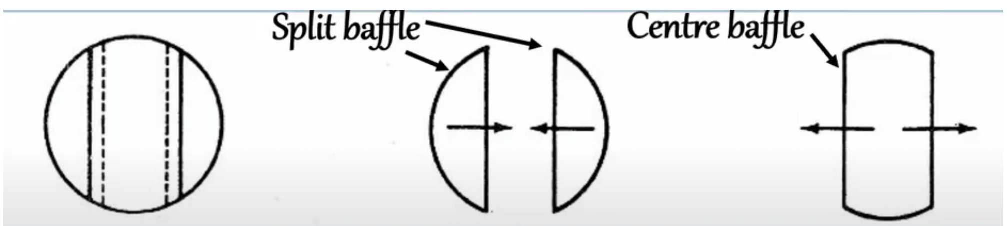

Figure 2. 2 Double Segmental Baffles

2.2.3. Triple Segmental Baffle: This kind of baffle divides the baffle plate into three sections. Two split baffles and one center baffle. Every baffle in this arrangement supports every tube. This minimizes tube vibration and pressure drop.

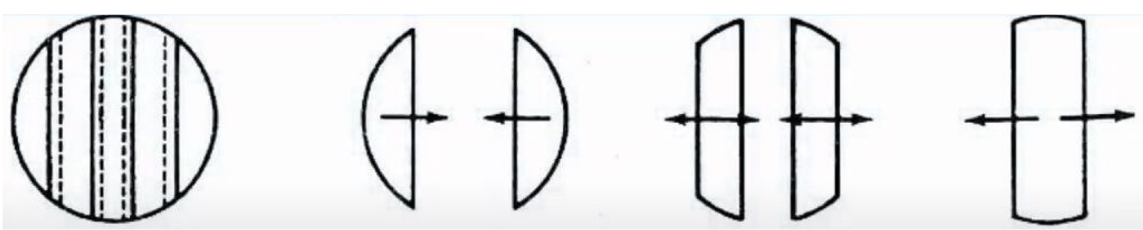

Figure 2. 3 Triple Segmental Baffles

2.3 Disc and Ring Baffle: Also known as disc and doughnut baffles. These baffles are installed inside the heat exchanger housing and are made up of several circular discs and annular rings. Disk and ring baffles are made up of inner disks that guide the flow radially throughout the pipe and alternating outer rings, also known as doughnut-shaped baffles. The disc should normally be bigger than the doughnut hole (EnggCyclopedia, 2023).

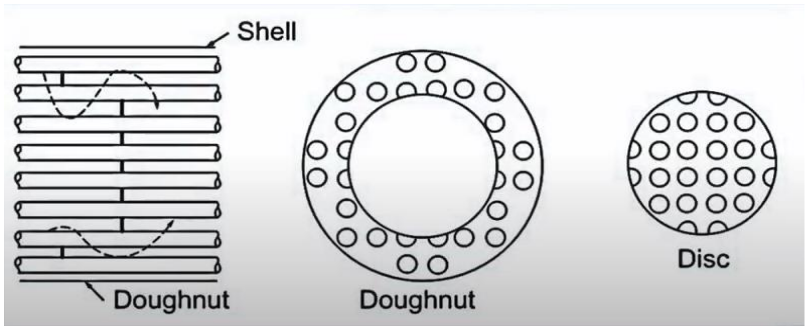

Figure 2. 4 Disc and Ring Baffles

2.4 Orifice Baffle: The shell side fluid in this kind of baffle travels through the gap created by the pipe's exterior and the baffle hole. Since these baffles are the least effective, they are not frequently utilized. If these baffles are obstructed by dirt and debris, they cannot be cleaned. Since corrosion and erosion can harm the pipe, this kind of baffle shouldn't be used with liquids that fall quickly onto the external surface (EnggCyclopedia, 2023).

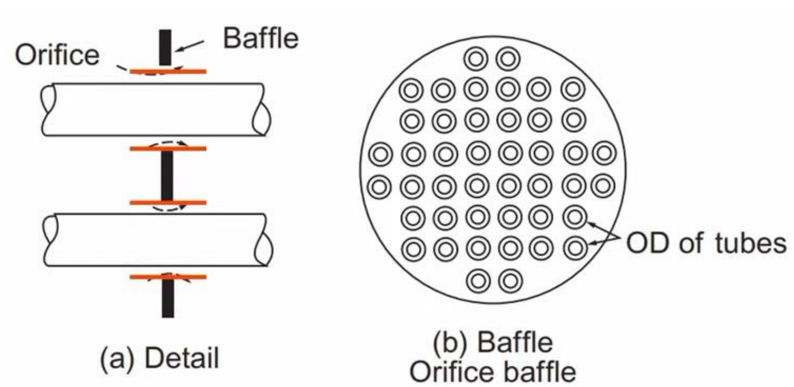

Figure 2. 5 Orifice Baffle

2.5 Baffle Cut

Baffle cut refers to the height of the segment that is cut out of each baffle to allow fluid from the shell to pass through it. It is usually expressed as a percentage of the inside shell diameter that the baffle does not cover. The amount of fluid that can pass through a shell and tube heat exchanger is determined by the baffle's size, or opening (Webbusterz Engineering, 2023).

Baffle cut orientation varies from 15% to 45% of the shell inside diameter and can be different from both horizontal and vertical cuts. Greater or lesser baffle cuts than the ideal baffle cut usually lead to large eddies, dead zones behind the baffles, and higher than anticipated pressure drops. A more constrained flow is produced by a smaller baffle cut, which increases turbulence, pressure drop, and heat transfer. In contrast, a larger baffle cut permits more fluid flow while lowering heat transfer, pressure drop, and turbulence. This has a negative impact on heat transfer, though, since it creates an uneven flow with big eddies that leads to dead zones or stagnant areas behind the shell's baffles, lowering the heat transfer coefficient. The ideal baffle cut is determined by heat exchanger design's specific needs, such as pressure drop tolerance, desired heat transfer rate, and fluid properties (Webbusterz Engineering, 2023).

2.6 Baffle Spacing

Baffle spacing refers to the center distance between consecutive baffles in a heat exchanger. The heat exchanger's baffle spacing controls the heat transfer rate, turbulence production, and fluid flow path. Increases in flow restriction, turbulence, and heat transfer coefficient are proportional to the baffle spacing. Additionally, when the distance reduces, the pressure drop rises, which may have a detrimental effect on the performance of heat exchanger. Larger baffle spacing, on the other hand, permits the fluid to flow more freely, lowering pressure drop and turbulence while also lowering the heat transfer coefficient. Sufficient distance between baffles is necessary to attain the highest possible heat transfer rate while minimizing pressure drop. Consequently, the deflection distance cannot be less than one-fifth of the shell inside diameter, or two inches, whichever is larger, nor can it be less than the inside diameter of the shell. Baffle spacing should be between 0.3 and 0.5 times the diameter of shell (EnggCyclopedia, 2023).

Different types of shell and tube heat exchangers can be easily configured by changing the arrangement of the bundle tubes. The heat transfer coefficient and pressure drop within the shell of a shell-and-tube heat exchanger can be studied using numerical modeling of a small heat exchanger considering factors such as baffle spacing, baffle cut, and shell diameter. The flow and temperature fields within the shell can be solved using a commercially available Computational Fluid Dynamics package. Multiple CFD simulations can be conducted for single shell and single tube heat exchangers or multi-tube heat exchangers with varying baffles number and baffle cuts. CFD techniques involve computer-aided analyses used to simulate heat exchangers with fluid flow and heat transfer. In this process, CFD breaks down the entire heat exchanger into individual elements, enabling the determination of temperature, pressure distributions, and velocity vectors (Ender Ozden, 2010).

CFD is a technique for modeling patterns of fluid flow and transfer of heat through the application of advanced algorithms and computers. In the past, the development of new products necessitated physical testing using prototypes. However, with CFD allows for virtual testing, rendering many physical experiments unnecessary. CFD modeling tools leverage mathematical models and numerical analysis techniques to predict and understand fluid behavior. The most prevalent CFD solutions are grounded in the Navier-Stokes equations, that consider factors such as flow velocity, pressure, and viscosity to explain how a fluid moves and interacts with its surroundings under specific boundary conditions (Sarkar, 2023). The governing equations in Computational Fluid Dynamics are rooted in the conservation laws of a fluid's physical properties. These laws encompass the conservation of mass, momentum, and energy, and they form the foundation for understanding and simulating the behavior of fluids in various scenarios.

Equation for Conservation of Mass is given as follows:

$$\frac{D\rho}{Dt} + \rho(\nabla.\vec{v}) = 0 \qquad \text{Equation 2.1}$$

where, $\vec{v}$ is the velocity, ∇ the gradient operator and ρ is the density

$$\vec{\nabla} = \vec{\imath}\frac{\partial}{\partial x} + \vec{\jmath}\frac{\partial}{\partial y} + \vec{k}\frac{\partial}{\partial z}$$

Assuming a constant density and incompressible flow, the continuity equation becomes:

$$\frac{D\rho}{Dt} = 0 \rightarrow \overline{\nabla.\vec{v}} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{dz} = 0 \qquad \text{Equation 2.2}$$

The Navier-Stokes Equation, also known as the Conservation of Momentum, is given by:

$$\frac{\partial}{\partial t}(\rho\vec{v}) + \nabla.(\rho\vec{v}\vec{v}) = -\nabla p + \nabla.\bar{\bar{\tau}} + \rho\vec{g} \qquad \text{Equation 2.3}$$

where $\bar{\bar{\tau}}$ is viscous stress tensor, $\wp$ is static pressure, and $\rho\vec{g}$ is the gravitational force per unit volume.

As per Stoke's Hypothesis, the viscous stress tensor $\bar{\bar{\tau}}$ can be can be expressed as follows:

$$\tau_{ij} = \mu \frac{\partial v_i}{dx_j} + \frac{\partial v_j}{\partial x_i} - \frac{2}{3}(\nabla.\vec{v})\delta_{ij}$$

The Navier-Stokes equation can be simplified to below if it is assumed that the fluid is incompressible and that its viscosity coefficient (μ) remains constant.

$$\rho \frac{D\vec{v}}{Dt} = -\nabla\wp + \mu\nabla^2\vec{v} + \rho\vec{g} \qquad \text{Equation 2.4}$$

The first law of thermodynamics, conservation of energy, states that an increase in heat and work added to a system results in an increase in energy within the system

$$dE_t = dQ + dW \qquad \text{Equation 2.5}$$

where dE_t is the increment in the total energy of the system, dQ is the heat added to the system and dW is the work done on the system.

Heat transfer across a surface is given by the following basic equation:

$$Q = U \times A \times \Delta T_m \qquad \text{Equation 2.6}$$

where

Q = Heat transferred

U = Overall heat transfer coefficient

A = Area

ΔT_m = Log mean temperature difference

The log mean temperature difference ΔT_m is

$$\Delta T_m = \frac{(T_1 - t_2) - (T_2 - t_1)}{ln\frac{(T_1 - t_{2)}}{(T_2 - t_{1)}}}$$

where

T_1 = temperature of fluid at shell side inlet

T_2 = temperature of fluid at shell side outlet

t_1 = temperature of fluid at tube side inlet

t_2 = temperature of fluid at tube side outlet

Meshing plays a crucial role in computational fluid dynamics (CFD) by facilitating the simplification of fluid flow analysis. The targeted region undergoes subdivision into small cells, forming a network that serves as the basis for analysis. This method enables the concentration of efforts on solving fluid flow equations for individual cells rather than the entire object. By doing so, simulations become more manageable for computers. The quality and resolution of the mesh significantly impact the accuracy of the results. Finer meshes yield more detailed outcomes but demand higher computational power and resources to execute. Striking a balance between mesh refinement and computational efficiency is essential for obtaining reliable and practical insights from CFD simulations (Sarkar, 2023).

Various researches have been conducted to reduce losses and enhance heat transfer while keeping costs to a minimum. Ender Ozden et al. (2010) investigated how shell diameter, baffle spacing, baffle cut, and pressure drop are related to the coefficient of heat transfer. They conducted the simulation for a heat exchanger with a single pass single shell and tube. They discovered that more cross flow and heat transfer area were attained with a reduction in baffle spacing. The effectiveness of heat transfer in the region behind the baffles diminished when there was greater spacing between them. This was due to the change in flow direction and the development of a recirculation zone after the fluid strike the baffle. Conversely, with less space between the baffles, the fluid strike the back face of the previous baffles again, leading to an increased in the effective heat transfer area. Consequently, it was observed that greater heat transfer occurred with a higher number of baffles.

Avinash D. Jadhav et al. (2014) used numerical analysis to examine how the pressure drop and heat transfer coefficient are affected by the baffle cut, baffle spacing, and shell diameter. They contrasted their findings with those obtained using the Bell-Delaware approach. They conducted series of simulations with a different number of baffles and turbulent flow for two baffle cut values. By adjusting the flow rate, they looked at effect of the ratio of shell diameter to baffle spacing on performance of heat exchanger. It was found that the results depend on the choice of turbulence model, and the optimal model among those taken into consideration was chosen by contrasting the Bell-Delaware method results with the CFD results.

Arjun Sunil et al. (2014) conducted research to predict the performance of a shell and tube heat exchanger. The assessment of the heat exchanger's performance was carried out utilizing the CFD package FLUENT and was compared against existing experimental values. The study also aimed to calculate the heat exchanger's performance by incorporating helix baffles as an alternative to regular Segmental Baffles, and the results were subjected to comparison. The outcomes of the numerical experimentation indicated an enhancement in heat exchanger's performance when helical baffles were used in comparison to segmental baffles.

Noor Afsar et al. (2018) utilized Computational Fluid Dynamics to analyze a simple shell and tube heat exchanger. They conducted series of simulations with varying mass flow rates by manipulating the baffle orientation and baffle cut values. The values of the heat transfer coefficient, shell side pressure drop, and shell outlet temperature were computed using the CFD results. The influence of mesh density on outlet characteristics was investigated through an analysis of mesh dependency. Through a comparison of results between horizontal and vertical baffle orientations, the influence of baffle orientation was observed. Notably, heat transfer coefficients were higher in the horizontal orientation than in the vertical orientation, and the horizontal baffle showed a larger pressure drop than the vertical baffles for lower baffle cut values.

Chetan Namdeo Patil et al. (2014) conducted calculations to assess the impact of baffle cut on the heat transfer coefficient and pressure drop while maintaining constant baffle spacing. Their findings revealed that the pressure drop was lower for the 30% baffle cut, and the heat transfer coefficient was nearly identical for the 30% and 25% baffle cuts.

Kiran K et al. (2014) examined the influence of baffle spacing in a shell and tube heat exchanger on heat transfer, pressure drop, and shell side outlet temperature. They hypothesized that baffle spacing has little effect on exit temperature, however mass flow rate has a big impact. Additionally, they observed that the pressure loss exhibits significant variations with changes in baffle spacing and mass flow rat.

Prasanna J et al. (2013) examined the hydrodynamic and heat transfer effects of altering the baffle cut and spacing on shell and tube heat exchangers. They discovered that a 25% baffle cut yields marginally better performance. Heat transfer increases when baffle spacing decreases.

CHAPTER THREE: RESEARCH METHODOLOGY

This research is aimed to predict the performance of a shell and tube heat exchanger with varying numbers of baffles and baffle cuts using Computational Fluid Dynamics. In this study, three different baffle cuts are implemented in the shell, creating distinct flow paths across the tube bundle. The geometry modeling is executed using ANSYS software, and the models are compared by altering the number of baffles and baffle cut configurations within the shell. Subsequently, each geometric model undergoes CFD analysis, and results are obtained through meshing the geometry with appropriate parameters. This research involves three steps: geometry development, mesh generation and finally simulation and visualization.

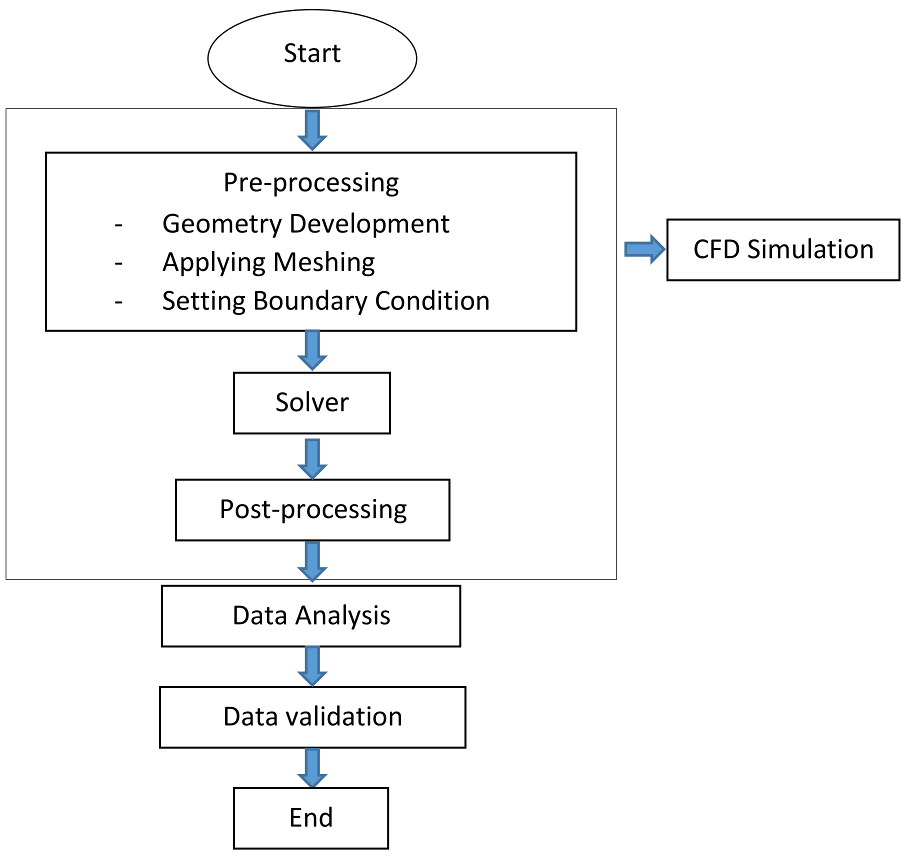

Figure 3. 1 Flow Chart

3.1 Pre-processing

The preprocessing stage encompasses tasks such as creating the geometry, preparing the geometry for Computational Fluid Dynamics (CFD) simulation, and dividing the domain into small volumes or cells through a process known as meshing or grid generation.

3.1.1 Geometry Development

This entails designing the geometry with three different numbers of segmental baffles and baffle cuts. The dimensions of various parts of the heat exchanger are detailed in the table below:

Design Parameter

Table 3. 1 Design Parameters

Shell diameter($D_{s)}$	100 mm
Tube diameter(d)	20 mm
Shell inlet & outlet tube diameter($d_{s)}$	20 mm
Heat exchanger length(L)	450 mm
No. of tubes($N_{t)}$	7
Tube bundle geometry and pitch	Triangular, 25 mm
Shell side inlet temperature(T)	363K
Tube side inlet temperature(t)	300K
Baffle cut($B_{c)}$	25%, 35%, 45%
No. of baffle($N_{b)}$	8, 6, 4
Baffle spacing	Equispaced

Table 3. 2 Properties of working fluid

Working fluid	Water
Specific Heat	4182 J/Kg-k
Density	998.2 Kg/m^3
Viscosity	0.001003 Kg/m-s
Thermal Conductivity	0.6 W/m-k

Table 3. 3 Properties of material

Material	Aluminum
Specific Heat	871 J/Kg-k
Density	2719 Kg/m^3
Thermal Conductivity	202.4 W/m-k

The shell and tube heat exchanger with a baffle cut 25%, featuring different numbers of baffles, is illustrated below. Specifically, the configurations with 8, 6, and 4 baffles are presented in Fig 3.2, 3.2, and 3.3, respectively. The baffles are uniformly spaced throughout the length of the heat exchanger, with cold water flowing through the tubes and hot water circulating inside the shell over the tubes.

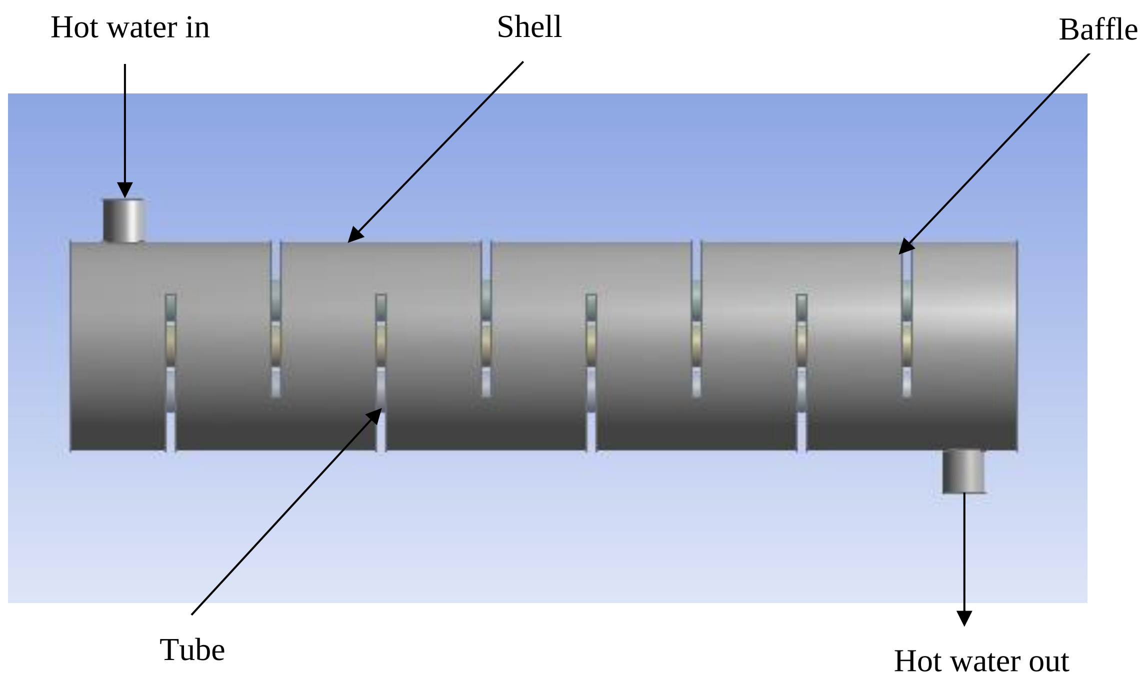

Figure 3. 2 Shell and tube with 8 number of baffles

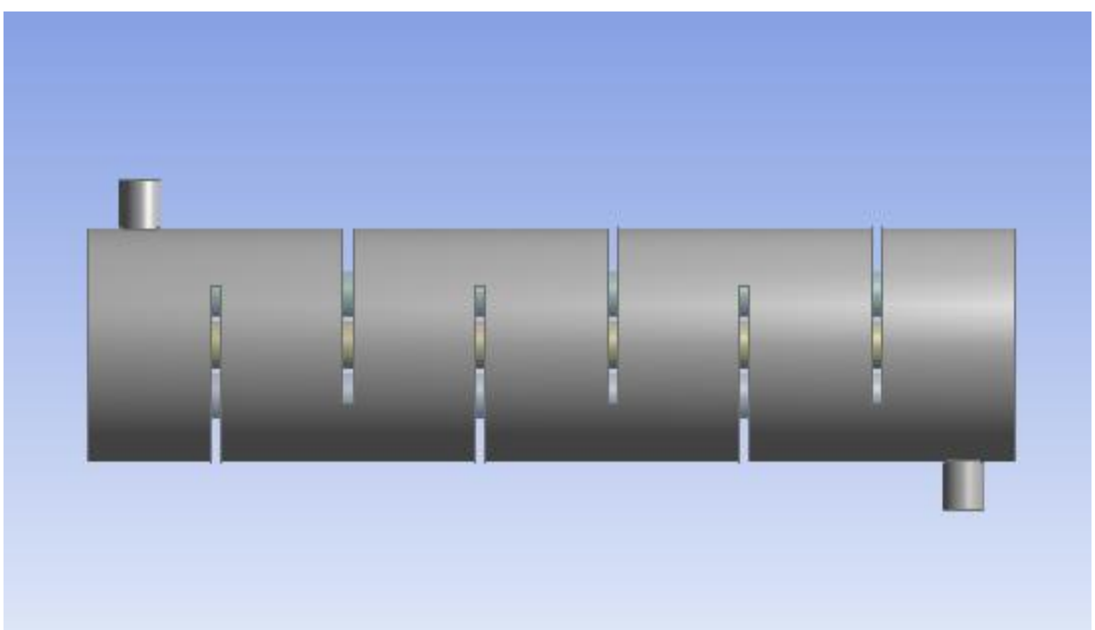

Figure 3. 3 Shell and tube with 6 number of baffles

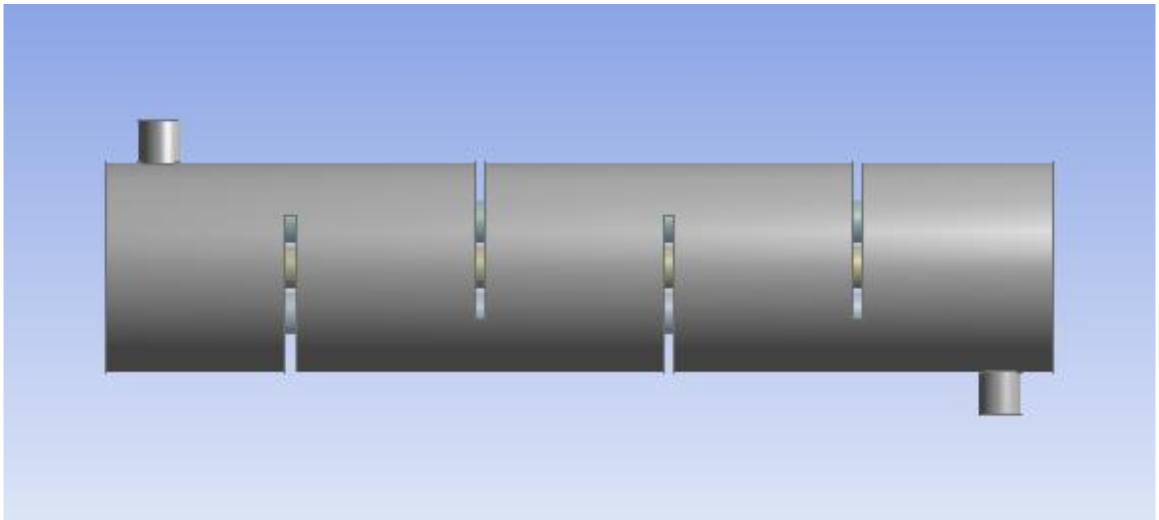

Figure 3. 4 Shell and tube with 4 number of baffles

3.1.2 Mesh Generation

Meshing of the geometry is accomplished using the ANSYS meshing tool. The shell volume and surfaces of the model are meshed utilizing tetrahedral elements. Various mesh sizes are employed, varying with the baffles number and baffle cut configurations. For 8 number of baffles and 25 % baffle cut, the number of elements is 885468. For 35% baffle cut the element is 887338 and for 45% baffle cut, the element is 890960 respectively.

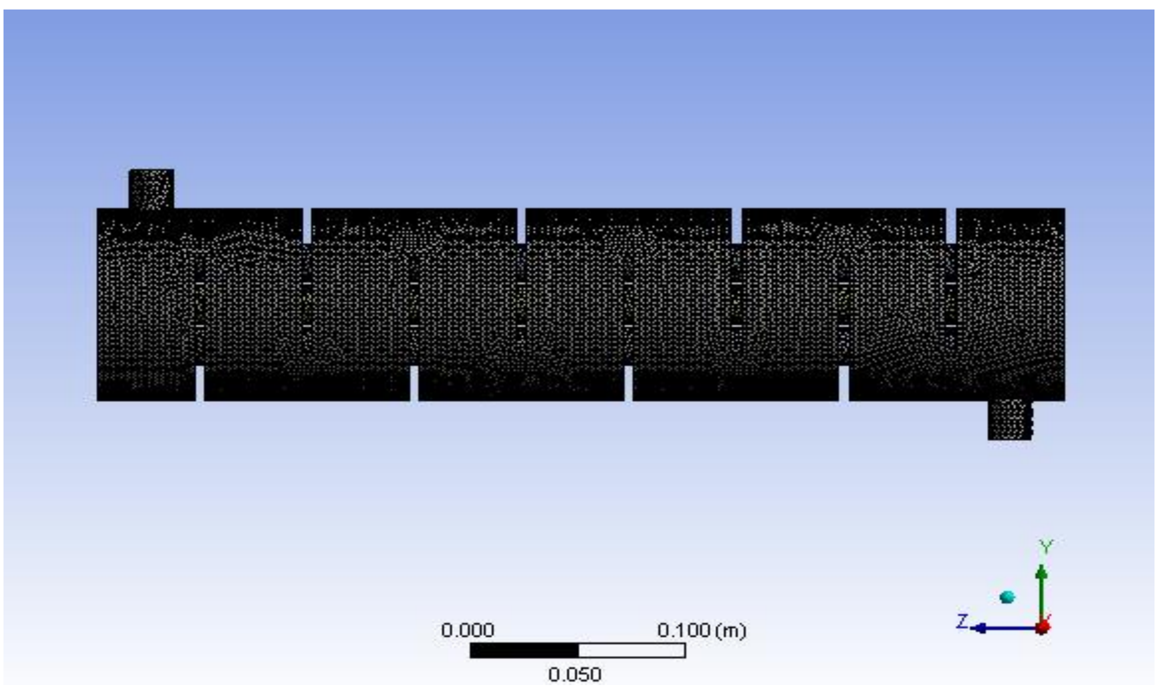

Figure 3. 5 Overall Meshing

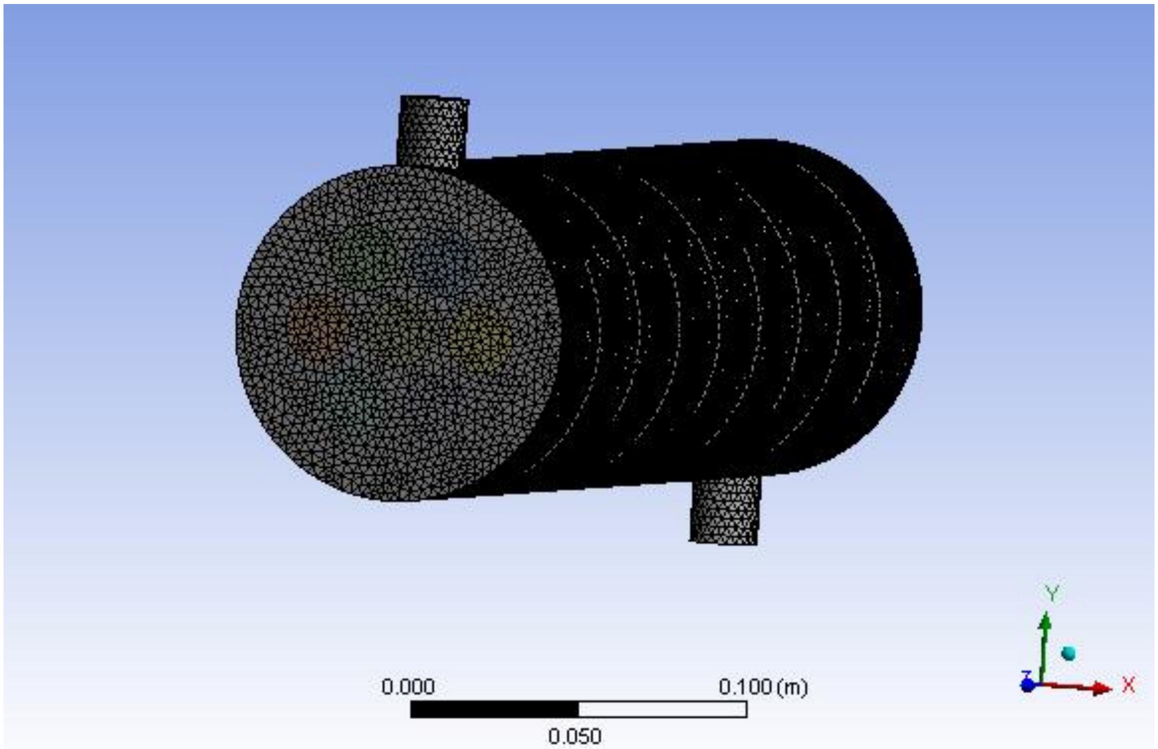

Figure 3. 6 Magnified view of meshing

3.1.3 Boundary Conditions

For various zones, different boundary conditions are used. The outlets and inlets are pressure outlets and velocity inlets respectively. Both the hot and cold fluids input velocities 0.4 m/s and 1.2 m/s respectively are maintained constant. The default outlet pressure, or atmospheric pressure, is maintained. The inlet temperature of the hot fluid is 363 K, while the cold fluid is maintained at 300 K. In accordance, the other wall conditions are defined. The ambient air temperature is maintained at 300k.

3.2 Solving or Processing

In this step, the discretized equations are being iteratively solved by the CFD simulation software using the CFD solver. It may take a while or a lot of computer power to complete this step. The simulation is set for 60 iterations with the boundary condition mentioned above.

3.3 Post-processing

After the completion of the solution, the simulation's outcomes are qualitatively and quantitatively analyzed and visualized using plots, reports, monitors, 2D/3D images, and animations. This involves the analysis and visualization of flow, temperature and pressure inside the shell with different segmental baffle cut. For varying numbers of baffles and baffle cuts, the cross-sectional velocity, streamline, temperature, and pressure contours are are studied with the help of simulation in ANSYS. A plane at the cross section is used to create the contours, and the features are represented in a color map based on their magnitude. Following assumptions are considered during the analysis:

a) The flow is incompressible and the fluid’s density remains constant.
b) No consideration given to leakage from the area between the baffle and tube.
c) The study does not account for heat that is transferred to the baffles.
d) The working fluid is water and considered to have constant properties.
e) The impact of the header is not considered.

CHAPTER FOUR: RESULTS AND DISCUSSION

4.1 Design of shell and tube heat exchanger

In this study, the design of a shell and tube heat exchanger is conducted using ANSYS software. The geometry modeling of the heat exchanger involves employing ANSYS, incorporating three variations in the number of baffles (8, 6, and 4) and three different baffle cuts (25%, 35%, and 45%). The geometric models are compared by altering the number of baffles and the baffle cut within the shell. A consistent temperature is maintained for the flow through the tubes, with water serving as the working fluid. The model used is a shell with a diameter (D) of 100 mm, circular cross-section inlet and outlet diameters (ds) of 20 mm, and an estimated shell length of 450 mm. 7 tubes, each with a diameter (d) of 16 mm, are present. To assess the impact of baffle cut on flow, pressure drop, and temperature within the shell, three different numbers of segmental baffles (8, 6, and 4) are utilized, each with baffle cuts of 25%, 35%, and 45%. Other parameters and boundary conditions are kept constant, assuming constant fluid properties throughout the analysis. After geometric models' creation, each undergoes Computational Fluid Dynamics analysis. Additionally, to derive results from the CFD analysis, the geometry is meshed.

4. 2 Simulation of shell and tube heat exchanger

The analysis involves three variations of shell and tube heat exchangers, each featuring distinct numbers of baffles and diverse baffle cuts. In this study, hot water circulates through the shell side, while cold water traverses the tube side of the heat exchanger. Utilizing ANSYS simulation, velocity, streamline, temperature, and pressure contours across different cross-sections, varying the number of baffles and baffle cuts are examined. Contours are generated by selecting a plane at the cross-section, with characteristics depicted in a color map based on their respective magnitudes.

4.2.1 Simulation with 8 number of baffles

This involves simulating a shell-and-tube configuration with eight baffles and three distinct baffle cut percentages (25%, 35%, and 45%). Using the simulation tool ANSYS, the velocity, streamline, temperature, and pressure contours at cross-sections are thoroughly examined. These characteristics are visually represented, depicting their magnitudes through a color map.

4.2.1.1 Velocity Contour

The velocity profile is analyzed to comprehend to comprehend the distribution of flow across the cross-section at various positions within the heat exchanger. Velocity plots across the cross-sections, corresponding to different baffle cuts (25%, 35%, and 45%) with 8 baffles, are presented in Fig. 4.1, 4.3, and 4.5, respectively. Distinct flow patterns emerge with varying baffle cuts. The velocity contour associated with a 25% baffle cut reveals the creation of a recirculation zone at the backside of the baffle as the hot fluid passes through the baffles and into the shell. In this region, the flow is notably reduced, as depicted in Fig. 4.1. Consequently, the available area for shell-side fluid flow is constrained, fostering more efficient heat transfer in this specific configuration.

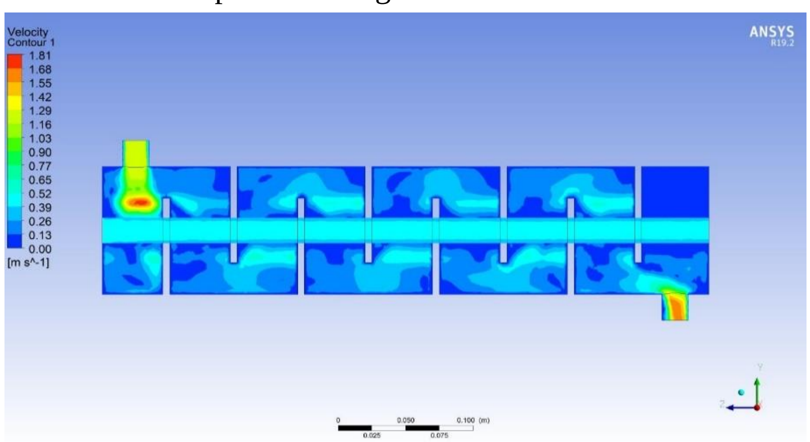

Figure 4. 1 Velocity contour at 25% baffle cut

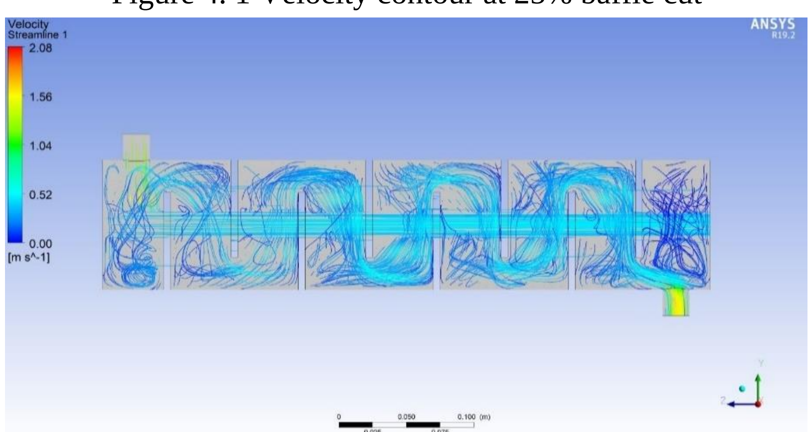

Figure 4. 2 Streamline at 25% baffle cut

The velocity contour for the 35% baffle cut is depicted in Figure 4.3. In this configuration, the flow area is expanded compared to 25% baffle cut on the shell and tube. However, the expansion results in dead zones forming between the two adjacent baffles, contributing to lower heat transfer efficiency.

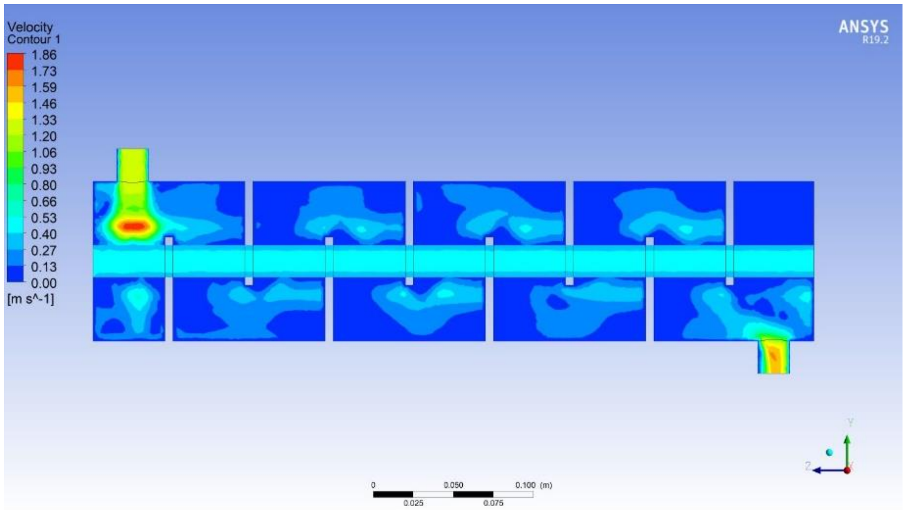

Figure 4. 3 Velocity contour at 35% baffle cut

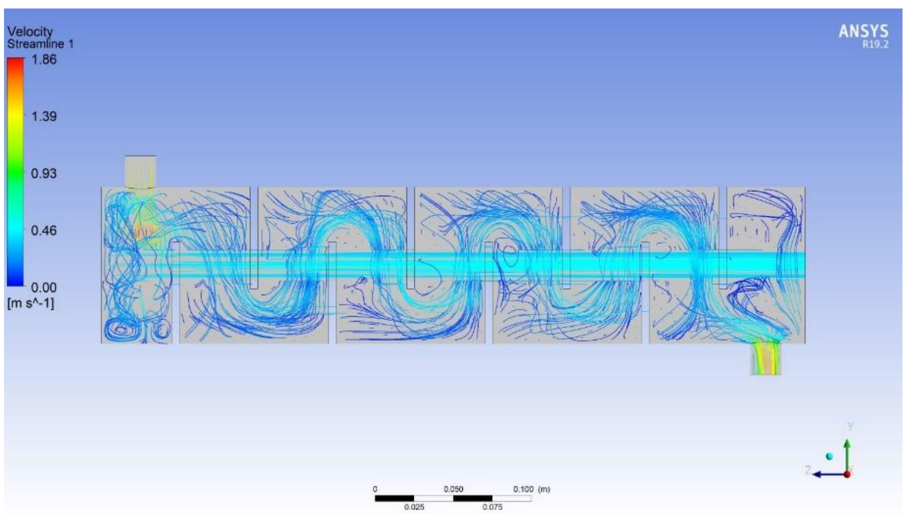

Figure 4. 4 Streamline at 35% baffle cut

The velocity contour for the 45% baffle cut is illustrated in Figure 4.5. In this, the flow area is greater compared to the baffle cuts of 25% and 35%. However, it is observed that there is minimal cross-flow, resulting in comparatively lower heat transfer efficiency compared to the previously discussed cases.

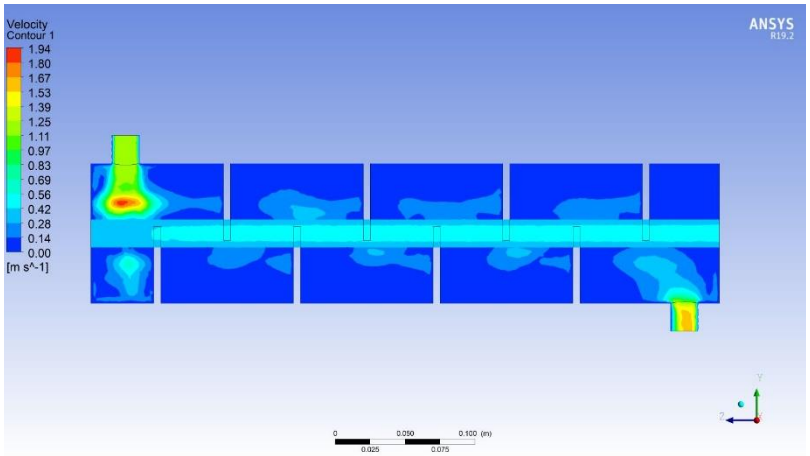

Figure 4. 5 Velocity contour at 45% baffle cut

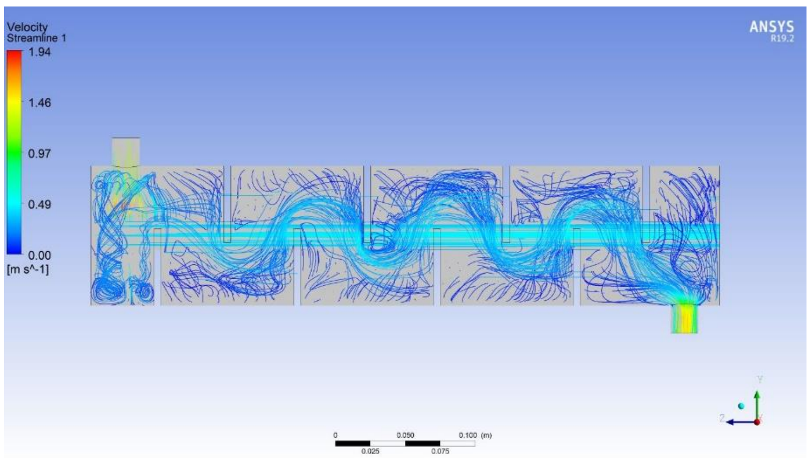

Figure 4. 6 Streamline at 45% baffle cut

4.2.1.2 Temperature Contour

The temperature contour plots across the cross-section, considering eight baffles and three different baffle cuts, are presented below. The hot water entering the shell and tube heat exchanger is initially set to 363K and variations are seen along the heat exchanger's length. For a 25% baffle cut, the temperature contour plots are shown in Fig. 4.7. As the hot water traverses the shell and baffles, a recirculation zone forms at the back side of the baffle, leading to reduced flow and subsequently lower temperatures in this region. Consequently, a more uniform temperature distribution is achieved with a 25% baffle cut. The outlet temperature on the shell side is measured at 354.34K, indicating higher efficiency of heat transfer in comparison to shell and tube heat exchangers with baffle cuts 35% and 45%.

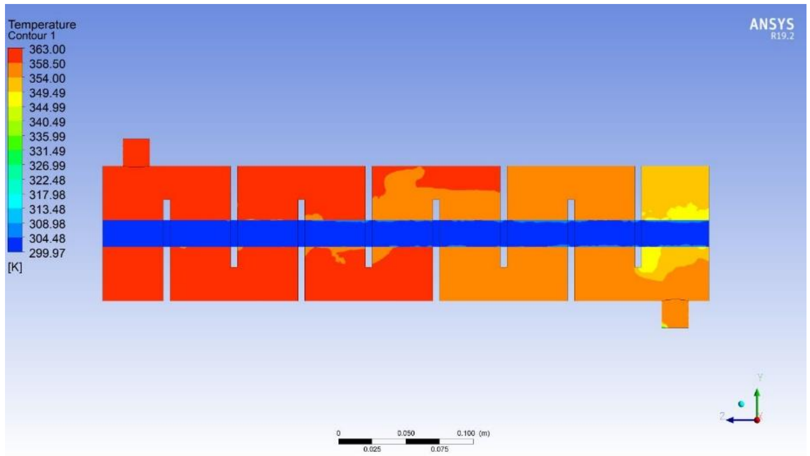

Figure 4. 7 Temperature contour at 25% baffle cut

The temperature contour plots across the cross-section for a baffle cut of 35% are depicted in Fig. 4.8. The presence of a recirculation zone at the back side of the baffle, accompanied by reduced flow, results in heightened heat transfer occurring predominantly at the edges. Consequently, the temperature of the hot fluid experiences more pronounced reductions at the edges in this configuration.

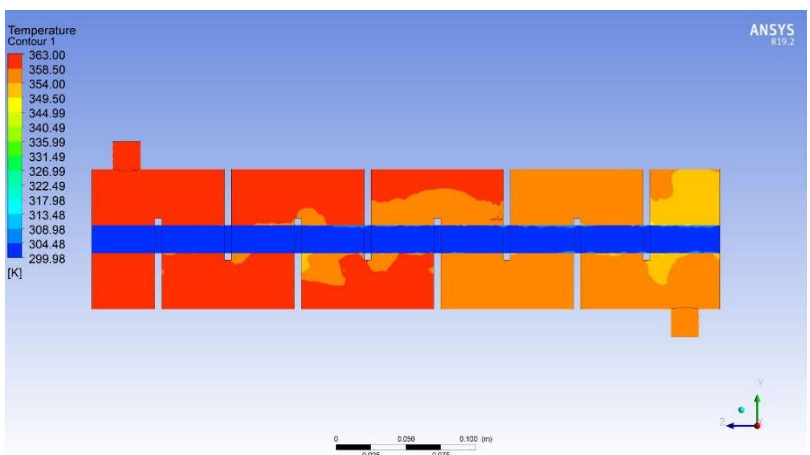

Figure 4. 8 Temperature contour at 35% baffle cut

The temperature contour plots across the cross-section for a baffle cut of 45% are presented in Fig. 4.9. In this configuration, the expanded area for the flow of shell-side fluid, in comparison to baffle cuts of 25% and 35%, results in reduced cross-flow. Consequently, the temperature in this scenario experiences a lesser decrease compared to the aforementioned cases, leading to lower heat transfer efficiency. The outlet temperature on the shell side is measured at 356.23K.

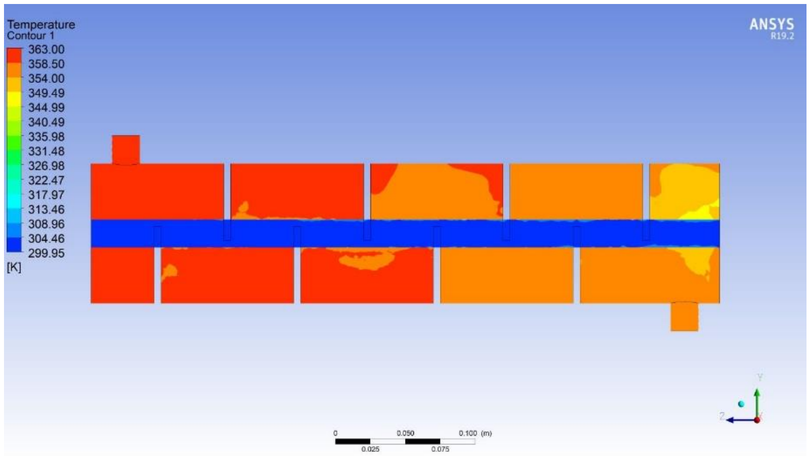

Figure 4. 9 Temperature contour at 45% baffle cut

4.2.1.3 Pressure Contour

The pressure contour for a baffle cut of 25% is illustrated in Figure 4.10. With a 25% baffle cut, the area available for fluid flow is restricted, leading to a higher pressure drop inside the shell.

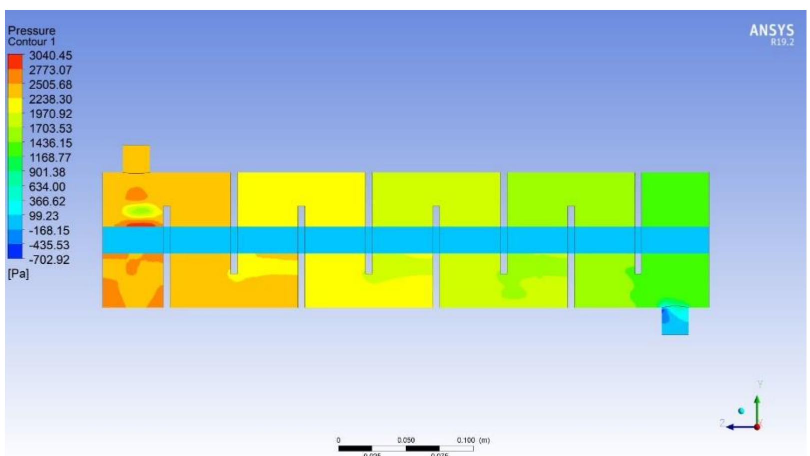

Figure 4. 10 Pressure contour at 25% baffle cut

The pressure contours for a baffle cut of 35% are depicted in Figure 4.11. It is found that an increase in baffle cut corresponds to a decrease in pressure drop inside the shell. Consequently, pressure drop is lower with a 35% baffle cut than it is with a 25% baffle cut.

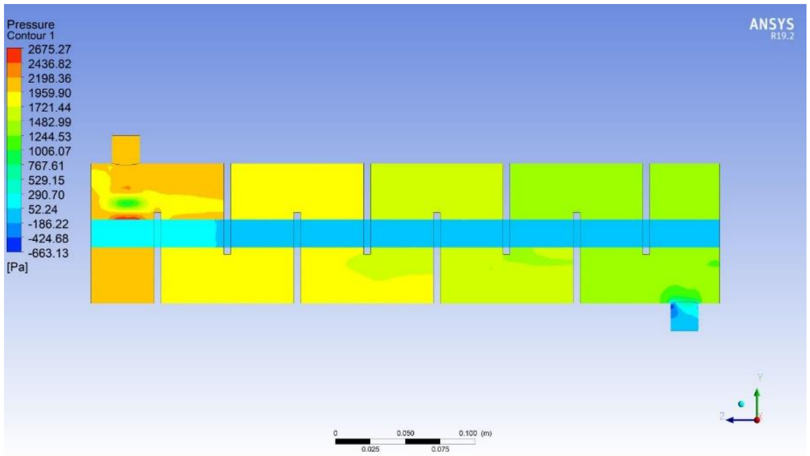

Figure 4. 11 Pressure contour at 35% baffle cut

The pressure contours for a baffle cut of 45% is shown in Figure 4.12. With a 45% baffle cut, the increased area for fluid passage results in a reduced pressure drop inside the shell compared to both the 25% and 35% baffle cut configurations.

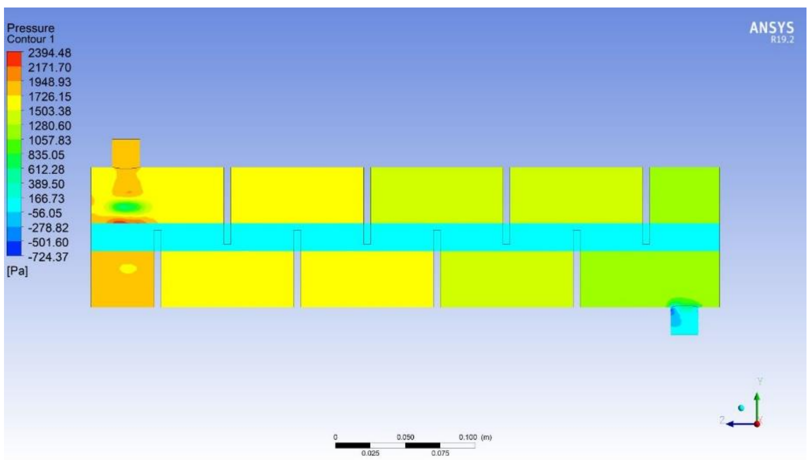

Figure 4. 12 Pressure contour at 45% baffle cut

4.2.2 Simulation with 6 number of baffles

This encompasses the simulating the shell-and-tube configuration with 6 baffles and three distinct baffle cuts (25%, 35%, and 45%). The resulting velocity, streamline, temperature, and pressure contours at cross-sections are depicted in the figures below.

4.2.2.1 Velocity Contour

The velocity plots across the cross-section at various baffle cuts (25%, 35%, and 45%) with 6 baffles are illustrated in Fig. 4.13, 4.15, and 4.17, respectively. Different flow patterns emerge corresponding to the distinct baffle cuts. With decrease in the number of baffles, the spacing between them increases, resulting in a wider recirculation zone. Fig. 4.13 depicts the velocity contour for a 25% baffle cut. In comparison to the shell and tube arrangement with 8 baffles, the larger recirculation zone in this configuration results in less interaction between the tubes and the hot fluid.

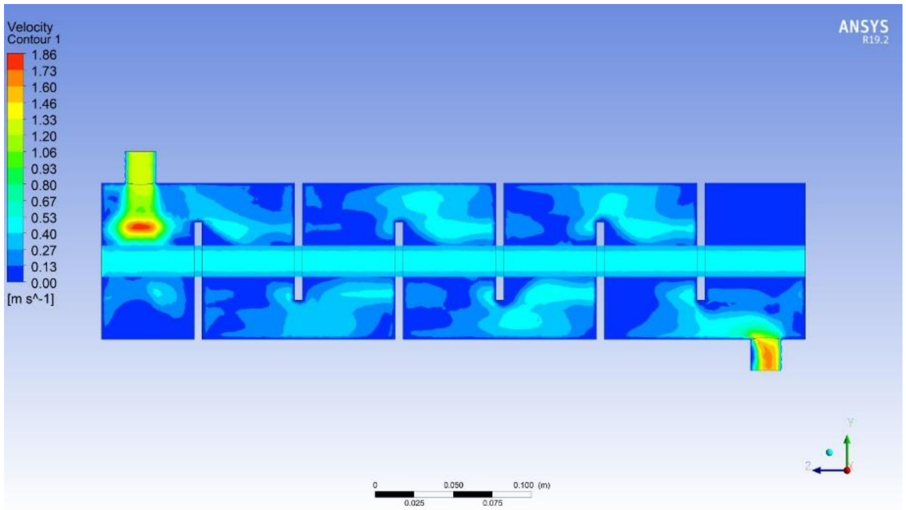

Figure 4. 13 Velocity contour at 25% baffle cut

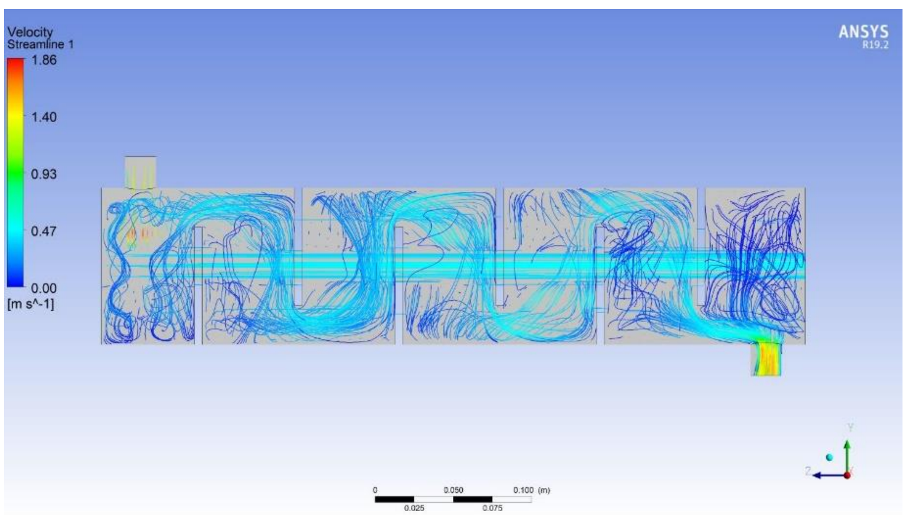

Figure 4. 14 Streamline at 25% baffle cut

The velocity contour for a baffle cut of 35% is presented in Figure 4.15. In this scenario, the flow area is expanded compared to the 25% baffle cut. However, there is a notable formation of dead zones in specific areas where the velocity becomes zero. This occurrence diminishes heat transfer efficiency in these dead zones between the two adjacent baffles.

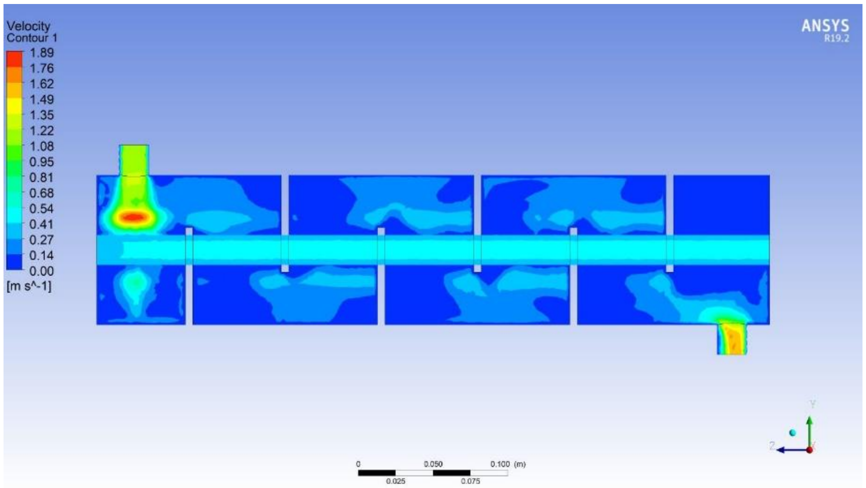

Figure 4. 15 Velocity contour at 35% baffle cut

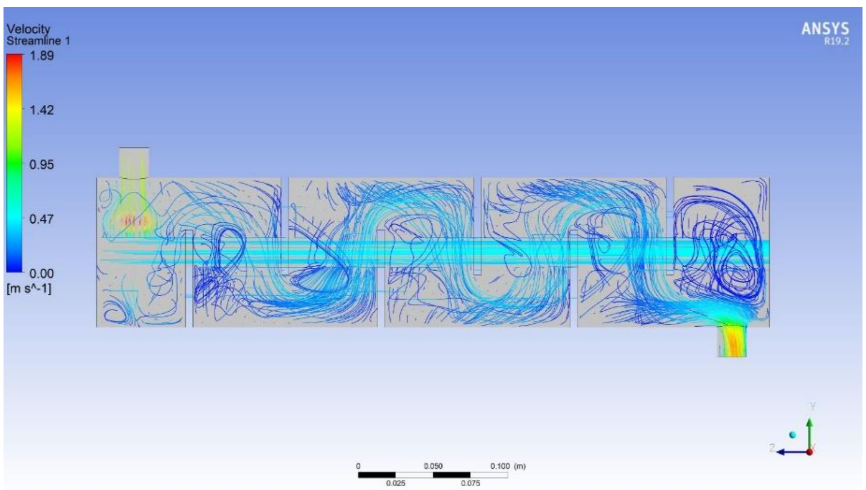

Figure 4. 16 Streamline at 35% baffle cut

The velocity contour for a baffle cut of 45% is depicted in Figure 4.17. In this configuration, the flow area is greater compared to the baffle cuts of 25% and 35%. However, minimal cross-flow is observed, resulting in relatively lower heat transfer compared to the preceding two cases.

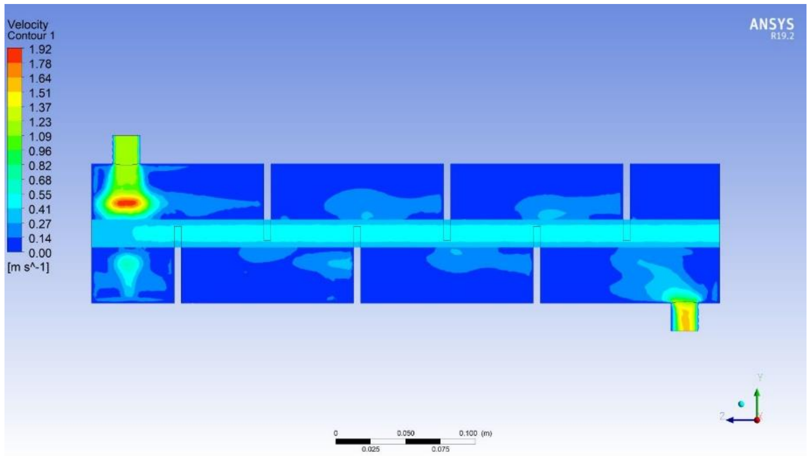

Figure 4. 17 Velocity contour at 45% baffle cut

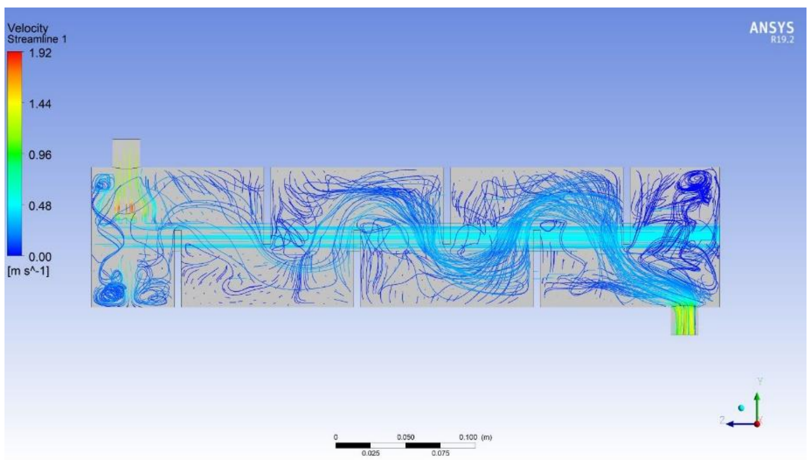

Figure 4. 18 Streamline at 45% baffle cut

4.2.2.2 Temperature Contour

The temperature contour plots across the cross-section, considering 6 baffles and three different baffle cuts, are presented below. As previously discussed, the wider recirculation zone in this case results in reduced interaction between the tubes and the hot fluid when compared to a shell-and-tube configuration with 8 baffles. This diminished contact leads to lower heat transfer efficiency, resulting in a slower rate of temperature decrease for the hot fluid than in the previous case.

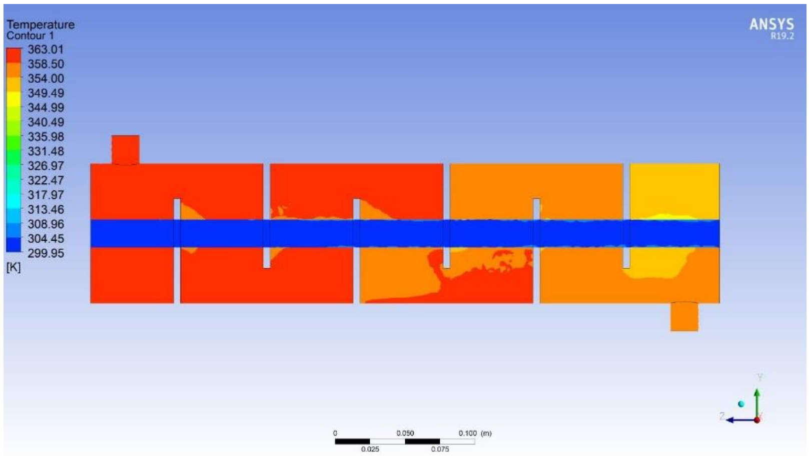

Figure 4. 19 Temperature contour at 25% baffle cut

The temperature contour plots across the cross-section for a baffle cut of 35% are depicted in Fig. 4.20. In this configuration, the flow area is expanded compared to the 25% baffle cut. However, reduced heat transfer results in a slower decrease in the temperature of the hot fluid in comparison to the baffle cut of 25%.

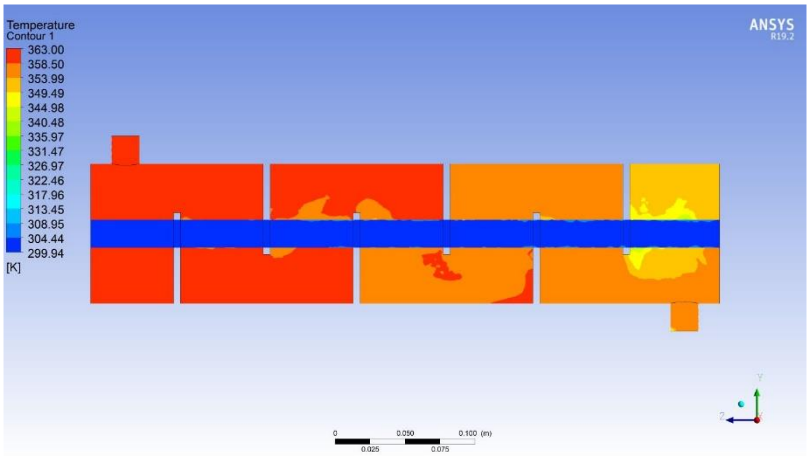

Figure 4. 20 Temperature contour at 35% baffle cut

The temperature contour plots across the cross-section for a baffle cut of 45% are presented in Fig. 4.21. In this, the increased area for shell fluid flow is accompanied by less cross-flow. Consequently, the temperature in this case experiences a lesser decrease compared to the above two cases, resulting in lower heat transfer efficiency. Additionally, the temperature reduction at the edges of the baffle is more pronounced than at the faces.

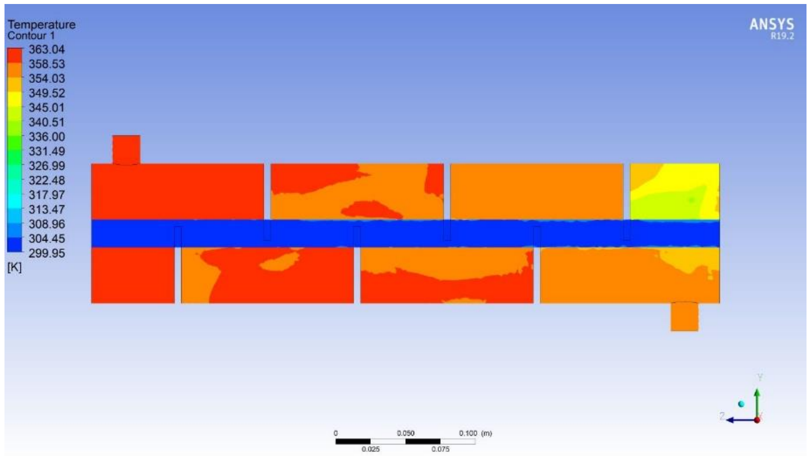

Figure 4. 21 Temperature contour at 45% baffle cut

4.2.2.3 Pressure Contour

The pressure contours for a baffle cut of 25% are illustrated in Figure 4.22. Notably, with a 25% baffle cut, the area available for fluid flow is restricted, resulting in a higher pressure drop inside the shell.

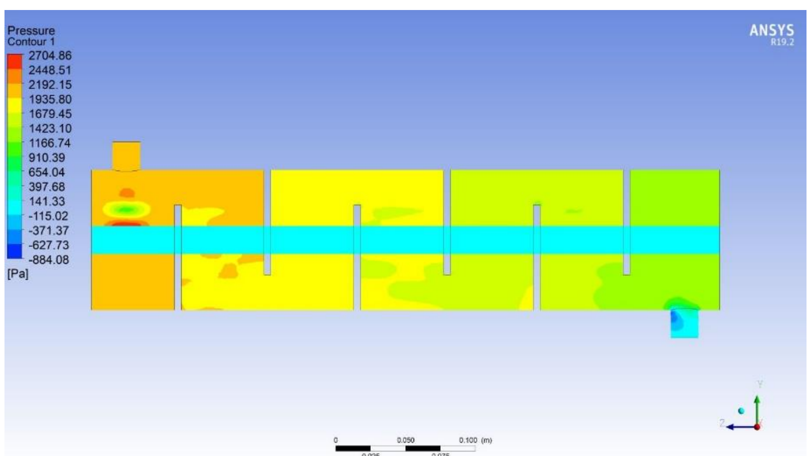

Figure 4. 22 Pressure contour at 25% baffle cut

The pressure contours for a baffle cut of 35% are presented in Figure 4.23. It is evident that an increase in baffle cut results in a larger area available for flow compared to the configuration with a 25% baffle cut. Consequently, the pressure drop inside the shell is decreased, resulting in an overall lower pressure drop compared to the case with a 25% baffle cut.

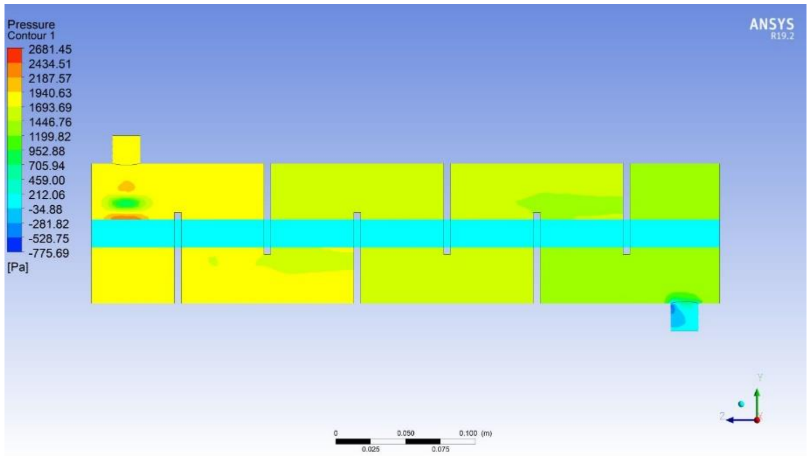

Figure 4. 23 Pressure contour at 35% baffle cut

The pressure contours for a baffle cut of 45% are displayed in Figure 4.24. With a 45% baffle cut, the area available for the fluid to pass through is larger compared to both the 25% and 35% baffle cut cases. Consequently, the pressure drop inside the shell is reduced in comparison to the 25% and 35% baffle cut configurations.

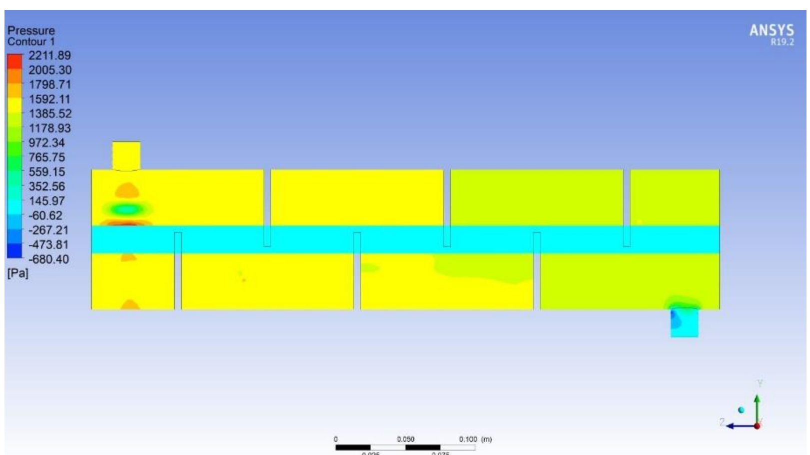

Figure 4. 24 Pressure contour at 45% baffle cut

4.2.3 Simulation with 4 number of baffles

This involves simulating a shell-and-tube configuration with 4 baffles and three different baffle cuts (25%, 35%, and 45%). The velocity, streamline, temperature and pressure contours at cross section are shown in below figures.

4.2.3.1 Velocity Contour

The velocity plots across the cross-section for various baffle cuts (25%, 35%, and 45%) are displayed in Fig. 4.25, 4.27, and 4.29, respectively. In this configuration, the recirculation zone is wider in comparison to the previous cases. The wider recirculation zone results in reduced contact between the hot fluid and cold fluids, as depicted below.

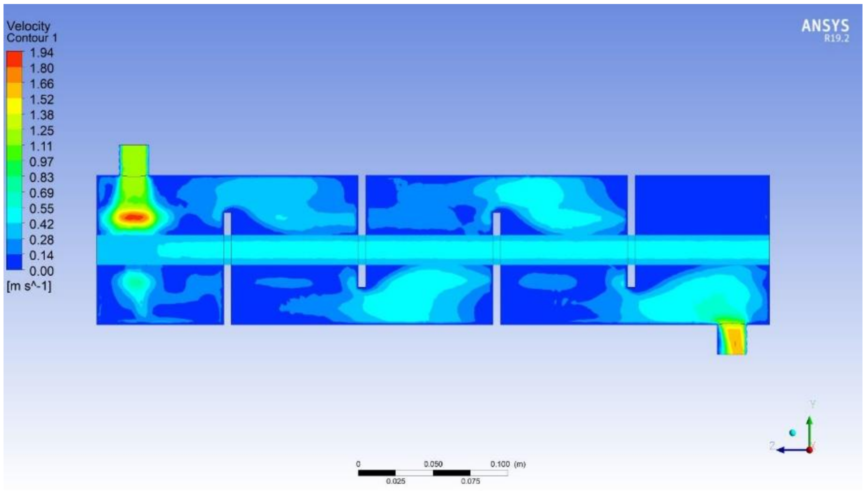

Figure 4. 25 Velocity contour at 25% baffle cut

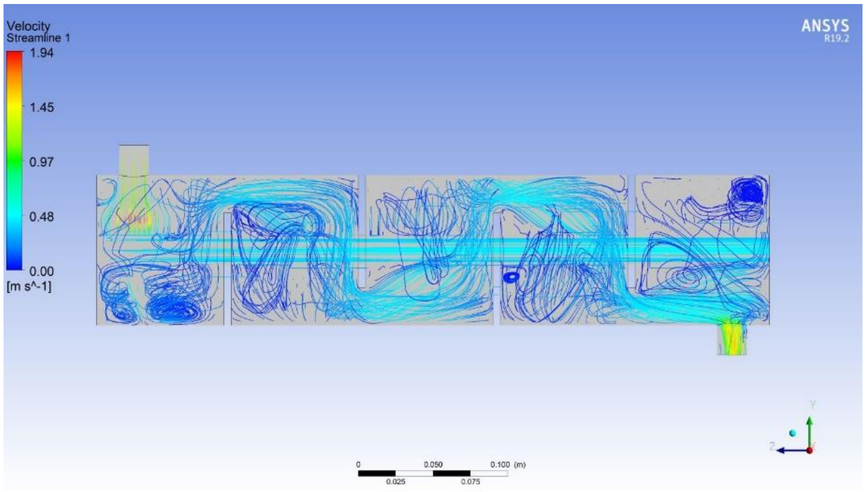

Figure 4. 26 Streamline at 25% baffle cut

The velocity plots across the cross-section with a baffle cut of 35% are presented in Fig. 4.27. In comparison to the previous case, recirculation zone formation at the backside of the baffles is less in this case. Moreover, the heat exchanger's efficiency declines with time due to a reduction in heat transfer in these recirculation zones.

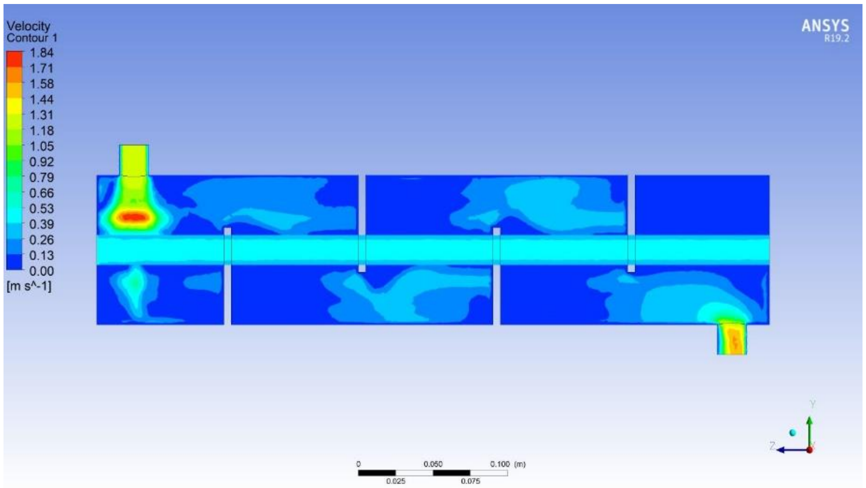

Figure 4. 27 Velocity at 35% baffle cut

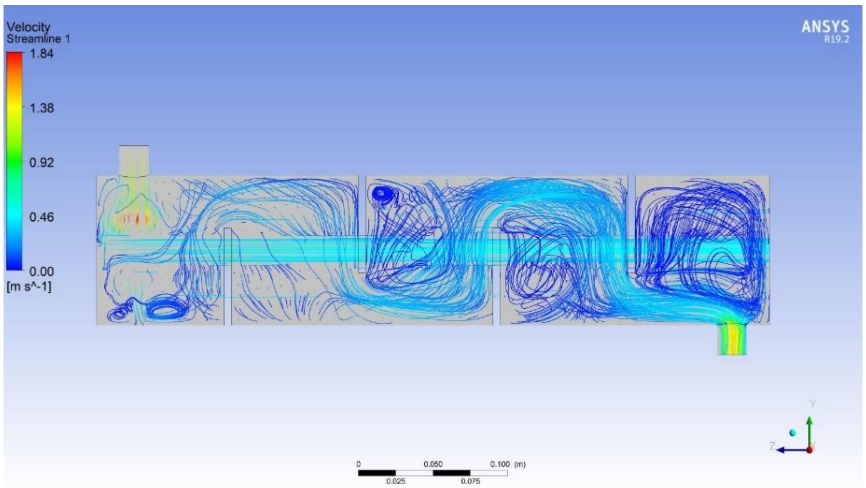

Figure 4. 28 Streamline at 35% baffle cut

The velocity plots across the cross section with baffle cut 45% is shown below in Fig. 4.29. With increase in the baffle cut, the area for the formation of recirculation zone decreases and, in this instance, the heat exchanger's efficiency is decreased and there is less heat transfer.

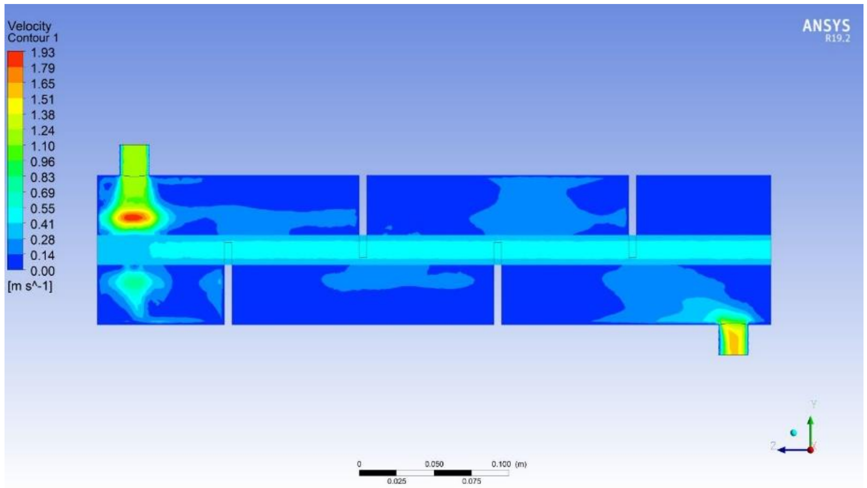

Figure 4. 29 Velocity at 45% baffle cut

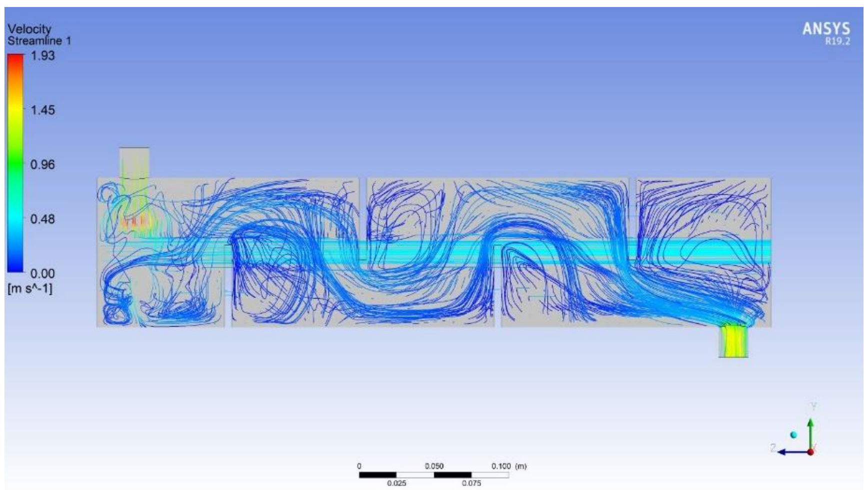

Figure 4. 30 Streamline at 45% baffle cut

4.2.3.2 Temperature Contour

The temperature contour plots across the cross-section with a baffle cut of 25% are displayed in Fig. 4.31. The contours indicate that a significant portion of heat transfer occurs in the end portion, primarily due to less contact between hot fluids and tubes. As noted earlier, the increased recirculation zone in this case results in reduced heat transfer than the other two cases. The outlet temperature at shell side is found to be 355.72K.

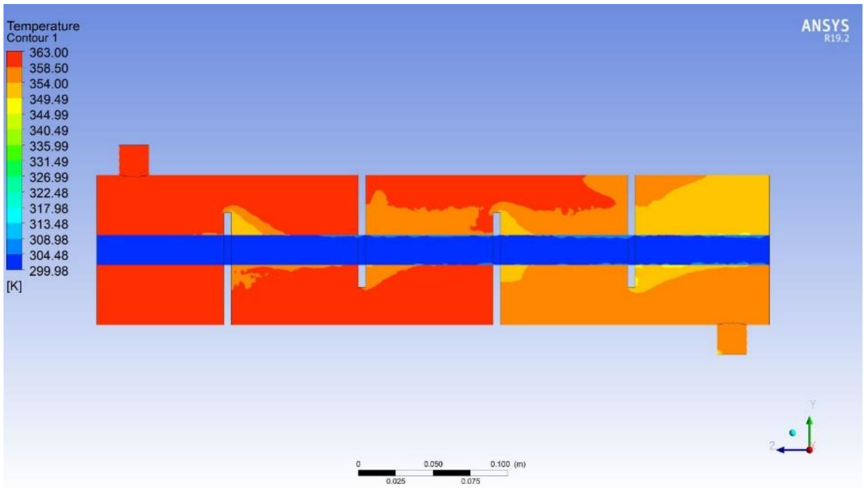

Figure 4. 31 Temperature contour at 25% baffle cut

The temperature contour plots across the cross-section with a baffle cut of 35% are presented in Fig. 4.32. The recirculation zone widens with increasing baffle spacing and cut, which leads to less hot fluid contact with the tube than in the previous case. In this configuration, the reduced contact results in a lower heat transfer efficiency.

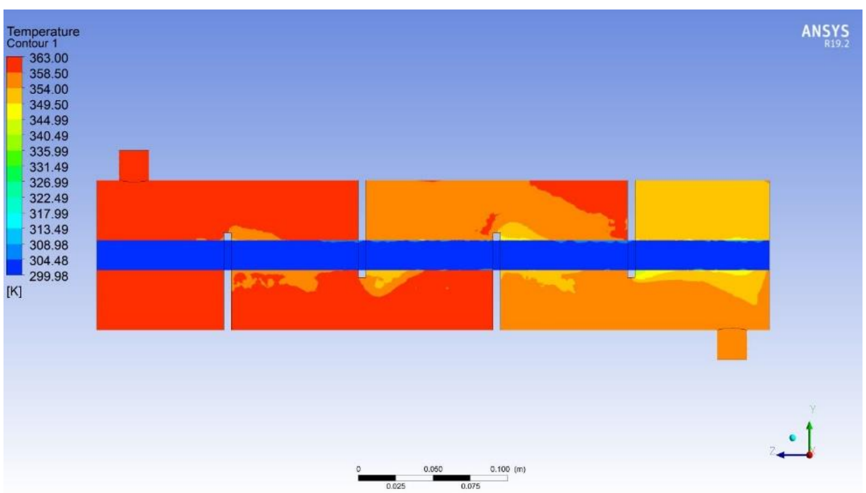

Figure 4. 32 Temperature contour at 35% baffle cut

The temperature contour plots across the cross-section with a baffle cut of 45% are illustrated in Fig. 4.33. In this configuration, with a 45% baffle cut, the fluid can easily pass across the baffle, resulting in reduced heat transfer compared to the cases with 25% and 35% baffle cuts.

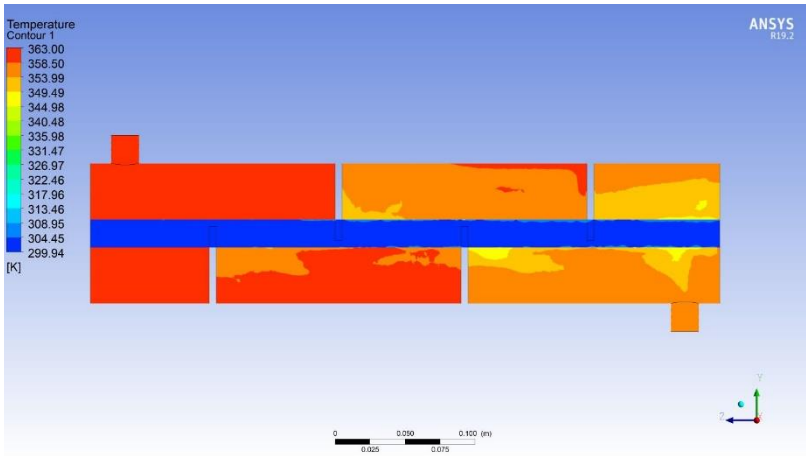

Figure 4. 33 Temperature contour at 45% baffle cut

4.2.3.3 Pressure Contour

The pressure contour for a baffle cut of 25% is shown in Fig. 4.22. It is clear that the pressure drop is greater for a shell and tube heat exchanger with a 25% baffle cut than for the other two configurations, and it decreases with increasing baffle cut. This higher pressure drop necessitates a greater pumping power to maintain the flow, subsequently decreasing the overall system efficiency.

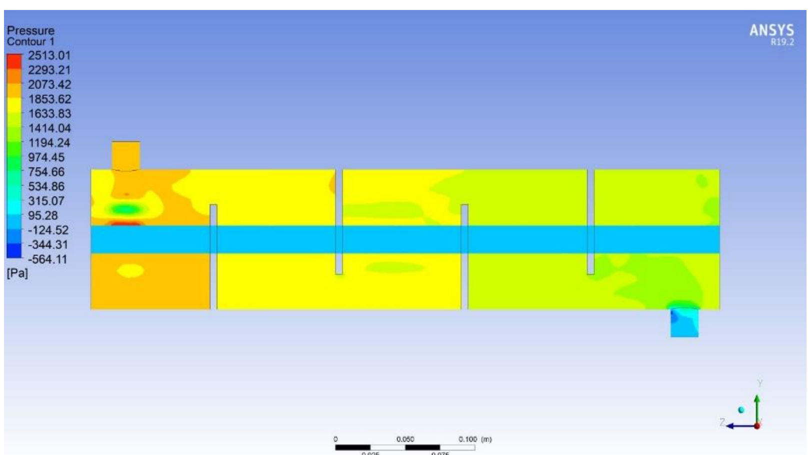

Figure 4. 34 Pressure contour at 25% baffle cut

The pressure contour for a baffle cut of 35% is presented in Fig. 4.35. Notably, an increase in baffle cut corresponds to a reduction in pressure drop. In this configuration, less pumping power is required to maintain the flow compared to the case with a 25% baffle cut.

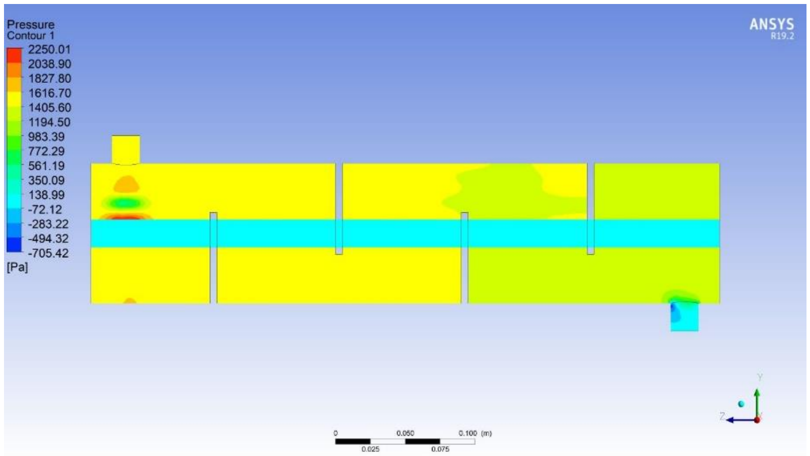

Figure 4. 35 Pressure contour at 35% baffle cut

Similarly, the pressure contour for a baffle cut of 45% is illustrated in Fig. 4.36. This case has a lower pressure drop than the other two and thus requires less pumping power to maintain the flow, which results in the lowest cost.

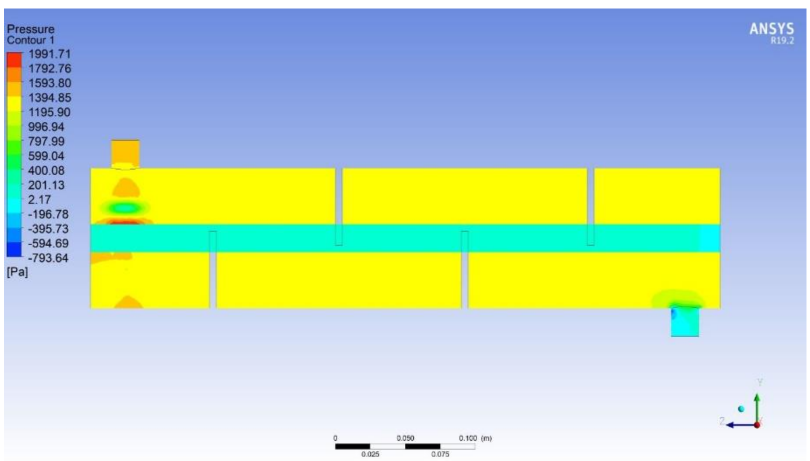

Figure 4. 36 Pressure contour at 45% baffle cut

4.3 Validation of Results

For the validation and comparative analysis of the current CFD model, the simulation study conducted by Mishra in 2016 has been selected. The study examined shell and tube heat exchangers with different baffle counts. The validation process involves assessing pressure drop and overall heat transfer for configurations with 8, 6, and 4 baffles. The graphs below compare shell and tube heat exchangers with 8, 6, and 4 baffles. The comparative analysis consistently indicates a trend of increased heat transfer and pressure drop with the number of baffles, while maintaining a constant baffle cut value. Notably, shell and tube heat exchanger with 8 baffles exhibits the highest level of heat transfer. Specifically, current research indicates a 6% enhancement in heat transfer for the configuration with 8 baffles compared to 4 baffles. This aligns with existing literature findings, further validating the accuracy and reliability of our simulated results.

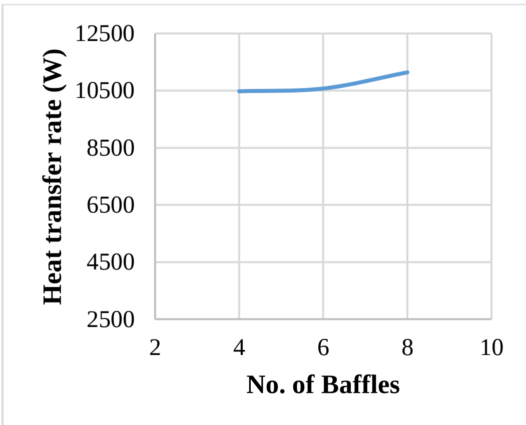

Figure 4. 37 Heat transfer variation with no. of baffles in current research

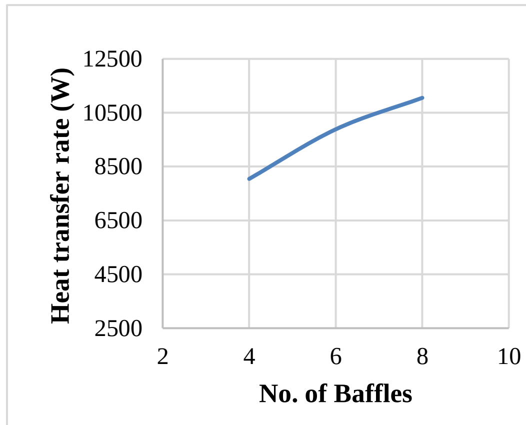

Figure 4. 38 Heat transfer variation with no. of baffles in existing research

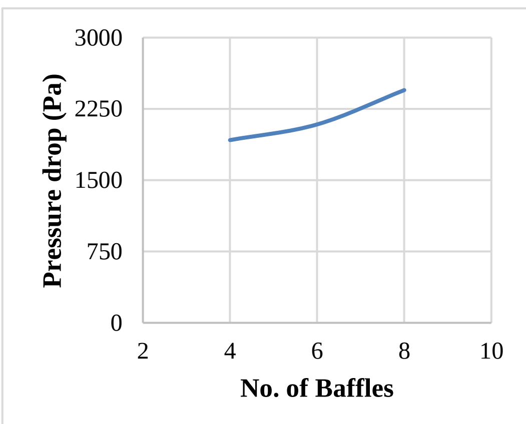

Figure 4. 39 Pressure drop variation with no. of baffles in current research

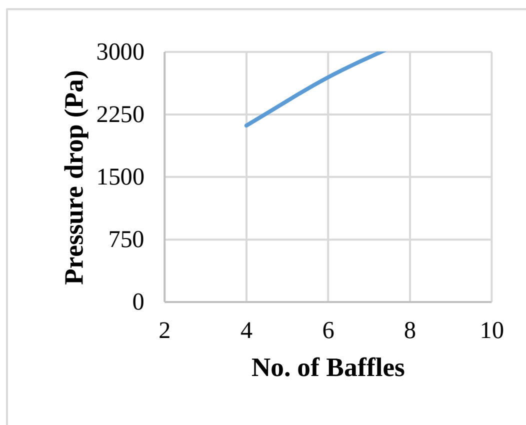

Figure 4. 40 Pressure drop variation with no. of baffles in existing research

4.4 Effect of number of baffles and baffle cut on heat transfer

The simulation results indicate that with increase in number of baffles for a constant length of the shell and tube heat exchanger, the space between adjacent baffles decreases, leading to reduced area available for cross flow. This reduction in space results in the development of high turbulence, contributing to an elevated heat transfer rate. Furthermore, because there are more baffles, the fluid must move through the shell a longer distance which essentially increases the area of heat transfer and consequently the rate of heat transfer. The heat transfer is observed to be 6% higher with 8 baffles compared to 4 baffles.

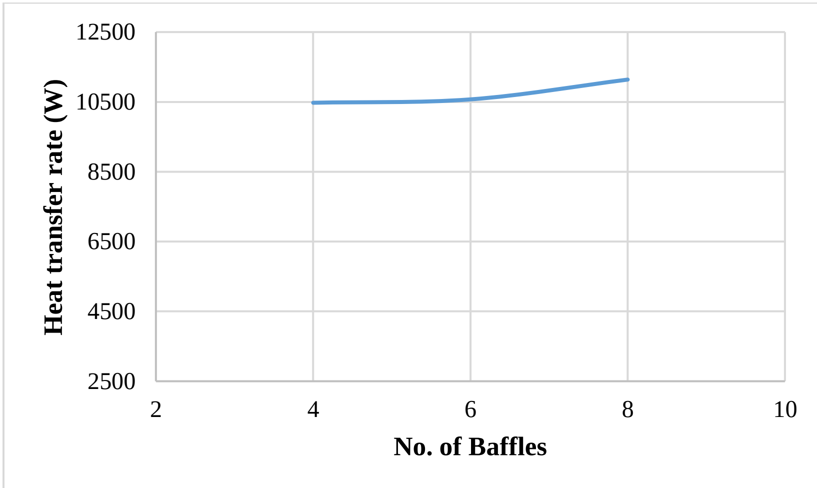

Figure 4. 41 Heat transfer rate variation with different number of baffles

Similarly, it is observed that heat transfer decreases as baffle cut increases. In comparison to a 45% baffle cut, a 25% baffle cut results in 5% more heat transfer for the same number of baffles.

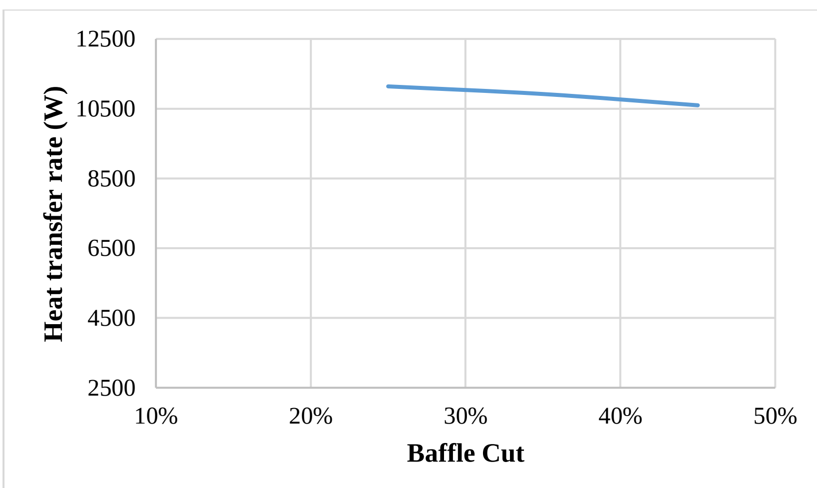

Figure 4. 42 Heat transfer rate variation with different baffle cut

4.5 Effect of number of baffle and baffle cut on pressure drop

The CFD results reveal that the pressure drop in the shell and tube heat exchanger increases with a higher number of baffles. This elevated more pumping power required to maintain the flow because of this increased pressure drop. The increased baffle count narrows the gap between successive baffles, causing the fluid to follow a more constrained path. As the fluid moves through this restricted path, its pressure decreases. The pressure drop is found to be 21.48% higher with 8 baffles compared to 4 baffles in the heat exchanger.

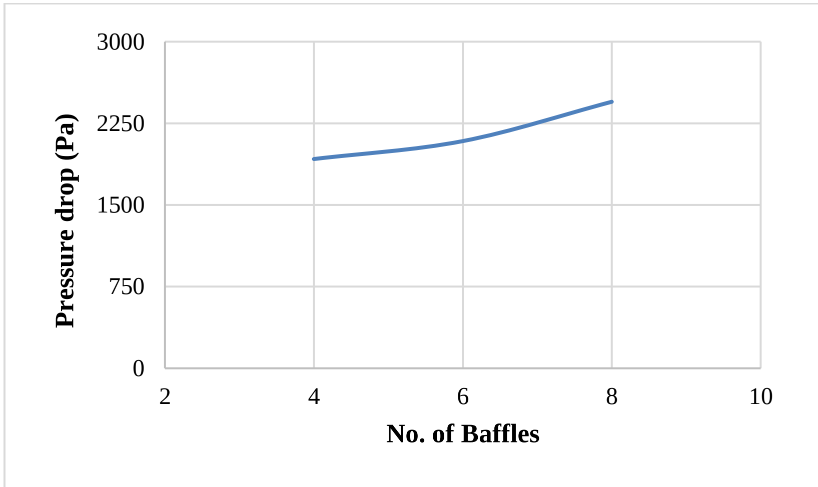

Figure 4. 43 Pressure drop variation with different number of baffle

Similarly, analysis of CFD results indicates that an increase in baffle cut size corresponds to a noticeable decrease in the pressure drop within the shell-and-tube heat exchanger. Specifically, in shell and tube configuration with a 45% baffle cut the shell side pressure drop is found to be 27.55% less compared to the scenario with a 25% baffle cut, while maintaining the same number of baffles.

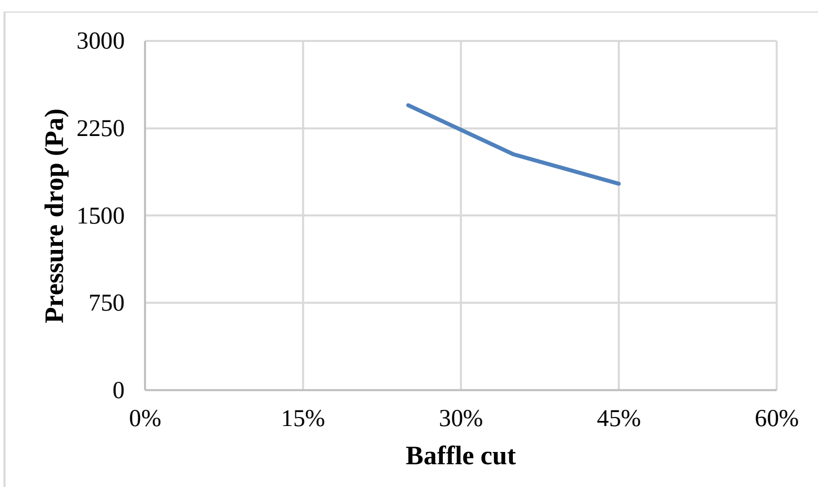

Figure 4. 44 Pressure drop variation with different baffle cut

CHAPTER 5: CONCLUSIONS AND RECOMMENDATIONS

5.1 Conclusions

In this research, CFD is used to analyze a small shell-and-tube heat exchanger. Several CFD simulations are conducted for shell-and-tube heat exchangers featuring varying numbers of baffles and baffle cuts. Computational modeling proves to be an efficient technique for studying this type of thermal element. By adjusting the baffles number and baffle cut values, nine different models of shell and tube heat exchangers are created and simulations are performed to examine their impact on heat transfer and pressure drop. The study observes the influence on the heat exchanger by comparing results obtained with different combinations of baffles and baffle cuts. With an appropriate baffle cut orientation, the modified baffle arrangement increases turbulence and enhances the performance of heat exchanger.

Examining the velocity profile facilitates understanding of the cross-sectional flow distribution at various heat exchanger locations. The velocity contour and streamline reveal the formation of a recirculation zone behind baffles, where the flow is reduced. These zones hinder heat transfer, and their efficiency decreases over time, affecting the overall heat exchanger efficiency. Furthermore, the pressure drop within the shell is found to be reduced by increasing the baffle cut inside the shell. Notably, a 25% baffle cut results in higher pressure drop compared to other percentages, decreasing as the baffle spacing and cut increase. Higher pressure drop necessitates more pumping power, reducing the overall system efficiency.

The study aims to provide insights into the effects of changing two parameters, namely number of baffles and baffle cut inside the shell. The findings demonstrate that heat transfer on the shell side is reduced by a higher baffle cut and increases as the number of baffles increases. More baffles increase the significance of baffle cut on shell-side heat transfer. Heat transfer for 8 baffles with 25% baffle cut is 12.22% higher than for 4 baffles with 35% baffle cut. This study also encompasses an analysis of pressure drop, revealing that the pressure drop rises with an increase in baffle number and declines with an increase in baffle cut. The pressure drop is 21.48% higher with 8 baffles compared to 4 baffles.

Similarly, for an equal number of baffles, a 45% baffle cut yields a 27.55% lower pressure drop than a 25% baffle cut. Finally, this study highlights the importance of achieving the ideal configuration for shell and tube heat exchangers by carefully balancing number of baffles and baffle cut, which will lead to increased heat transfer and decreased pressure drop.

5.2 Recommendations

The objective of this study is to create a detailed model of a small shell-and-tube heat exchanger in order to investigate the flow, temperature fields, and pressure drop within the shell. The findings of the CFD simulation provides insights knowledge on the pressure drop, temperature distribution, and shell side flow under fixed velocity inlets. The analysis extends to the exploration of varying baffle cut and baffle spacing in the simulation results. Further investigation of this thesis can be pursued by considering the following key points:

i. Employing CFD in conjunction with complementary experiments has the potential to expedite the shell-and-tube heat exchanger design process and raise the overall standard of the final design.
ii. Altering parameters such as outer diameter, inner diameter, and material properties enables the shell and tube heat exchangers design that can optimize heat transfer coefficients cost-effectively.

REFERENCES

B. Peng, Q. W. (2007). An experimantal study of Sheel-and-Tube Heat Exchnagers with continuous helical baffles. *Journal of Heat Transfer, vol. 129*, pp. 1425–1431.

Bhalkikar, C. N. (2014). CFD Analysis of Shell and Tube Heat Exchanger to Study the Effect of Baffle Cut on the Pressure Drop and Heat Transfer Coefficient. *International Journal for Scientific Research and Development, vol. 2*, 649–654.

Ender Ozden, I. T. (2010). Shell side CFD analysis of a small shell-and-tube heat exchanger. *Energy Conversion and Management, vol. 51*, pp. 1004–1014.

EnggCyclopedia. (2023). Retrieved from Enggcyclopedia: https://enggcyclopedia.com/2023/02/baffles-in-heat-exchangers.

Gnielinski, E. S. (1997). Pressure drop on the shell side of shell-and-tube heat exchangers with segmental baffles. *Chemical Engineering and Processing, vol. 36*, pp. 149–159.

Ipieca. (2022). *Heat Exchangers (2022)*. Retrieved from Ipieca: https://www.ipieca.org/resources/energy-efficiency-database/heat-exchangers-2022.

J. Prasanna, H. R. (2013). A numerical analysis of hydrodynamic and heat transfer effects of shell and tube heat exchanger for different baffle space and cut. *Mechanica Confab, vol. 2*.

K. Kiran, A. M. (2014). Investigation of baffle spacing effect on shell side heat transfer characteristics in shell and tube heat exchanger using computational. *Thermal Engineering, vol. 73*, 26022–26026.

Koli, A. D. (July 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,*, 1–7.

Mishra, A. (2016). CFD Analysis of Shell and Tube Heat Exchanger with Segmental Baffles.

Noor Afsar, D. M. (2018). CFD Analysis of Shell and Tube Heat Exchanger with Different Baffle Orientation and Baffle Cut. *Department of Mechanical Engineering, Khulna University of Engineering and Technology, Khulna-9203*.

Othman, K. H. (2009). CFD Simulation of Heat Transfer in Shell and Tube Heat Exchanger. *University Malaysia Pahang*.

Ozden, E. (2010). Shell side CFD analysis of a small shell-and-tube heat exchanger. *Energy Conversion and Management 51*, 1004–1014.

Rajput, R. (2012). *Heat and Mass Transfer.* New Delhi: S.Chand & Company Ltd.

Raju, K. S. (2011). *Fluid Mechanics, Heat Transfer and Mass Transfer.* Singapore: A Jhon Wiley &Sons, Inc.,Publication.

Sarkar, S. (2023, May). *Ketiv*. Retrieved from Ketiv.com: https://ketiv.com/blog/what-is-computational-fluid-dynamics-and-why-you-need-it.

Sunil, A. S. (2014). Design of Shell and Tube Heat Exchanger using Computational Fluid Dynamics Tools.

Swanand, G. a. (June 2020). Numerical simulation of the effect of baffle cut and baffle spacing on shell side heat exchanger performance using CFD. *Chemical Product and Process Modeling, vol.16*, 145-154.

Webbusterz Engineering. (2023). Retrieved from Webbusterz.org: https://www.webbusterz.org/baffles-in-heat-exchangers

APPENDIX

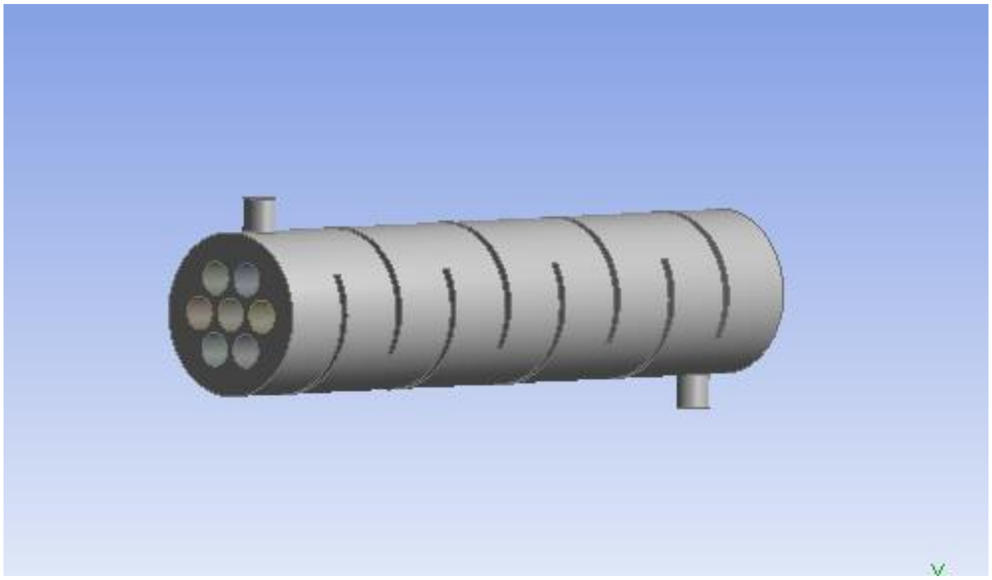

Figure A. 1: Overall geometry of STHE with 8 number of baffles

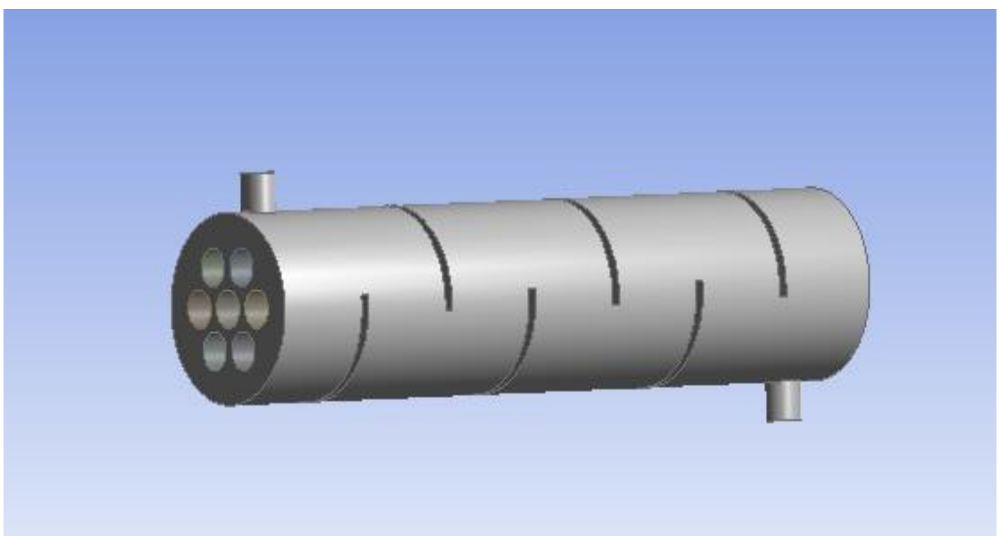

Figure A. 2: Overall geometry of STHE with 6 number of baffles

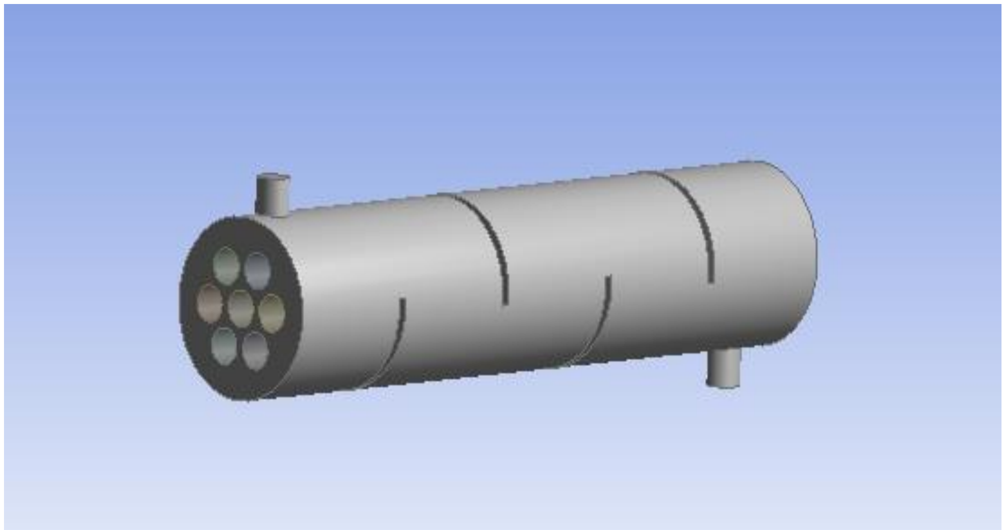

Figure A. 3: Overall geometry of STHE with 4 number of baffles

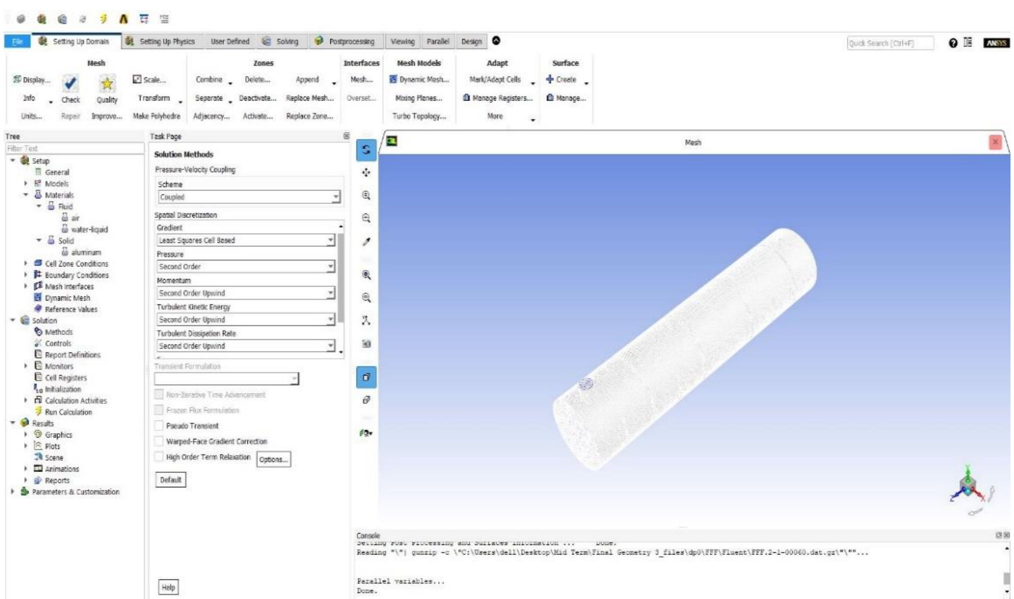

Figure A. 4: Solution Method

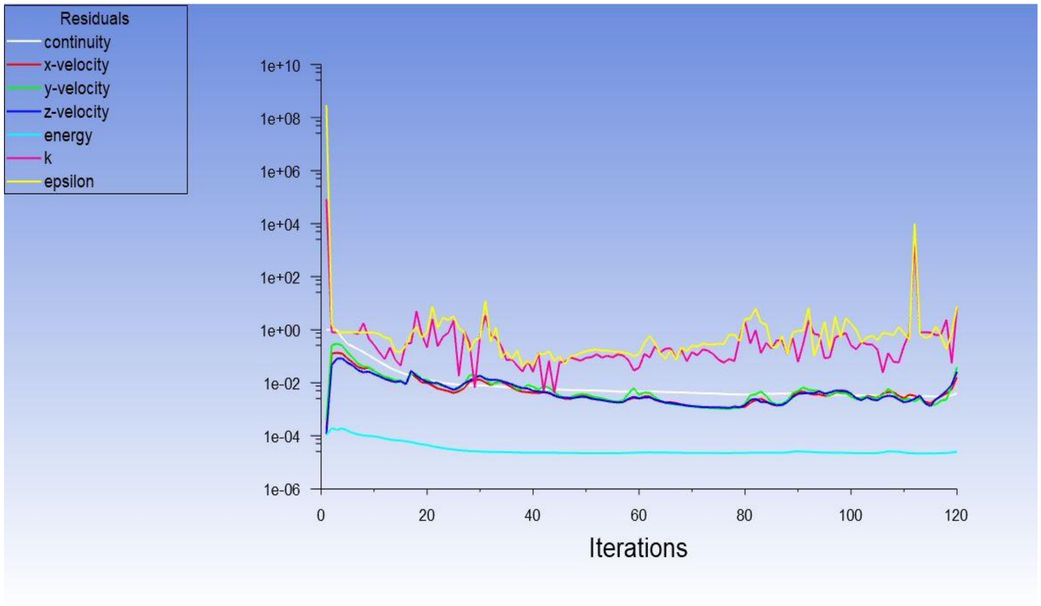

Figure A. 5: Iteration result