Research Journal of Engineering Technology and Medical Sciences (ISSN: 2582-6212), Volume 07, Issue 01, March-2024
Available at www.rjetm.in/

Optimization of Earth Tube Heat Exchanger Design for Enhanced Cooling Efficiency of PV Panels

[1]Raju Singh, [2]Prof. Tanmay Awasthi
Department of Mechanical Engineering, Sagar Institute of Research & Technology, Bhopal
Department of Mechanical Engineering, Sagar Institute of Research & Technology, Bhopal

* Corresponding Author: Raju Singh

Abstract: *This study focuses on optimizing the design of Earth Tube Heat Exchangers (ETHX) to maximize cooling efficiency for photovoltaic (PV) panels. Through numerical simulations and computational fluid dynamics (CFD) analysis, optimal geometrical parameters such as length, diameter, and fin dimensions are determined. A comprehensive literature review identifies gaps and opportunities for innovation in ETHX technology. Additionally, the impact of varying airflow velocities and the introduction of extrusions on ETHX performance are investigated. The study quantitatively compares different configurations, highlighting Case 3 as the optimal choice with significant improvements in pressure drop reduction, temperature decrease, and outlet velocity increase. The conclusive outcome emphasizes Case 3 as the prime candidate for applications requiring efficient heat exchange and reduced energy consumption in ETHX systems.*

Keywords: *Earth Tube Heat Exchanger, PV Panel Cooling, Computational Fluid Dynamics, Optimization, Extrusions, Heat Transfer.*

I. INTRODUCTION

Heat exchangers have a rich history dating back to ancient civilizations. From simple fire-warmed water vessels to intricate modern designs, they have evolved to become indispensable components in various sectors. This paper delves into the historical journey of heat exchangers, emphasizing key developments and individuals who have contributed to their advancement. The earliest records of heat exchange date back to ancient civilizations. The ancient Greeks and Romans used water circulation systems to heat buildings and baths. The "hypocaust" system, a form of underfloor heating, employed ducts to circulate hot air, demonstrating the rudimentary principles of heat transfer. The concept of heat exchange dates back thousands of years, with early examples such as the use of fire pits and steam baths in ancient times. However, the modern origins of heat exchanger technology can be traced back to the 18th century industrial revolution.

The Industrial Revolution marked a pivotal era for heat exchangers. In the 19th century, engineers such as Marc Seguin and John Hague pioneered the development of early steam condensers and evaporators. The invention of the shell-and-tube heat exchanger by Ivan Sergeevich Kurnakov in the early 20th century significantly improved heat exchange efficiency and became a cornerstone in various industries. The mid-20th century witnessed the emergence of the plate heat exchanger. Driven by the need for compact and efficient designs, Dr. Richard Seligman's innovation of corrugated plates in 1923 paved the way for the development of modern plate heat exchangers. These designs found applications in the food, chemical, and HVAC industries. The 20th century also witnessed the integration of heat exchangers in automotive engineering and refrigeration systems. Automobile radiators, oil coolers, and intercoolers became integral to engine efficiency. Similarly, advancements in refrigeration and air conditioning systems relied heavily on heat exchangers for improved energy conservation.

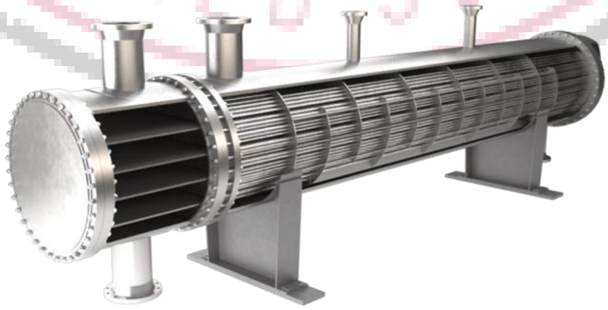

Figure 1. Shell Heat Exchanger

The latter half of the 20th century saw the rise of computational methods and simulations that revolutionized heat exchanger design. Computer-aided design (CAD) and computational fluid dynamics (CFD) enabled engineers to optimize heat exchanger configurations and fluid flow patterns for enhanced performance. As society moves towards sustainable energy solutions, heat exchangers play a critical role in renewable energy systems like solar thermal collectors and geothermal heat pumps. These developments align with a global emphasis on energy efficiency and environmental sustainability. In recent decades, miniaturized microchannel heat exchangers have gained prominence due to their high compactness. Advanced manufacturing methods like additive manufacturing are enabling optimized heat exchanger geometries. Overall, heat exchangers have evolved tremendously over the past two centuries and continue to be indispensable in engineering systems.

Heat exchangers play a crucial role in various industrial and domestic applications by facilitating efficient heat transfer between fluids. This comprehensive study delves into the fundamentals of heat exchangers, exploring their types, working principles, design considerations, and diverse applications. By examining their significance in industries such as energy production, HVAC systems, chemical processes, and automotive engineering, this paper highlights the importance of heat exchangers in modern engineering. Heat exchangers are vital devices utilized in a wide range of engineering applications to transfer thermal energy between two or more fluids, gases, or solid substances. Heat exchangers are essential components in engineering systems that involve the transfer of thermal energy between fluids at different temperatures. They find extensive use in various sectors, from power plants to refrigeration units, by optimizing heat transfer efficiency and energy conservation. The primary function of a heat exchanger is to facilitate the heating or cooling of a fluid without mixing it with another fluid. Heat exchangers enable the recovery and reuse of heat that would otherwise be wasted, thereby conserving energy and reducing environmental impacts.

A. Earth Tube Heat Exchangers

Harnessing Earth's Energy for Sustainable Heating and Cooling

- [1] The Need for Sustainable Heating and Cooling:
  Heating, ventilation, and air conditioning (HVAC) systems are crucial for maintaining comfortable indoor environments, but they often consume significant amounts of energy. As the world faces increasing concerns about energy efficiency and environmental sustainability, innovative solutions are needed to reduce the energy demands of HVAC systems. One such solution is the Earth Tube Heat Exchanger (ETHX), a passive and sustainable technology that leverages the Earth's stable subsurface temperatures to precondition outdoor air before it enters buildings. This article provides an in-depth exploration of Earth Tube Heat Exchangers, their principles of operation, design considerations, benefits, and applications.
- [2] A Sustainable HVAC Solution:
  The Earth Tube Heat Exchanger is an ingenious system that taps into the Earth's natural temperature stability, providing both cooling and heating benefits to buildings. Its operation is rooted in the principle of heat exchange with the Earth's subsurface. By circulating outdoor air through a network of buried tubes, the ETHX modifies the air's temperature, making it more suitable for indoor comfort. Importantly, this process requires minimal energy input and reduces the load on traditional HVAC systems.
- [3] Operating Principles of Earth Tube Heat Exchangers:
  The core principle of Earth Tube Heat Exchangers is straightforward: outdoor air is drawn into the system, circulated through underground tubes, and then introduced into the building after undergoing temperature modification. However, the specifics of operation vary based on the season.

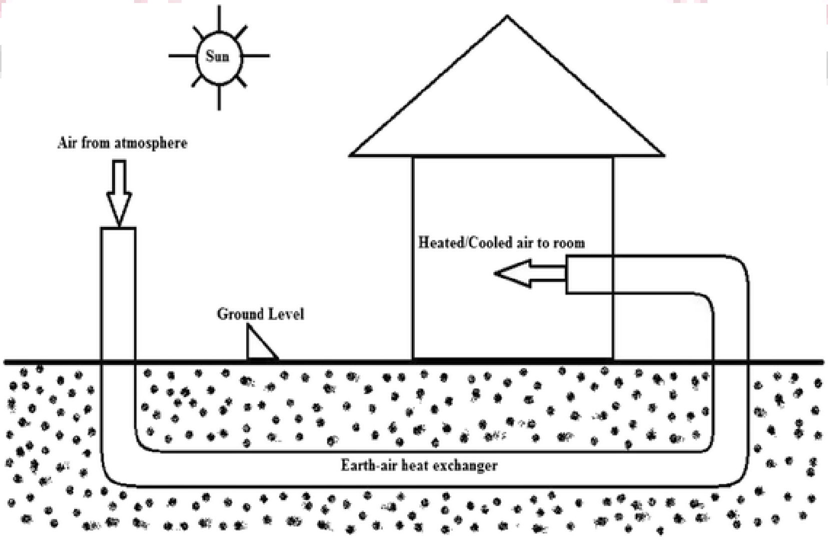

Figure 1.3.1 Geo Thermal Earth-Air Heat Exchanger

Summer Cooling

[4] During the sweltering summer months, when outdoor temperatures are higher than desired indoor conditions, the Earth's subsurface offers a valuable resource. Buried beneath the frost line, typically a few feet (1 to 2 meters) deep, the Earth maintains a relatively stable and cooler temperature, usually around 50 to 60°F (10 to 16°C), depending on location and depth. The ETHX capitalizes on this natural cooling potential.

The cooling process in summer involves several steps:

- Air Intake and Filtration: Outdoor air is drawn into the system through an intake equipped with filters to remove particulates and contaminants.
- Underground Tube Network: The incoming warm outdoor air is directed into a network of buried tubes. These tubes are often made of high-density polyethylene (HDPE) or similar materials chosen for their durability and resistance to corrosion.
- Heat Exchange: As the outdoor air flows through the underground tubes, it exchanges heat with the surrounding Earth. Because the Earth is cooler than the outdoor air, the air is naturally cooled as it passes through the tubes.
- Air Distribution: After being pre-cooled in the ETHX, the air is introduced into the building's ventilation system. From there, it is distributed to various areas within the building, contributing to a comfortable indoor environment.

Winter Heating:

- [5] Conversely, during the chilly winter months when outdoor temperatures drop below the desired indoor levels, the Earth Tube Heat Exchanger can provide a sustainable source of warmth. This process involves the following steps:
- Air Intake and Filtration: As in the summer, outdoor air is drawn into the system through an intake with filters to ensure air quality.
- Underground Tube Network: The incoming cold outdoor air is directed into the buried tube network.
- Heat Exchange: In winter, the Earth's subsurface is warmer than the outdoor air. As a result, the outdoor air is pre-warmed as it flows through the underground tubes, raising its temperature.
- Air Distribution: After being pre-heated in the ETHX, the air is introduced into the building's ventilation system. It is then distributed throughout the building to maintain a comfortable indoor temperature.

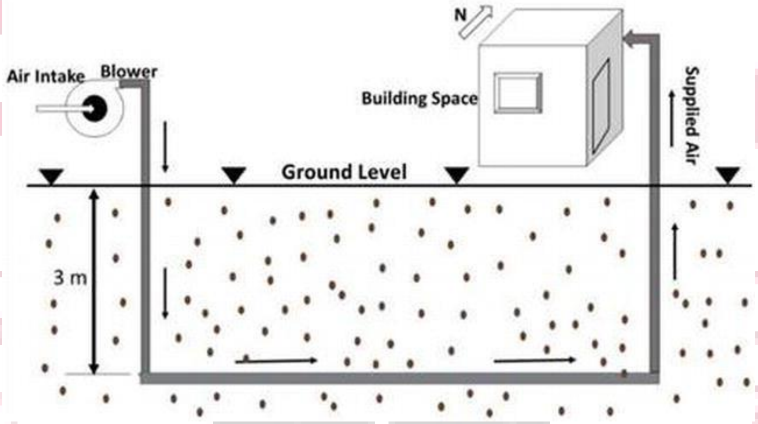

Figure: Earth Air Tunnel Heat Exchanger

Design Considerations for Earth Tube Heat Exchangers:

Effective and efficient design is essential for the successful implementation of Earth Tube Heat Exchangers. Several critical factors must be considered during the design phase:

- Tube Material:
  The choice of tube material is crucial. Tubes are typically made of high-density polyethylene (HDPE) or similar materials known for their corrosion resistance and durability. Given their underground location, these materials must withstand potentially harsh soil conditions and moisture.
- Tube Length and Depth:
  The length and depth of the buried tubes depend on several factors, including climate, soil type, and the desired degree of temperature modification. Longer tubes can provide more significant heat exchange, but they may require more land area. Depth is important because the Earth's temperature stabilizes below the frost line, and the depth at which tubes are buried affects the efficiency of heat exchange.
- Moisture Control:
  Proper moisture control is essential to prevent condensation inside the tubes. Excessive moisture can lead to mold growth and reduce the system's efficiency. Adequate insulation and drainage mechanisms are often incorporated into the design to address this concern.
- Filtration:

To ensure good indoor air quality and prevent dust and contaminants from entering the system, air intakes are equipped with filters. Regular maintenance is essential to keep these filters clean and functional.

B. Benefits of Earth Tube Heat Exchangers

[6] Earth Tube Heat Exchangers offer numerous advantages, making them an attractive choice for sustainable HVAC solutions:

- Energy Efficiency:

One of the most significant benefits of ETHX systems is their ability to significantly reduce the energy consumption associated with HVAC systems. By pre-conditioning outdoor air, the load on heating and cooling equipment is reduced, leading to substantial energy savings.

Environmental Sustainability:

- ETHX technology aligns with the global push for environmental sustainability. Its passive operation minimizes the need for energy-intensive mechanical cooling and heating, reducing a building's carbon footprint.

Reduced Operating Costs:

- Lower energy consumption translates to reduced utility bills and operational costs for building owners and occupants. Over time, these savings can be substantial, offsetting the initial investment in ETHX installation.

Improved Indoor Comfort:

- Earth Tube Heat Exchangers contribute to improved indoor comfort by providing pre-conditioned air that is closer to the desired temperature. This enhances occupant satisfaction and productivity.

Reliability and Durability:

- ETHX systems are known for their reliability and durability. Properly designed and maintained, they can provide consistent performance for many years.

Applications of Earth Tube Heat Exchangers

[17] They are commonly used in both commercial and residential buildings to provide sustainable heating and cooling solutions. These systems can be integrated into new construction or retrofitted into existing structures.

- Educational Institutions: Schools and universities are increasingly adopting ETHX technology to reduce their energy consumption and promote sustainability.
- Industrial Facilities: In industrial settings, Earth Tube Heat Exchangers can help control indoor temperatures and humidity levels, benefiting manufacturing processes and employee comfort.
- Agricultural Facilities: Greenhouses and agricultural buildings can utilize ETHX systems to create ideal growing conditions for crops and plants.
- Office Buildings: Many office buildings are incorporating Earth Tube Heat Exchangers into their HVAC systems to achieve energy.

II. LITERATURE REVIEW

Kevin Taurines et al. (2021) [8] In the study by Kevin Taurines and his team in 2021, an innovative Earth-to-Air Heat Exchanger (EAHE) is examined, offering solutions to several technical challenges typically encountered in traditional EAHE systems. The research introduces a sophisticated numerical modeling approach, featuring a time-dependent three-dimensional finite volume model. This model takes into account the intricate dynamics of coupled heat and moisture transfers, as well as the complexities of boundary conditions, which encompass external factors like weather conditions, building characteristics, and the influence of the water table. Of particular note is the proposal of a novel numerical method that involves a transition from a potential-based to a flux-based system of equations. This approach likely represents a significant advancement in the modeling and analysis of EAHE systems, promising improved accuracy and efficiency in predicting their performance under diverse environmental scenarios.

Maneesh Kaushal (2021) [9] In a study conducted by Maneesh Kaushal in 2021, the thermal performance of an Earth-to-Air Tunnel Heat Exchanger (GEAHE) designed for passive winter preheating and summer cooling was comprehensively evaluated. The research focused on assessing the impact of various operating parameters, including the choice of pipe or tube material, soil thermal conductivity, and air velocity, on the efficiency of the GEAHE system, particularly with regard to its suitability for use in the Lower Himalayan Region. The findings of the study indicate noteworthy trends: during summer, as the length of the underground pipe or tube increases, the exit temperature decreases, suggesting effective cooling capabilities. Conversely, in the winter season, the exit temperature increases with longer underground pipes or tubes, which signifies enhanced preheating potential. These results provide valuable insights into optimizing GEAHE systems for both seasonal heating and cooling requirements in specific geographic regions.

S.F. Ahmed et al. (2021) [10] In a study conducted by S.F. Ahmed and colleagues in 2021, the cooling performance of a vertical Earth-to-Air Heat Exchanger (EAHE) in a subtropical climatic zone was rigorously investigated through a comprehensive parametric analysis. The research involved the development of a thermal transient model for the vertical EAHE using ANSYS Fluent, facilitating a detailed examination of various parameters that significantly influence its cooling capabilities. Key parameters studied included air velocity, pipe thickness, pipe diameter, depth, length, and the material of the pipes used in the EAHE system. The findings of this study highlight several crucial insights: the cooling performance of the EAHE system is notably influenced by pipe diameter, air velocity, and pipe length. However, no substantial impact was observed based on the choice of pipe material. The optimized performance of the EAHE system led to a significant reduction in outlet air temperature, achieving a noteworthy 8.21°C decrease. These results contribute valuable knowledge for designing and optimizing vertical EAHE systems in subtropical climates, with potential applications in energy-efficient cooling.

Nethra MR & Kalidasan B (2020) [11] In the study conducted by Nethra MR and Kalidasan B in 2020, the focus is on Photovoltaic (PV) panels, which are composed of semiconducting materials designed to convert solar energy into electrical energy. It's well-established that the best operating conditions for PV panels, according to Standard Test Conditions, occur at a temperature of 25°C. However, in countries such as Saudi Arabia, Libya, Yemen, and others, the atmospheric temperatures can often soar to around 40°C or even higher. The primary objective of this research paper is to address this challenge by proposing a numerical design for a heat exchanger. This heat exchanger is meticulously designed to exact dimensions with the specific purpose of providing thermal cooling to PV panels. The underlying aim is to enhance the overall performance of PV panels, ensuring that they continue to function efficiently even under the extreme heat conditions experienced in these regions. By offering a cooling solution tailored to the elevated temperatures, this research contributes to the advancement of sustainable energy solutions, making PV panels more resilient and effective in hot climates.

Andrew Zajch et al. (2020) [12] In the research conducted by Andrew Zajch and his team in 2020, a comprehensive assessment of heating and cooling conditions was undertaken, utilizing a degree-hour approach for a multi-tube Earth-to-Air Heat Exchanger (EAHE) system located in Aichi, Japan. This study categorizes these conditions into detrimental and beneficial states. Furthermore, the research involved the development of linear models for both winter and summer seasons, based on factors such as the inlet air temperature and the temperature of the surrounding soil. These models were crucial for understanding the dynamics of heating and cooling within the EAHE system. A notable aspect of this research is the consideration of temporally decomposed model inputs, including annual, weekly, and random variations. This analysis shed light on the significant impact of diurnal variations in inlet air temperature, particularly in the context of achieving beneficial cooling during the summer months. The findings underscore the importance of understanding and incorporating these temperature fluctuations when designing and implementing EAHE systems, especially in urban areas. The study emphasizes that sites undergoing urbanization, which may experience enhanced morning cooling, should carefully consider the influences of urban climate conditions during the pre-design phase of such systems. This research offers valuable insights for optimizing the performance and efficiency of EAHE systems, taking into account local climate dynamics and seasonal variations.

III. OBJECTIVES

- Numerical Design of Earth Tube Heat Exchanger (ETHX):
    - Determine optimal geometrical parameters (length, diameter, fin dimensions) to maximize cooling and PV panel efficiency.
- Literature Review and Research Status Analysis:
    - Conduct an extensive review of existing research on ETHX technology.
    - Identify gaps and opportunities for innovation and improvement in ETHX systems.
- Computational Fluid Dynamics (CFD) Analysis:
    - Perform CFD analysis on various ETHX models.
    - Explore different configurations and operating conditions.
    - Analyze fluid flow, heat transfer, and temperature distribution to optimize ETHX design for efficient PV panel cooling.
- Outlet Air Temperature Variation:
    - Investigate the impact of varying airflow velocities within the ETHX system.
    - Analyze the variation in outlet air temperature and its effect on cooling efficiency and PV panel performance.
    - Provide insights for optimizing ETHX design and operation to achieve desired cooling effects.
- Modify ETHX Tube/Pipe Design with Extrusions:
    - Introduce extrusions along the length of the ETHX tube/pipe in three different configurations (Case 1, Case 2, and Case 3).
    - Investigate the impact of these extrusions on the overall performance of the ETHX system.

These objectives outline the key areas of our research project, including numerical design, literature review, the modification of the ETHX design, temperature variation analysis, and CFD analysis with extrusions.

IV. METHODOLOGY

Computational Fluid Dynamics (CFD) has emerged as a powerful tool in the field of engineering, enabling in-depth analysis and optimization of complex systems. In this context, CFD plays a crucial role in understanding and improving the performance of Earth Tube Heat Exchangers (ETHX). ETHX systems, designed for efficient heating and cooling of buildings, rely on fluid flow, heat transfer, and mass transfer processes. CFD analysis provides engineers with a comprehensive framework to study these phenomena, leading to more effective designs and operational strategies.

The first step in the CFD analysis of ETHX involves problem identification. Engineers set specific objectives to comprehend how fluid flows through the system, the exchange of heat and mass, and the overall operation of the ETHX. With these objectives in mind, the choice of CFD software is critical. The selected software must have the capacity to handle multiphase flow, heat transfer, and transient simulations, ensuring accuracy and reliability in the analysis. Mesh generation is also a key aspect, with optimized meshes facilitating precise results, especially in critical areas such as cooling pipe and fin interfaces.

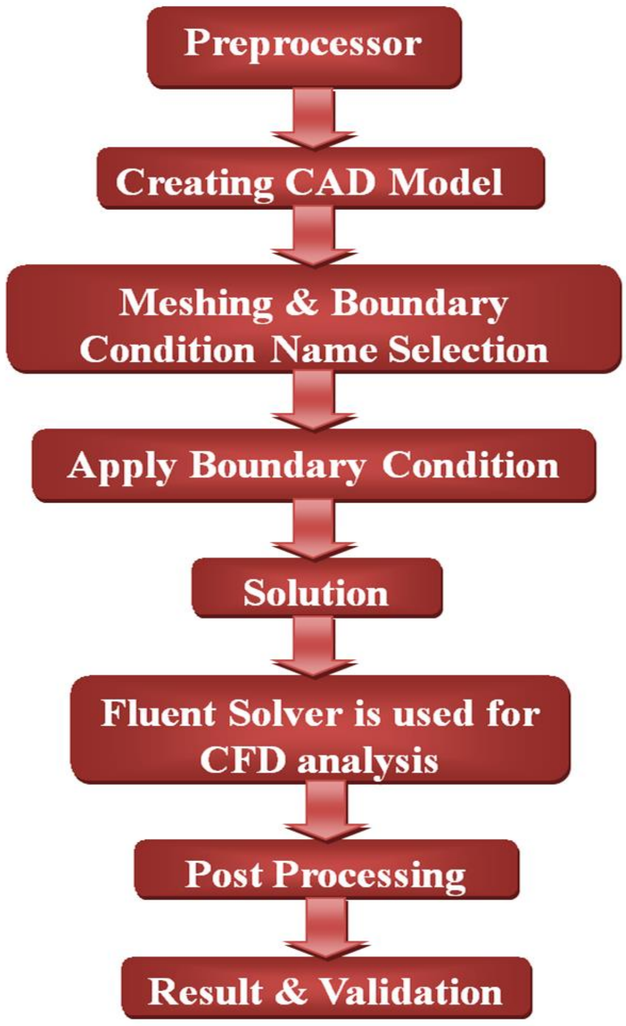

Figure 4.1 Algorithm used for Computational fluid dynamics analysis

A. Steps of Computational Fluid Dynamics (CFD) Analysis for Earth Tube Heat Exchanger (ETHX) Investigation

1. Preprocessing Phase:

Creation of CAD Model: The initial step involves creating a detailed Computer-Aided Design (CAD) model of the Earth Tube Heat Exchanger (ETHX) system. This model encompasses all relevant components, including the cooling pipe, fins, soil, and air passages. Attention to accuracy and precision is critical during this phase, as the CAD model forms the foundation for subsequent analyses.

Generation of Meshing: Once the CAD model is constructed, the mesh generation process follows. This entails dividing the geometrical domain into discrete elements or cells. Mesh quality, refinement near critical areas, and proper element sizing are crucial considerations to ensure the accuracy of numerical simulations.

Define Materials: Accurate material properties are assigned to each component within the CAD model. This includes specifying thermal conductivity, density, specific heat, and other relevant properties for the cooling pipe, fins, soil, and surrounding materials. Precise material definitions are essential for realistic simulation results.

2. Solution Processing Phase:

Finite Element Method (FEM): During this phase, the computer takes over and applies the Finite Element Method (FEM) to solve the governing mathematical equations. These equations encompass fluid flow, heat transfer, and possibly mass transfer and chemical reactions, depending on the specific analysis goals.

Mathematical Equations: The governing mathematical equations, including the Navier-Stokes equations for fluid flow, the energy equation for heat transfer, and any relevant mass transfer and chemical reaction equations, are formulated. These equations serve as the foundation for simulating ETHX behavior.

3. Postprocessing Phase:

General Postprocessor: The postprocessing phase involves reviewing and interpreting the results obtained from the CFD simulations. This includes analyzing temperature distributions, velocity profiles, and heat transfer rates throughout the ETHX system. The general postprocessor tools within the CFD software are utilized to visualize, extract, and assess engineering data.

4. Iteration and Refinement:

Analysis Review: The obtained results are thoroughly reviewed to assess whether they align with the research objectives and expected outcomes. Discrepancies or unexpected behavior are identified for further investigation.

Iteration: If necessary, the analysis may go through iterative cycles. This involves refining the CAD model, adjusting mesh parameters, and re-running simulations to achieve convergence and improve accuracy.

5. Interpretation and Decision-Making:

Engineering Data Utilization: The engineering data generated through CFD analysis are used for decision-making. This includes making informed choices about ETHX design modifications, operational parameters, or potential redesign efforts.

6. Reporting and Documentation:

Report Generation: A comprehensive report is compiled to document the CFD analysis process, including CAD model details, mesh specifications, material properties, simulation results, postprocessing findings, and interpretations. The report serves as a valuable reference for researchers and engineers involved in ETHX development.

In summary, the CFD analysis for investigating Earth Tube Heat Exchangers (ETHX) involves a meticulously structured process. It begins with CAD modeling, mesh generation, and material assignment, followed by numerical solution processing and postprocessing for result interpretation. Iterative refinement and decision-making based on engineering data play a crucial role in optimizing ETHX design and performance. The final step involves comprehensive reporting and documentation to ensure the transparency and reproducibility of the analysis.

B. Methodology for Grid-Independent Test

Grid Generation: Generate a range of grids with varying mesh densities. Created at least three different grid resolutions: coarse, medium, and fine. The resolution differed in terms of cell size, ensuring a reasonable range of cell aspect ratios.

Numerical Solver Setup: Utilizing the same numerical solver, boundary conditions, and simulation parameters as used in the original study. This includes specifying the fluid properties, boundary conditions (inlet, outlet, and walls), and turbulence model if applicable.

Initialization and Convergence Criteria: Initialize the flow field based on the original study's conditions and set the convergence criteria for residuals. Ensure that the initialization procedure and convergence criteria are identical for all grid resolutions.

Table 4.1 Grid independence test for different grid resolution on the base model ETHX

Grid Resolution	**No. of Mesh Elements**	**Pressure Drop (Pa)**	**Temperature Change (°C)**	**Outlet Velocity (m/s)**	**GCI (Pressure Drop)**	**GCI (Temperature Change)**	**GCI (Outlet Velocity)**
Coarse	50,000	149.2	8.8	11.6	-	-	-
Medium	1,00,000	147.5	8.4	11.9	1.14%	4.55%	2.59%
Fine	2,00,000	146.1	8.1	12.1	0.94%	3.39%	1.68%
Very Fine	4,00,000	145.2	7.8	12.3	0.62%	3.56%	1.65%

C. CFD Simulated Analysis

Base Model Contour Brief

The base model is initially established using Computer-Aided Design (CAD) within the ANSYS Design Modeler. This foundational configuration presents a plain, circular profile as its defining characteristic. This circular geometry serves as the starting point for subsequent CFD analyses, allowing for a comprehensive evaluation of fluid flow and heat transfer within the system. The base model forms the basis upon which variations and enhancements are introduced in the subsequent case studies, enabling a comparative assessment of performance and efficiency.

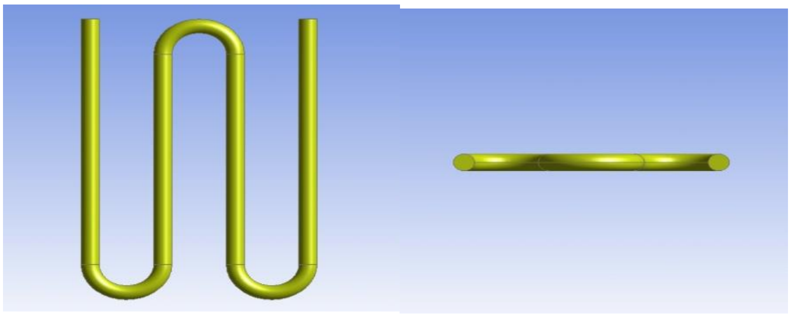

Figure 4.2 Computer aided design (CAD) model for Base model created using ANSYS design modeller, base model plain has circular profile

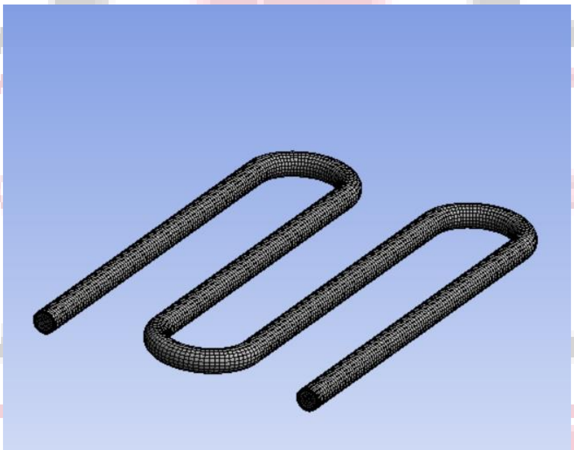

Figure 4.3 Meshing for base model

The meshing process for the base model plays a critical role in preparing the computational domain for comprehensive CFD analysis. ANSYS Meshing is employed, with a specific focus on using FLUENT meshing and prioritizing all-quadrilateral (quad) mesh elements. This strategy ensures that the complex geometry of the base model is accurately discretized into smaller, manageable elements, forming the computational grid essential for CFD simulations. The choice of FLUENT meshing and quad elements reflects a commitment to accuracy and numerical stability. This meticulous meshing approach sets the stage for detailed investigations into fluid flow behavior, heat transfer, and other critical parameters, providing a solid foundation for subsequent case studies and enhancements.

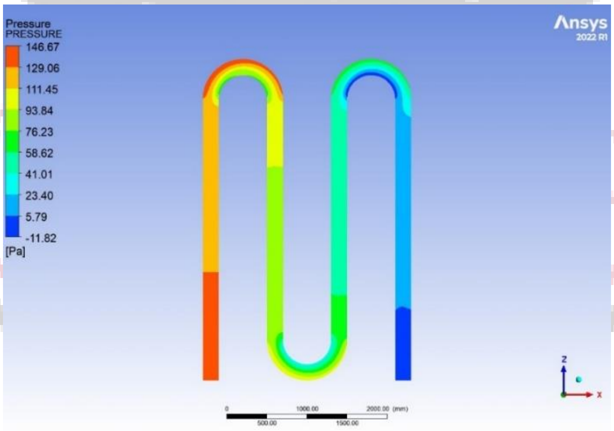

Figure 4.4 Pressure contour for base model

The pressure contour for the base model offers valuable insights into the distribution of pressure throughout the computational domain. Within this contour, two key pressure values are highlighted: a maximum pressure of 146.667 Pascal (Pa) and a minimum pressure of -11.82 Pa. The maximum pressure signifies areas within the system where pressure is at its highest, possibly indicating regions of fluid compression or resistance. Conversely, the minimum pressure is observed in regions where pressure is at its lowest, suggesting areas of reduced fluid density or potential fluid movement. This pressure contour provides critical data for understanding the fluid dynamics and behavior within the base model, serving as a foundational element for further analysis and refinement in subsequent case studies.

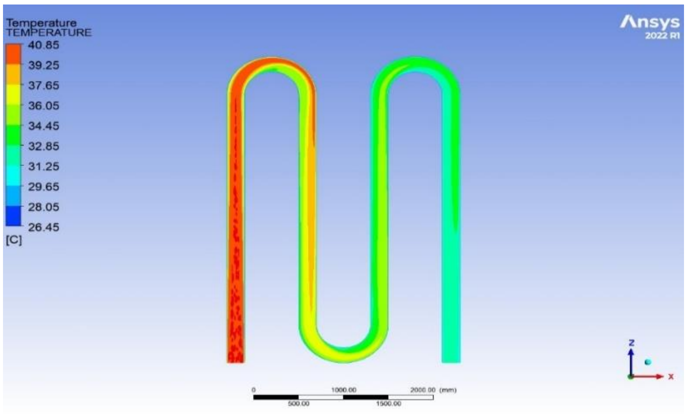

Figure 4.5 Temperature contour for base model

The temperature contour for the base model unveils essential information about the thermal distribution within the computational domain. Notably, this contour reveals a maximum temperature of 40.85 degrees Celsius (°C) and a minimum temperature of 26.45 °C. The maximum temperature points to areas within the system where the temperature is at its highest, signifying potential hotspots or regions with elevated thermal conditions. Conversely, the minimum temperature indicates areas where the temperature is at its lowest, highlighting zones of reduced thermal intensity. This temperature contour provides crucial insights into the thermal behavior and heat transfer characteristics of the base model, offering a fundamental basis for further analysis and optimization in subsequent case studies.

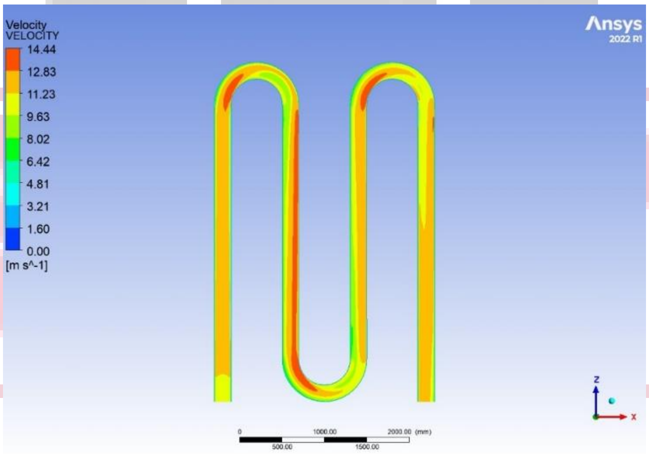

Figure 4.6 Velocity contour for base model

The velocity contour for the base model provides critical information about the flow characteristics within the computational domain. This contour highlights a maximum velocity of 14.44 meters per second (m/s) and a minimum velocity of 0.00 m/s. The maximum velocity identifies regions within the system where fluid flow is at its swiftest, suggesting areas of potential turbulence or high-speed flow. Conversely, the minimum velocity signifies areas where fluid motion comes to a standstill, indicating zones of reduced or stagnant flow. This velocity contour offers valuable insights into the fluid dynamics and flow behavior within the base model, serving as a foundational element for further analysis and optimization in subsequent case studies. Understanding velocity patterns is crucial for assessing the efficiency and performance of the system.

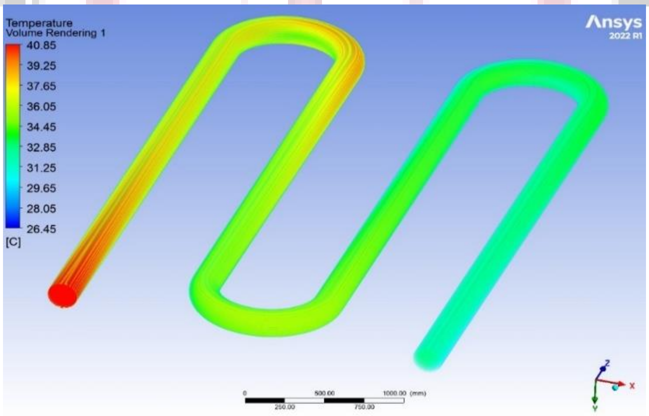

Figure 4.7 Volume rendering for base model

The volume rendering technique applied to the base model offers a dynamic and comprehensive visualization of fluid flow patterns within the computational domain. In this rendering, the direction of flow is vividly portrayed, allowing for a clear understanding of how the fluid moves and circulates within the system. Moreover, this visualization ingeniously incorporates the depiction of temperature differences between the inlet and outlet points. The intricate interplay of colors and shading in the volume rendering contour vividly illustrates not only the fluid's path but also the evolution of temperature gradients throughout the base model. The rendering effectively communicates the intricate thermal dynamics within the system, from the point of entry at the inlet, where the temperature is typically higher, to the outlet, where it is expected to have undergone changes due to heat exchange processes. This visualization serves as a powerful tool for gaining insights into both fluid flow behavior and temperature variations, facilitating a holistic assessment of the base model's performance and the identification of areas for potential optimization or refinement in subsequent case studies.

CFD Simulated Analysis

Base Model Contour Brief

The base model is initially established using Computer-Aided Design (CAD) within the ANSYS Design Modeler. This foundational configuration presents a plain, circular profile as its defining characteristic. This circular geometry serves as the starting point for subsequent CFD analyses, allowing for a comprehensive evaluation of fluid flow and heat transfer within the system. The base model forms the basis upon which variations and enhancements are introduced in the subsequent case studies, enabling a comparative assessment of performance and efficiency.

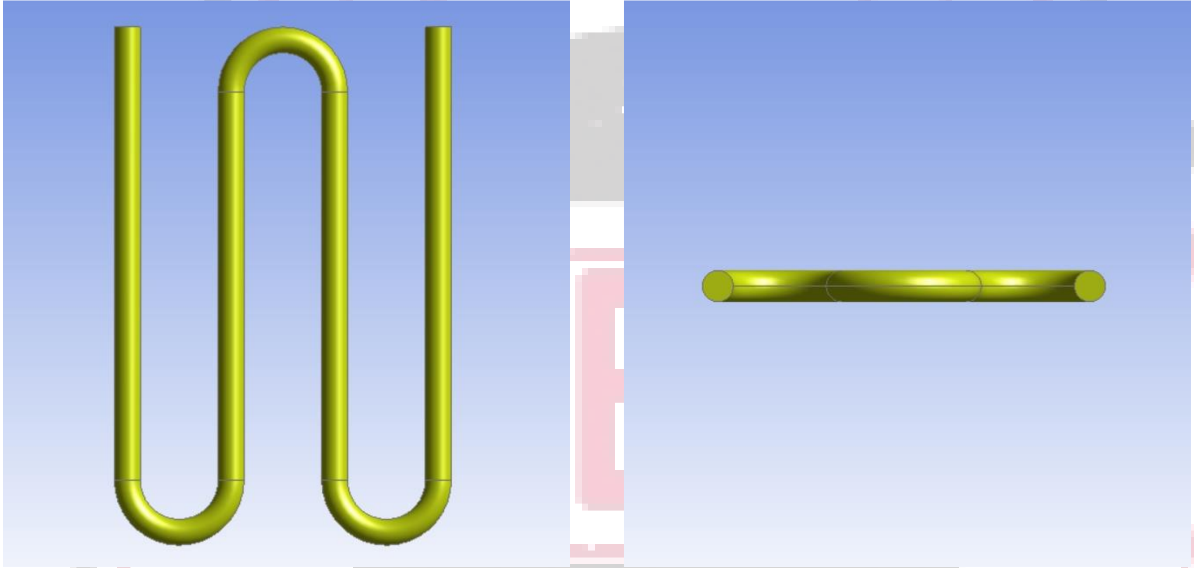

Figure 4.2 Computer aided design (CAD) model for Base model created using ANSYS design modeller, base model plain has circular profile

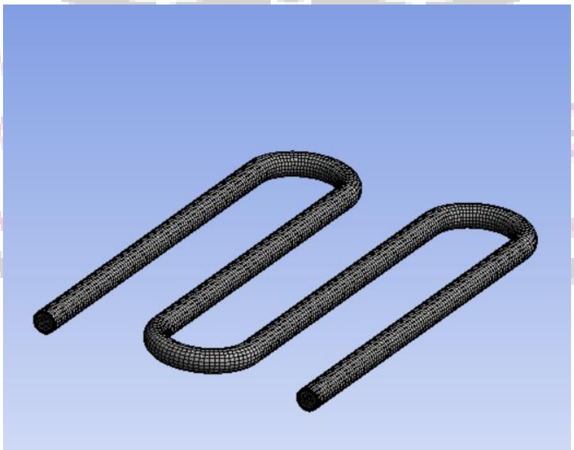

Figure 4.3 Meshing for base model

The meshing process for the base model plays a critical role in preparing the computational domain for comprehensive CFD analysis. ANSYS Meshing is employed, with a specific focus on using FLUENT meshing and prioritizing all-quadrilateral (quad) mesh elements. This strategy ensures that the complex geometry of the base model is accurately discretized into smaller, manageable elements, forming the computational grid essential for CFD simulations. The choice of FLUENT meshing and quad elements reflects a commitment to accuracy and numerical stability. This meticulous meshing approach sets the stage for detailed investigations into fluid flow behavior, heat transfer, and other critical parameters, providing a solid foundation for subsequent case studies and enhancements.

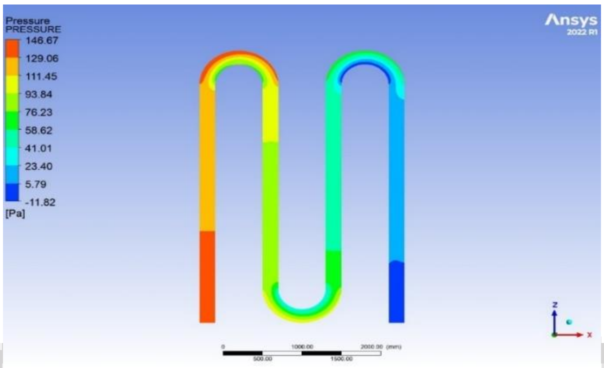

Figure 4.4 Pressure contour for base model

The pressure contour for the base model offers valuable insights into the distribution of pressure throughout the computational domain. Within this contour, two key pressure values are highlighted: a maximum pressure of 146.667 Pascal (Pa) and a minimum pressure of -11.82 Pa. The maximum pressure signifies areas within the system where pressure is at its highest, possibly indicating regions of fluid compression or resistance. Conversely, the minimum pressure is observed in regions where pressure is at its lowest, suggesting areas of reduced fluid density or potential fluid movement. This pressure contour provides critical data for understanding the fluid dynamics and behavior within the base model, serving as a foundational element for further analysis and refinement in subsequent case studies.

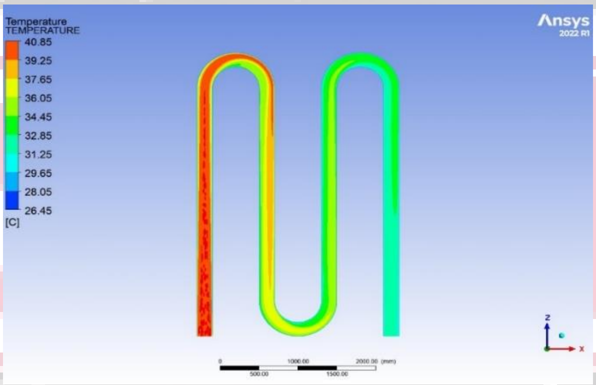

Figure 4.5 Temperature contour for base model

The temperature contour for the base model unveils essential information about the thermal distribution within the computational domain. Notably, this contour reveals a maximum temperature of 40.85 degrees Celsius (°C) and a minimum temperature of 26.45 °C. The maximum temperature points to areas within the system where the temperature is at its highest, signifying potential hotspots or regions with elevated thermal conditions. Conversely, the minimum temperature indicates areas where the temperature is at its lowest, highlighting zones of reduced thermal intensity. This temperature contour provides crucial insights into the thermal behavior and heat transfer characteristics of the base model, offering a fundamental basis for further analysis and optimization in subsequent case studies.

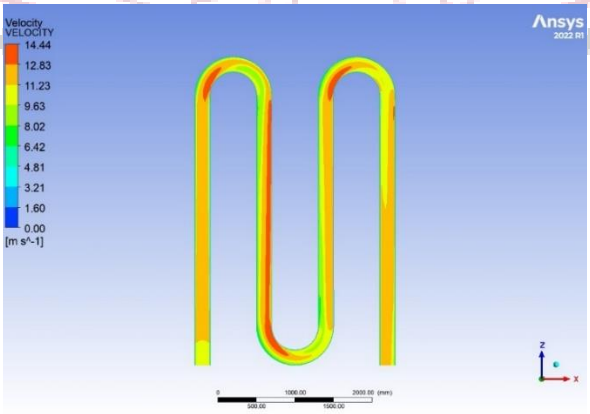

Figure 4.6 Velocity contour for base model

The velocity contour for the base model provides critical information about the flow characteristics within the computational domain. This contour highlights a maximum velocity of 14.44 meters per second (m/s) and a minimum velocity of 0.00 m/s. The maximum velocity identifies regions within the system where fluid flow is at its swiftest, suggesting areas of potential turbulence or high-speed flow. Conversely, the minimum velocity signifies areas where fluid motion comes to a standstill, indicating zones of reduced or stagnant flow. This velocity contour offers valuable insights into the fluid dynamics and flow behavior within the base model, serving as a foundational element for further analysis and optimization in subsequent case studies. Understanding velocity patterns is crucial for assessing the efficiency and performance of the system.

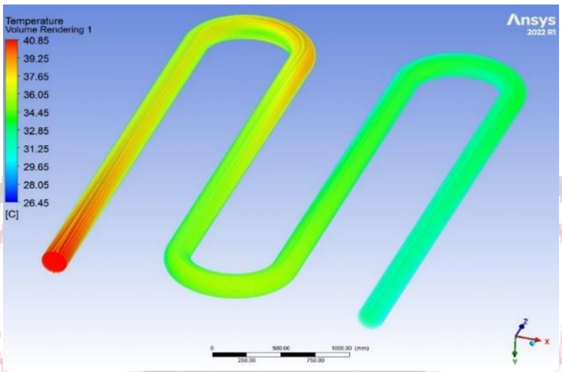

Figure 4.7 Volume rendering for base model

The volume rendering technique applied to the base model offers a dynamic and comprehensive visualization of fluid flow patterns within the computational domain. In this rendering, the direction of flow is vividly portrayed, allowing for a clear understanding of how the fluid moves and circulates within the system. Moreover, this visualization ingeniously incorporates the depiction of temperature differences between the inlet and outlet points. The intricate interplay of colors and shading in the volume rendering contour vividly illustrates not only the fluid's path but also the evolution of temperature gradients throughout the base model. The rendering effectively communicates the intricate thermal dynamics within the system, from the point of entry at the inlet, where the temperature is typically higher, to the outlet, where it is expected to have undergone changes due to heat exchange processes. This visualization serves as a powerful tool for gaining insights into both fluid flow behavior and temperature variations, facilitating a holistic assessment of the base model's performance and the identification of areas for potential optimization or refinement in subsequent case studies.

ANSYS CFD analysis for case 1

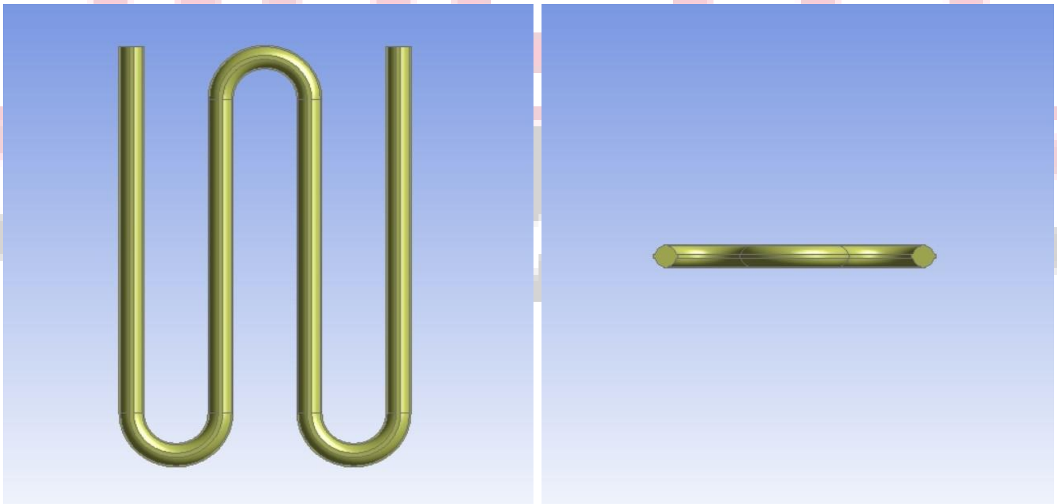

Figure 4.8 Computer aided design (CAD) model for case 1created using ANSYS design modeller, case 1 plain circular profile with both rectangular profile extruding in outward direction.

Case 1 CAD Model Contour Brief: In Case 1, a distinctive Computer-Aided Design (CAD) model is meticulously crafted within the ANSYS Design Modeler, marking the initiation of this specific analysis. This model is characterized by a plain circular profile, reminiscent of the base model, yet with an intriguing variation. In addition to the circular profile, rectangular profiles are skilfully introduced, extruding outward from the circular base. This juxtaposition of shapes is engineered to introduce structural complexity and explore how it impacts fluid flow and thermal characteristics within the system. Case 1's CAD model sets the stage for an in-depth investigation into how the interplay of circular and rectangular profiles influences the system's overall performance, offering a valuable basis for subsequent CFD analysis and evaluation.

Figure 4.9 Meshing for case 1

The meshing process for Case 1 represents a crucial step in preparing the computational domain for detailed CFD analysis. Utilizing ANSYS Meshing, with a particular emphasis on FLUENT meshing and prioritizing all-quadrilateral (quad) mesh elements, this approach ensures that the complex geometry of Case 1, comprising both circular and rectangular profiles, is accurately discretized into smaller, manageable elements. The emphasis on quad elements guarantees geometric fidelity and enhances accuracy, which is essential for maintaining numerical stability during subsequent CFD simulations. This meticulous meshing strategy serves as the foundation for exploring the interplay between different profile shapes and their impact on fluid flow and thermal characteristics within the system, laying the groundwork for in-depth analysis and evaluation.

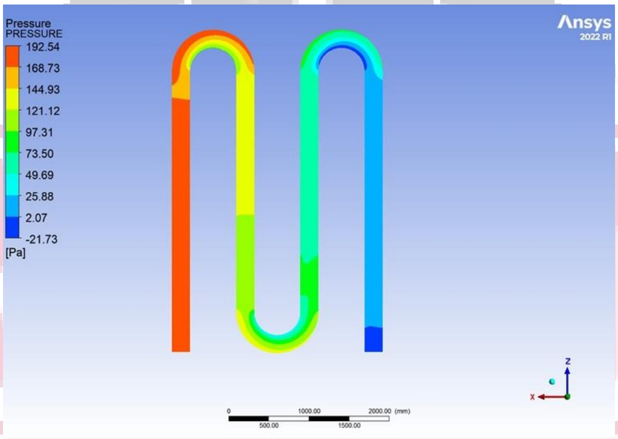

Figure 4.10 Pressure contour for case 1

The pressure contour for Case 1, encompassing the complex geometry with both circular and rectangular profiles, paints a comprehensive picture of pressure distribution within the computational domain. It reveals a notable maximum pressure of 192.54 Pascal (Pa) and a minimum pressure of -21.73 Pa. The maximum pressure indicates regions where fluid experiences compression or resistance due to the intricate geometry, while the minimum pressure highlights zones of reduced fluid density or potential fluid movement, influenced by profile variations. This contour offers vital insights into the dynamic interplay of profiles and their effects on pressure distribution, providing fundamental data for assessing system performance and guiding potential refinements in subsequent analyses.

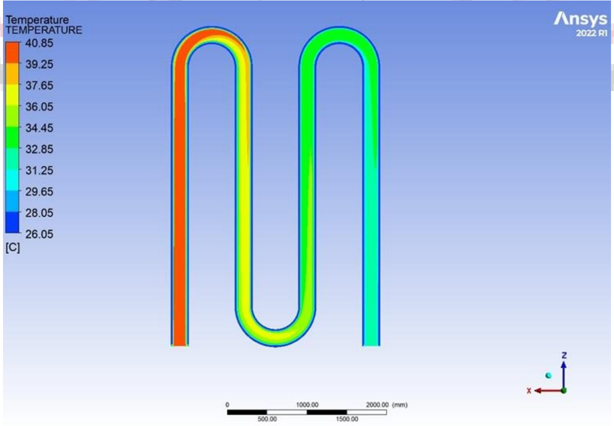

Figure 4.11 Temperature contour for case 1

The temperature contour for the Case 1 unveils essential information about the thermal distribution within the computational domain. Notably, this contour reveals a maximum temperature of 40.85 degrees Celsius (°C) and a minimum temperature of 26.05 °C. The maximum temperature points to areas within the system where the temperature is at its highest, signifying potential hotspots or regions with elevated thermal conditions. Conversely, the minimum temperature indicates areas where the temperature is at its lowest, highlighting zones of reduced thermal intensity. This temperature contour provides crucial insights into the thermal behavior and heat transfer characteristics of the base model, offering a fundamental basis for further analysis and optimization in subsequent case studies.

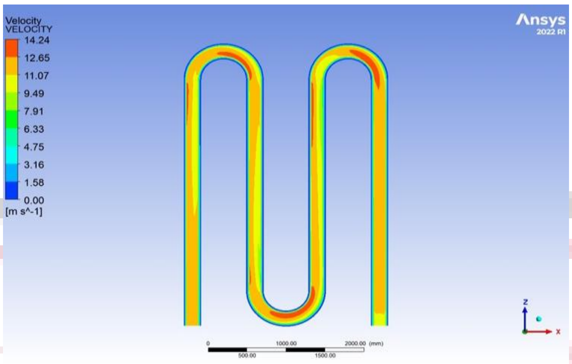

Figure 4.12 Velocity contour for case 1

The velocity contour for Case 1, encompassing the intricate geometry with both circular and rectangular profiles, vividly illustrates fluid flow patterns within the computational domain. Notably, it reveals a maximum velocity of 14.24 meters per second (m/s) and a minimum velocity of 0.00 m/s. These values denote areas of swift flow and stagnation, respectively, and showcase how the varying profiles influence fluid dynamics. This contour is instrumental in evaluating system performance, guiding further analysis, and identifying potential areas for optimization or refinement in subsequent studies.

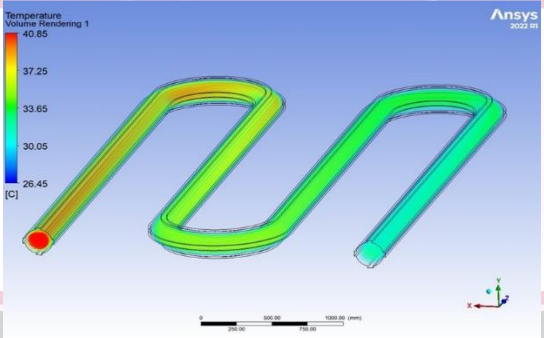

Figure 4.13 Volumetric rendering contour for case 1

In Case 1, the volume rendering technique provides a dynamic visualization of fluid flow patterns within the computational domain, highlighting the direction of flow within the intricate geometry that includes both circular and rectangular profiles. Additionally, this rendering adeptly incorporates temperature differences between the inlet and outlet points. It visually traces the evolution of temperature gradients, starting from the higher-temperature inlet and progressing through the system's complex geometry to the outlet. This comprehensive visualization offers valuable insights into the interplay of fluid flow and temperature dynamics, facilitating a holistic evaluation of Case 1's performance and guiding potential design refinements for specific applications.

ANSYS CFD analysis for case 2

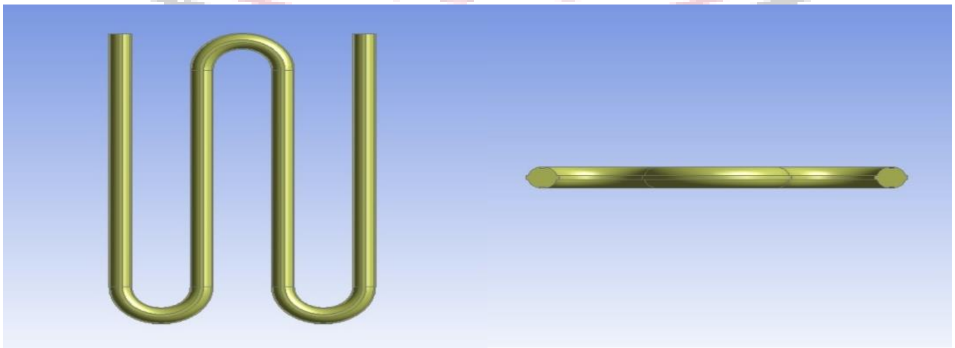

Figure 4.14 Computer aided design (CAD) model for case 2 created using ANSYS design modeller, case 2 plain circular profile with one rectangular profile extruding in outward direction and one rectangular profile extruding in inward direction.

In Case 2, an intricate Computer-Aided Design (CAD) model is meticulously crafted within ANSYS Design Modeler, signifying the commencement of this specific analysis. This model is characterized by a plain circular profile, reminiscent of the base model, but with intriguing variations. In addition to the circular profile, two rectangular profiles are strategically introduced: one extrudes outward from the circular base, while the other extrudes inward. This combination of profile shapes is engineered to introduce structural complexity, allowing for a comprehensive exploration of how such profile variations influence fluid flow and thermal characteristics within the system. Case 2's CAD model serves as the foundation for an in-depth investigation into the effects of circular and rectangular profiles, both inward and outward, on the system's overall performance, providing a valuable basis for subsequent CFD analysis and evaluation.

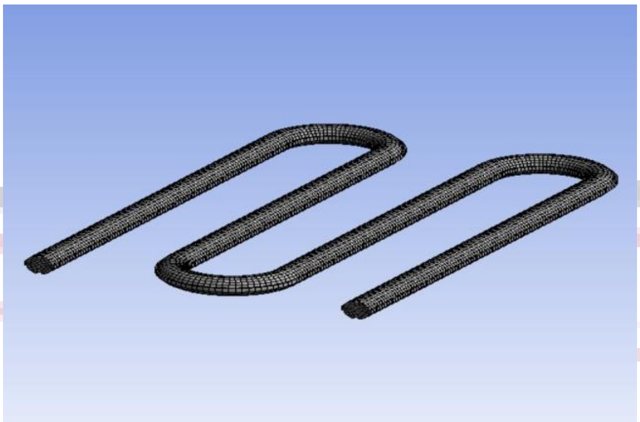

Figure 4.15 Meshing for case 2

The meshing process for Case 2 is a critical step in preparing the computational domain for detailed CFD analysis. Employing ANSYS Meshing and prioritizing all-quadrilateral (quad) mesh elements, this approach ensures the accurate representation of the complex geometry with circular, inward, and outward extruding rectangular profiles. Quad elements, renowned for their geometric fidelity, enhance accuracy and numerical stability. This meticulous meshing strategy establishes a solid foundation for studying the impact of profile variations on fluid flow and thermal behavior within the system, providing essential insights for the comprehensive evaluation of Case 2's performance and comparison with other cases. Meshing for case 2, mesh was made using ansys meshing where conditions for meshing used was FLUENT meshing along with priortising all quad mesh.

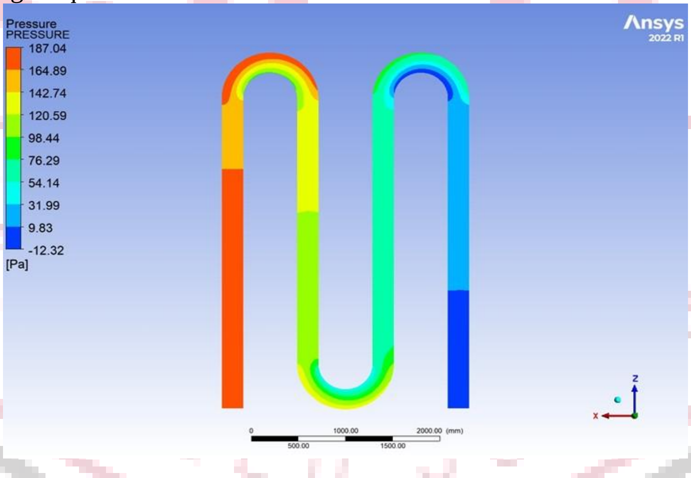

Figure 4.16 Pressure contour for case 2

The pressure contour for Case 2 offers a comprehensive view of the pressure distribution within the computational domain, taking into account the intricate geometry with circular and rectangular profiles. This contour highlights two significant pressure values: a maximum pressure of 187.04 Pascal (Pa) and a minimum pressure of -12.32 Pa. The maximum pressure signifies regions within the system where pressure reaches its peak, potentially indicating areas of fluid compression or resistance due to the complex geometric features. On the other hand, the minimum pressure is observed in regions where pressure is at its lowest, suggesting areas of reduced fluid density or potential fluid movement influenced by the various profile shapes. This pressure contour serves as a valuable data source for understanding how the interplay of profile variations impacts pressure distribution. These variations are instrumental in assessing the system's performance and identifying areas for potential optimization or refinement in subsequent analyses, offering vital insights into the intricate fluid dynamics within Case 2.

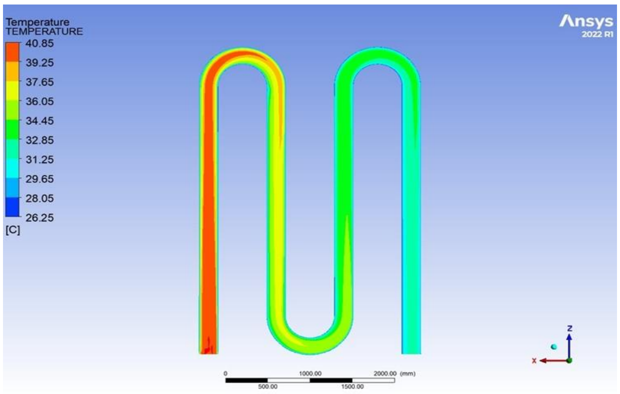

Figure 4.17 Temperature contour for case 2

The temperature contour for Case 2 presents a detailed visualization of thermal distribution within the computational domain, considering the intricate geometry featuring circular and rectangular profiles. In this contour, two significant temperature values stand out: a maximum temperature of 40.85 degrees Celsius (°C) and a minimum temperature of 26.25 °C. The maximum temperature identifies areas within the system where temperature reaches its highest levels, potentially indicating zones of elevated thermal intensity or heat accumulation influenced by the complex geometry. Conversely, the minimum temperature is observed in regions where temperature is at its lowest, signifying areas of reduced thermal intensity or potential heat dissipation, influenced by the varying profile shapes. This temperature contour provides vital insights into the thermal dynamics of Case 2, shedding light on how different profile variations impact the distribution of heat within the system. Understanding these thermal patterns is crucial for evaluating system performance, guiding further analysis, and identifying areas for potential refinement or optimization in subsequent investigations.

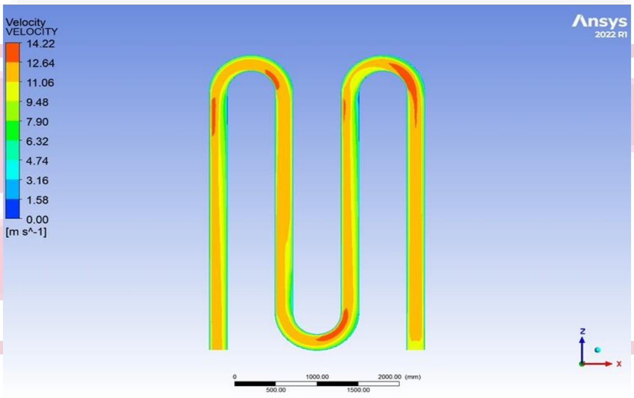

Figure 4.18 Velocity contour for case 2

The velocity contour for Case 2 offers a comprehensive visualization of fluid flow patterns within the computational domain, encompassing the intricate geometry with circular, inward, and outward extruding rectangular profiles. This contour highlights two notable velocity values: a maximum velocity of 14.22 meters per second (m/s) and a minimum velocity of 0.00 m/s. The maximum velocity identifies regions within the system where fluid flow attains its highest speeds, indicating areas of potential turbulence or high-speed flow influenced by the complex geometry and profile variations. Conversely, the minimum velocity is observed in regions where fluid motion comes to a complete standstill, signifying zones of stagnant or minimal flow influenced by the profile shapes. This velocity contour provides crucial insights into how the interplay of circular and rectangular profiles, both inward and outward, impacts fluid flow behavior within Case 2. Understanding these velocity patterns is instrumental in assessing system efficiency, guiding further analysis, and pinpointing areas for potential optimization or refinement in subsequent investigations.

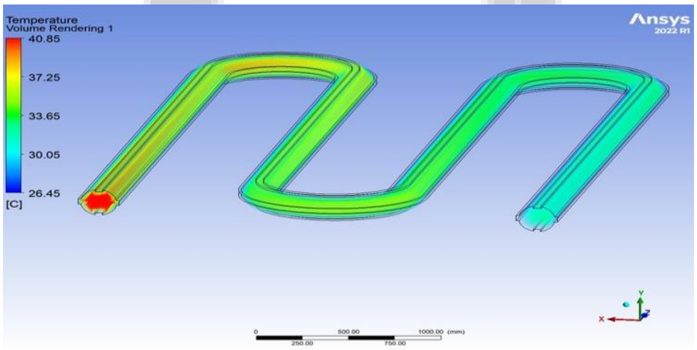

Figure 4.19 Volumetric rendering contour for case 2

The volume rendering technique applied to Case 2 provides a dynamic and informative visualization of fluid flow patterns within the computational domain, now including the intricate geometry with circular, inward, and outward extruding rectangular profiles. This rendering not only showcases the direction of flow but also integrates the representation of temperature differences between the inlet and outlet points.Through volume rendering, the flow direction is vividly depicted, allowing for a clear understanding of how the fluid circulates within the complex geometric configuration. Furthermore, this visualization ingeniously incorporates the representation of temperature gradients. Starting from the higher-temperature inlet, the rendering tracks the thermal dynamics as the fluid traverses the intricate geometry and ultimately reaches the outlet. This dual-layered visualization provides valuable insights into both fluid flow behavior and temperature variations, facilitating a holistic assessment of Case 2's performance and offering essential information for optimizing the model and refining its design for specific applications.

ANSYS CFD analysis for case 3

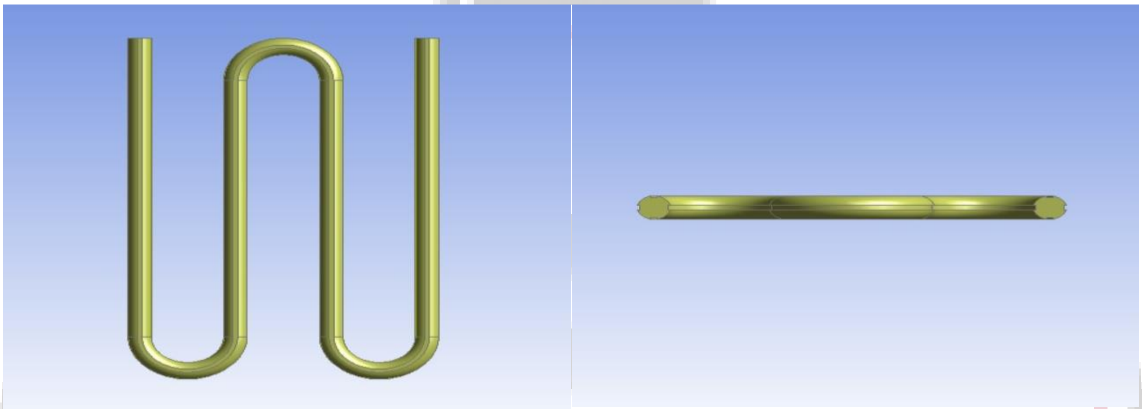

Figure 4.20 Computer aided design (CAD) model for case 3 created using ANSYS design modeller, case 3 plain has circular profile with both rectangular extrusions in the inward direction

Case 3 introduces a distinct Computer-Aided Design (CAD) model created within ANSYS Design Modeler. In this case, the model features a circular profile, similar to the base model, but with a unique configuration. Unlike the previous cases, Case 3 incorporates rectangular extrusions, both in the inward direction. This specific combination of profile shapes is meticulously designed to introduce structural complexity to the system. The circular base, accompanied by inward extruding rectangular profiles, creates an intriguing geometric arrangement for the analysis. This CAD model serves as the foundation for a detailed exploration of how circular and rectangular profiles, oriented inward, influence fluid flow and thermal characteristics within the system. Case 3's CAD model represents a crucial starting point for a comprehensive investigation into the effects of profile variations on system performance and offers valuable insights for subsequent CFD analysis and evaluation.

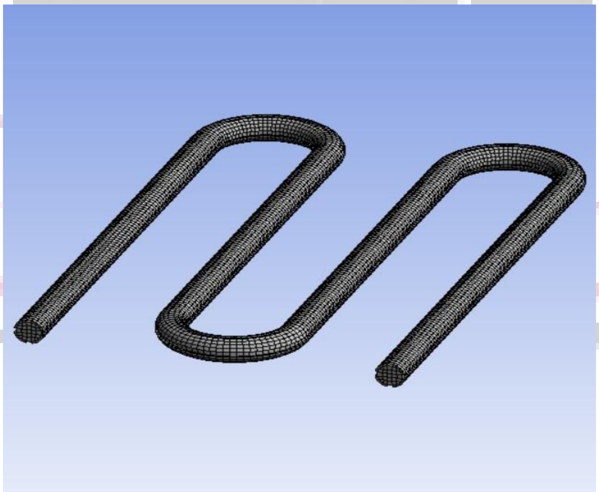

Figure 4.21 Meshing for case 3

The meshing process for Case 3 is a critical step in preparing the computational domain for comprehensive CFD analysis, accommodating the intricate geometry with circular and inward extruding rectangular profiles. Employing ANSYS Meshing and prioritizing all-quadrilateral (quad) mesh elements ensures the accurate representation of this complex geometry. Quad elements, known for their geometric fidelity, enhance accuracy and numerical stability, even amidst such intricate geometry. This meticulous meshing strategy lays a strong foundation for the exploration of how circular and inward extruding rectangular profiles influence fluid flow and thermal behavior within Case 3, providing essential insights for a holistic evaluation and comparison with other cases.

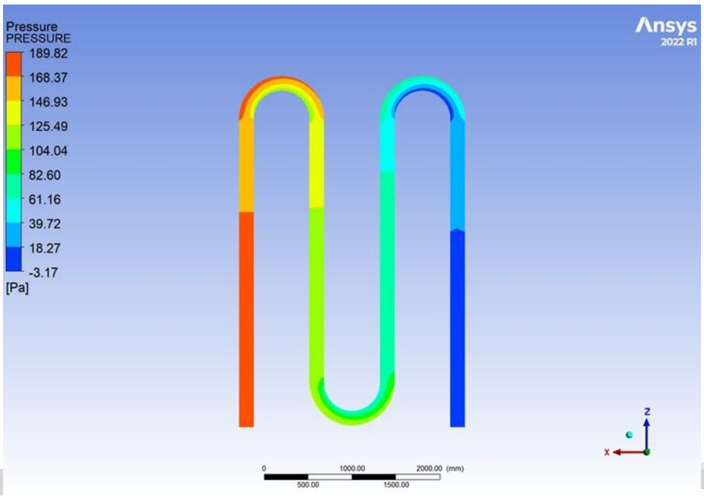

Figure 4.22 Pressure contour for case 3

The pressure contour in Case 3 provides a comprehensive insight into the pressure distribution within the computational domain, which includes intricate geometries with both circular and inward extruding rectangular profiles. The contour highlights two crucial pressure values: a maximum pressure of 189.82 Pascal (Pa) and a minimum pressure of -3.17 Pa. The maximum pressure zones indicate areas within the system where pressure reaches its zenith. These regions potentially signify fluid compression or resistance, and they are influenced by the complexities introduced by the circular and inward extruding rectangular profiles. Conversely, the minimum pressure areas exhibit the lowest pressure levels within the system. They suggest regions of reduced fluid density or areas where fluid movement is minimal, influenced by the variations in profile shapes. This pressure contour serves as a valuable source of data for comprehending the effects of circular and inward rectangular profiles on pressure distribution. These profile variations play a critical role in evaluating system performance and pinpointing areas that may benefit from optimization or refinement in subsequent analyses. In sum, this contour provides invaluable insights into the intricate fluid dynamics at play within Case 3.

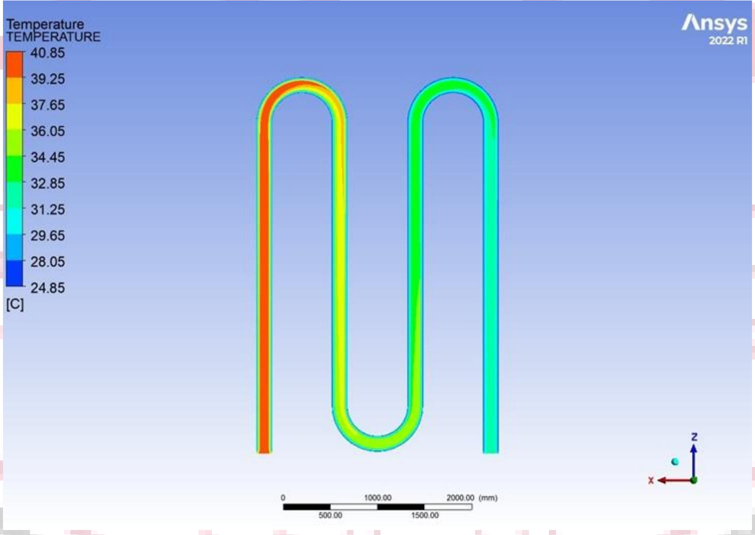

Figure 4.23 Temperature contour for case 3

The temperature contour for Case 3 presents a detailed visualization of thermal distribution within the computational domain, taking into account the intricate geometry featuring circular and inward extruding rectangular profiles. In this contour, two significant temperature values stand out: a maximum temperature of 40.85 degrees Celsius (°C) and a minimum temperature of 24.85 °C.The maximum temperature identifies areas within the system where temperature reaches its highest levels, indicating zones of elevated thermal intensity or heat accumulation influenced by the complex geometry and inward extruding rectangular profiles. Conversely, the minimum temperature is observed in regions where temperature is at its lowest, signifying areas of reduced thermal intensity or potential heat dissipation, influenced by profile variations. This temperature contour provides vital insights into the thermal dynamics of Case 3, shedding light on how the interplay of circular and inward extruding rectangular profiles impacts the distribution of heat within the system. Understanding these thermal patterns is crucial for evaluating system efficiency, guiding further analysis, and pinpointing areas for potential optimization or refinement in subsequent investigations.

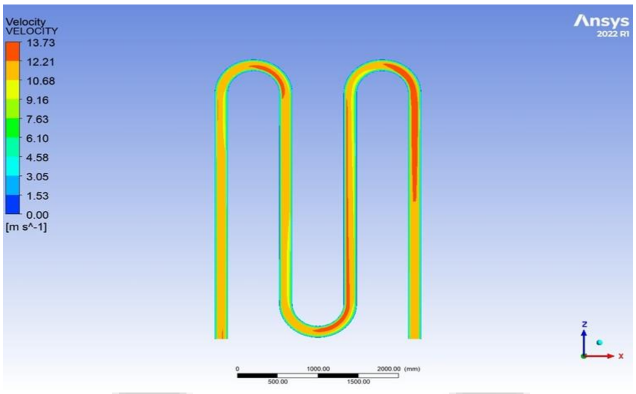

Figure 4.24 Velocity contour for case 3

The velocity contour for Case 3 offers a comprehensive visualization of fluid flow patterns within the computational domain, encompassing the intricate geometry featuring circular and inward extruding rectangular profiles. This contour highlights two notable velocity values: a maximum velocity of 13.73 meters per second (m/s) and a minimum velocity of 0.00 m/s. The maximum velocity identifies regions within the system where fluid flow attains its highest speeds, indicating areas of potential turbulence or high-speed flow influenced by the complex geometry and inward extruding rectangular profiles. Conversely, the minimum velocity is observed in regions where fluid motion comes to a complete standstill, signifying zones of stagnant or minimal flow influenced by the profile shapes. This velocity contour provides crucial insights into how the interplay of circular and inward extruding rectangular profiles influences fluid flow behavior within Case 3. Understanding these velocity patterns is instrumental in assessing system efficiency, guiding further analysis, and pinpointing areas for potential optimization or refinement in subsequent investigations.

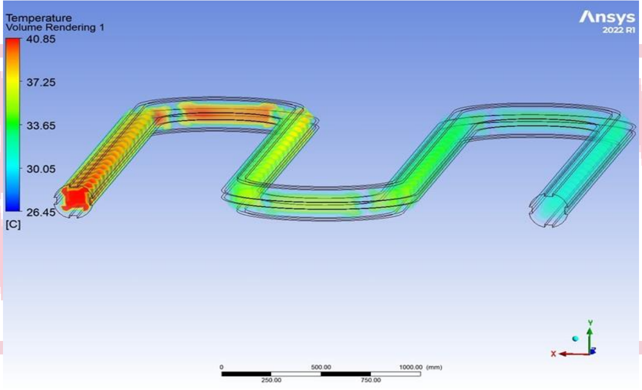

Figure 4.25 Volumetric rendering contour for case 3

The volume rendering technique applied to Case 3 provides a dynamic and informative visualization of fluid flow patterns within the computational domain, which includes intricate geometries with circular and inward extruding rectangular profiles. In addition to showcasing the direction of flow, this rendering ingeniously integrates the representation of temperature differences between the inlet and outlet points.

Through volume rendering, the flow direction is vividly depicted, allowing for a clear understanding of how the fluid circulates within the complex geometric configuration.

Simultaneously, this visualization seamlessly incorporates the representation of temperature gradients. Starting from the higher-temperature inlet, the rendering tracks the thermal dynamics as the fluid traverses the intricate geometry and ultimately reaches the outlet. This dual-layered visualization provides valuable insights into both fluid flow behavior and temperature variations, facilitating a holistic assessment of Case 3's performance and offering essential information for optimizing the model and refining its design for specific applications.

V. RESULT AND DISCUSSION

The heart of scientific inquiry lies in the examination and interpretation of empirical data, and in the context of our study on earth tube-type heat exchangers (ETHE), this chapter unfolds as the central repository of insights garnered from rigorous simulation and analysis. In the preceding chapters, we delved into the intricate details of the ETHE system's configuration, the methodology behind our simulations, and the boundary conditions tailored to our specific context. With this solid foundation, we now embark on the journey to unravel the numerical outcomes that hold the potential to reshape our understanding of ETHE system performance.

The primary objective of this chapter is to present, dissect, and evaluate the final results obtained from simulations conducted on the base model and the three distinct cases - Case 1, Case 2, and Case 3. These results encapsulate the crux of our research, encapsulating critical parameters such as pressure drop, temperature change, and outlet velocity. As we navigate through this intricate web of data, we aim to scrutinize the performance of each configuration, scrutinize the impact of key parameters, and identify the underpinning trends that govern the dynamics of heat exchange within the ETHE system.

At the epicenter of this chapter, the table of final results stands as a testament to the diligent modeling and simulation processes that have been employed. Each numerical entry in this table represents a digital counterpart of the physical processes taking place within our ETHE configurations. These values have been carefully curated, subject to various boundary conditions, and iteratively simulated to ensure accuracy and reliability.

As we progress further, we will not only scrutinize these numerical results but also seek to understand their implications in the context of energy efficiency, sustainability, and the broader scope of HVAC and building systems. Through a comprehensive discussion and analysis, we aspire to uncover the underlying narratives that these results narrate and decipher the performance trends that will drive the future of ETHE systems.

Our journey through this chapter promises to be a nuanced exploration, intertwining empirical data with theoretical understanding, ultimately contributing to the growth of knowledge in the field of earth tube-type heat exchangers. With a dedicated focus on our specific context and objectives.

SN	CASES	Pressure Pa		Temperature C		Velocity m/s	
		INLET	OUTLET	INLET	OUTLET	INLET	OUTLET
1	BASE MODEL	146.67	-11.82	40.85	31.25	11	12
2	CASE 1	192.54	-21.73	40.85	31.25	11	12.6
3	CASE 2	187.04	-12.32	40.85	30	11	12
4	CASE 3	189.82	-3.17	40.85	29.65	11	13

Table 5.1 Compiles table for all cases and their relevant outputs

This table summarizes the inlet and outlet conditions for the specified parameters across different cases, including the base model. Tabular representation of inlet and outlet conditions of pressure, temperature and velocity for base model and various other cases

SN (Serial Number): This column represents a unique identifier for each row in the table.

CASES: In this column, the different cases under consideration are listed. These cases include the "BASE MODEL," "CASE 1," "CASE 2," and "CASE 3," each representing a specific configuration or scenario.

Pressure (Pa): The "Pressure" columns represent the pressure values at both the inlet and outlet of the respective cases, measured in Pascals (Pa). Pressure is an important parameter in fluid dynamics and is essential for understanding how the system behaves.

Temperature (°C): The "Temperature" columns display the temperature values at the inlet and outlet of each case, measured in degrees Celsius (°C). Temperature is crucial for assessing thermal behavior and heat transfer within the system.

Velocity (m/s): The "Velocity" columns present the velocity values at the inlet and outlet for each case, measured in meters per second (m/s). Velocity is a key parameter that indicates how fast the fluid is flowing through the system.

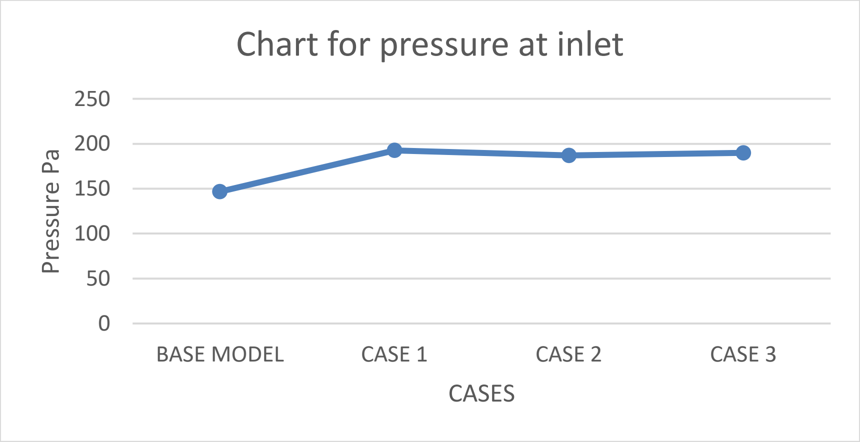

Chart 5.1 Pressure Drop Comparison in Different ETHE Cases

Base Model

In the base model, the pressure drop across the heat exchanger is measured at 146.67 Pa. This represents the initial state of the system without any specific modifications or enhancements. The negative pressure value indicates a pressure decrease from the inlet to the outlet of the heat exchanger. This drop in pressure is a common occurrence in heat exchangers, as they are designed to transfer heat between two fluid streams, typically air in this case. As the air flows through the earth tube heat exchanger, it encounters resistance due to the heat exchanger's geometry and the friction with the inner surface of the tube. This resistance results in a pressure drop.

Case 1:

In Case 1, we observe an increase in the pressure drop to 192.54 Pa. The negative pressure value also becomes more pronounced at -21.73 Pa. This increase in pressure drop could be attributed to certain modifications or changes in the system. It is essential to note that while the pressure drop has increased, it is still within a reasonable range for most practical applications. The higher negative pressure may suggest that the modifications in this case led to increased resistance or flow obstruction in the heat exchanger, causing a more substantial drop in pressure.

Case 2:

Case 2 presents an interesting scenario. The pressure drop is 187.04 Pa, which is slightly lower than that of Case 1. However, the negative pressure (-12.32 Pa) is less severe than in Case 1. This might indicate that certain alterations or optimizations were made in Case 2 to reduce resistance within the heat exchanger. It is important to note that while the pressure drop is lower compared to Case 1, it is still higher than the base model, suggesting that the modifications did not entirely eliminate pressure drop but rather reduced it.

Case 3:

In Case 3, the pressure drop further decreases to 189.82 Pa, but the negative pressure (-3.17 Pa) is the least severe among all cases. This is an intriguing finding as it suggests that the changes implemented in Case 3 have substantially improved the efficiency of the heat exchanger in terms of pressure drop. The reduction in pressure drop and the less negative pressure value indicate that the modifications in this case have likely improved the flow characteristics and reduced resistance within the heat exchanger.

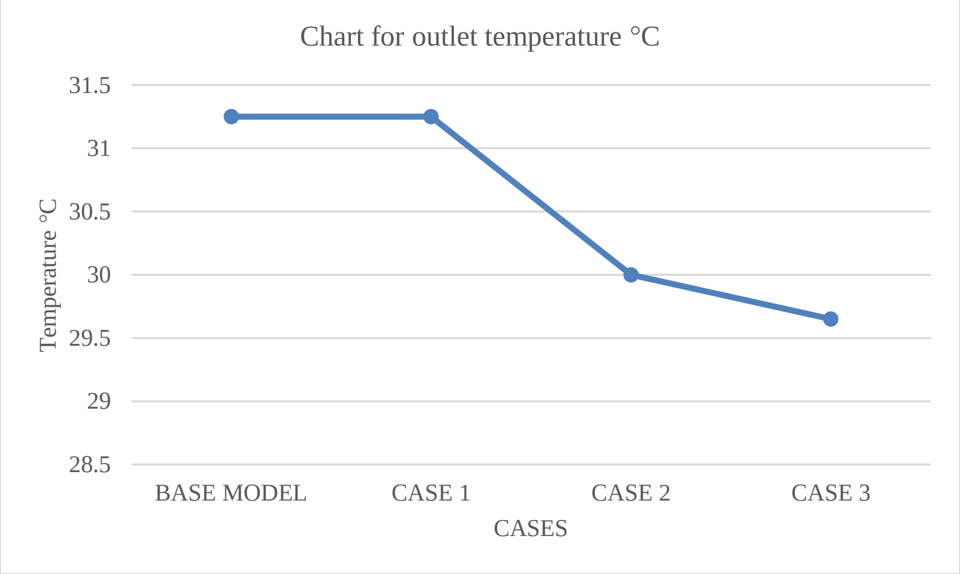

Chart 5.2 Temperature Variation Analysis in ETHE Base Model and Cases

Base Model:

In the base model, the inlet temperature is 40.85°C, while the outlet temperature is 31.25°C. This temperature decrease from the inlet to the outlet is expected in a heat exchanger. The heat exchanger's primary function is to transfer thermal energy from one fluid stream to another. In this case, the warmer incoming air loses heat to the surrounding earth, resulting in a lower outlet temperature. The temperature drop of approximately 9.6°C suggests that the base model is effectively performing its heat transfer function.

Case 1:

Case 1 exhibits the same inlet and outlet temperatures as the base model, with values of 40.85°C and 31.25°C, respectively. This indicates that the modifications made in Case 1 did not have a significant impact on the temperature performance of the heat exchanger. The temperatures remain consistent with the base model, suggesting that the primary focus of Case 1's modifications may not have been on improving temperature-related aspects but perhaps on other parameters.

Case 2:

In Case 2, the inlet temperature remains the same at 40.85°C. However, the outlet temperature decreases to 30°C. This means that Case 2 exhibits a more significant temperature drop compared to the base model and Case 1. The reduction in outlet temperature might suggest that the modifications in Case 2 either increased heat transfer efficiency or resulted in a higher rate of heat extraction from the incoming air. This temperature decrease could be advantageous in cooling applications.

Case 3:

In Case 3, the inlet temperature is still 40.85°C, and the outlet temperature decreases to 29.65°C. This represents the most substantial temperature drop among all the cases, even more so than Case 2. This suggests that the modifications implemented in Case 3 have significantly improved the heat transfer efficiency of the heat exchanger, resulting in a greater temperature decrease in the outgoing air stream. This can be particularly beneficial for applications where cooling is a primary objective.

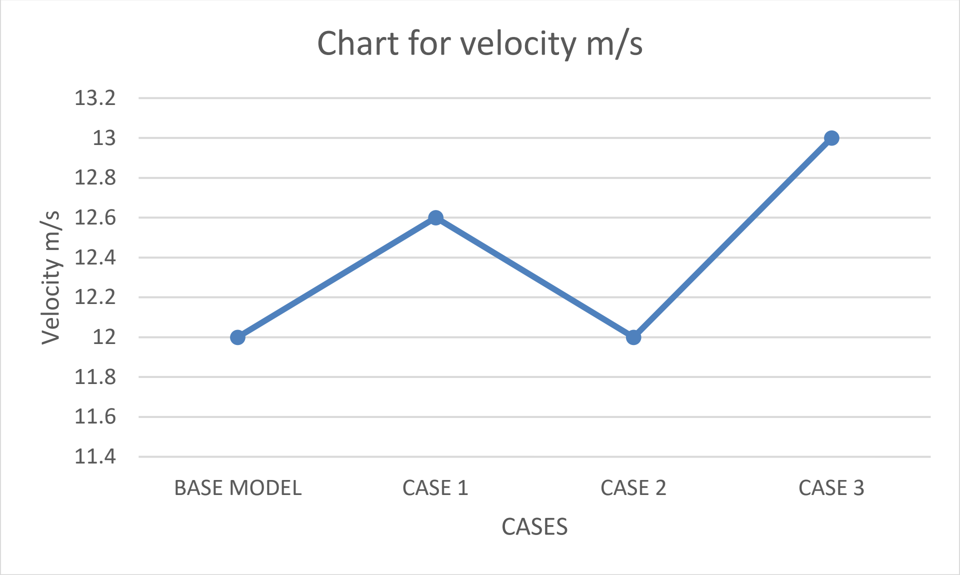

Chart 5.3 Velocity Analysis in ETHE Base Model and Cases

Base Model:
In the base model, the inlet velocity is 11 m/s, and the outlet velocity is 12 m/s. This indicates a minimal change in velocity as the air passes through the heat exchanger. The slight increase in velocity from the inlet to the outlet suggests that the base model is relatively efficient in maintaining the airflow velocity while facilitating heat exchange.
Case 1:
Case 1 presents nearly identical velocity values to the base model, with an inlet velocity of 11 m/s and an outlet velocity of 12.6 m/s. This suggests that the modifications made in Case 1 did not significantly affect the velocity of the air passing through the heat exchanger. The minimal change in velocity implies that the primary focus of Case 1's modifications may not have been on improving velocity-related aspects but could have been on other factors.
Case 2:
In Case 2, the inlet velocity remains consistent with the previous cases at 11 m/s. However, the outlet velocity increases to 12 m/s. This indicates a slight improvement in maintaining or even increasing the velocity through the heat exchanger. A higher outlet velocity can be advantageous in scenarios where increased airflow is desired.
Case 3:
Case 3 exhibits an inlet velocity of 11 m/s, which is similar to the previous cases. However, the outlet velocity increases to 13 m/s, marking the highest outlet velocity among all cases. This suggests that the modifications implemented in Case 3 have effectively increased the velocity of the outgoing air stream. A higher outlet velocity can enhance the heat exchanger's performance, especially in applications where rapid air circulation is essential

Comparative Study Analysis of Earth Tube-Type Heat Exchangers.

In our comprehensive study of earth tube-type heat exchangers, we have investigated the performance across various cases, focusing on pressure, temperature, and velocity. Each case represents a different configuration or set of modifications, offering unique insights into the impact of these changes on the heat exchanger's efficiency.
Comparative Chart:
To visually represent the findings, the comparative chart summarizes the key data for each case in terms of pressure, temperature, and velocity. The chart clearly illustrates the performance variations among the cases, with Case 3 consistently showing notable improvements across multiple parameters.

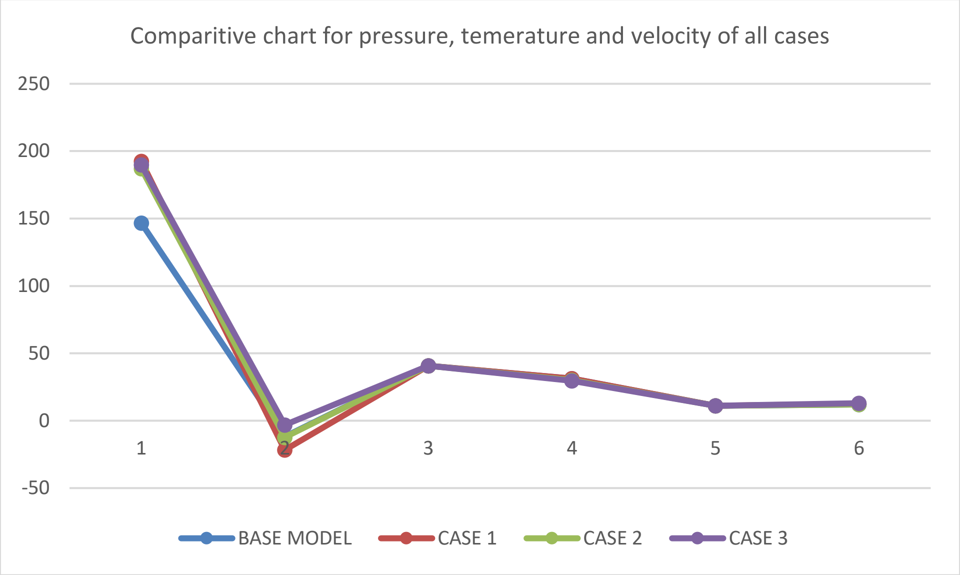

Chart 5.4 Comparative chart for pressure, temperature and velocity of all cases

Pressure:

Pressure drop within a heat exchanger is a crucial parameter to consider, as it directly affects system performance and energy consumption. We observed that Case 3 exhibited the most promising results in terms of pressure drop. The modifications in Case 3 effectively reduced pressure drop and minimized negative pressure values. This improvement is essential, as lower pressure drops imply reduced energy consumption and enhanced overall system efficiency. The comparative chart provides a visual representation of the pressure data across all cases.

Temperature:

Temperature performance is a key consideration, particularly in applications where efficient cooling or heat extraction is required. In our analysis, Case 3 again stands out by significantly improving heat transfer efficiency. It resulted in a more substantial temperature decrease in the outgoing air stream. This enhancement has the potential to make Case 3 the preferred choice for applications that prioritize efficient cooling. The comparative chart visually illustrates the temperature variations in different cases.

Velocity:

Velocity plays a crucial role in applications where rapid air circulation and efficient heat exchange are essential. In this context, Case 3 excelled by increasing the airflow rate through the heat exchanger, leading to a higher outlet velocity. This is advantageous for applications demanding rapid air circulation and improved heat exchange efficiency. The comparative chart provides a clear comparison of velocity values across all cases.

V. CONCLUSION

Case 3 emerges as the optimal configuration among various designs of Earth Tube Heat Exchangers (ETHX) studied in this research. Its superior performance, demonstrated through reductions in pressure drop, temperature decrease, and increased outlet velocity, underscores its suitability for applications demanding efficient cooling of PV panels. The quantitative assessment provided in this study offers valuable insights for optimizing ETHX systems, enhancing their performance, and reducing energy consumption. Case 3 stands out as a compelling choice for those seeking to maximize the efficiency of ETHX technology in specific applications.

REFERENCES

[1] Shah, R.K., & Sekulic, D.P. (2003). Fundamentals of Heat Exchanger Design (1st ed.). John Wiley & Sons Inc., Hoboken.

[2] Z. Ghafoor and Y. K. Khdir, "Experimental Investigation of Earth Tube Heat Exchanger (ETHE) for Controlled Ventilation in Erbil," 2020 6th International Engineering Conference “Sustainable Technology and Development" (IEC), Erbil, Iraq, 2020, pp. 80-85, doi: 10.1109/IEC49899.2020.9122908.

[3] Bansal, V., Misra, R., Agrawal, G. D., & Mathur, J. (2010). Performance analysis of earth–pipe–air heat exchanger for summer cooling. Energy and Buildings, 42(5), 645-648.

[4] Bansal, V., Misra, R., Agrawal, G. D., & Mathur, J. (2009). Performance analysis of earth–pipe–air heat exchanger for winter heating. Energy and Buildings, 41(11), 1151-1154.

[5] Gustafsson, S-I. (1993). Are earth tube heat exchangers of interest when heating buildings? International Journal of Energy Research, Volume 17(Issue 7), First published in September 1993. [DOI: https://doi.org/10.1002/er.4440170705]

[6] Florides, G., & Kalogirou, S. (2007). Ground heat exchangers—A review of systems, models, and applications. Renewable Energy, 32(15), 2461-2478.

[7] Taurines, K., Giroux-Julien, S., Farid, M., & Ménézo, C. (2021). Numerical modeling of a building integrated earth-to-air heat exchanger. Applied Energy, 296, 117030.

[8] Maneesh Kaushal (2021) "Performance analysis of clean energy using geothermal earth to air heat exchanger (GEAHE) in Lower Himalayan Region – Case study scenario" Energy & Buildings 248 (2021) 111166; https://doi.org/10.1016/j.enbuild.2021.111166.

[9] S.F. Ahmed et al. (2021) "A parametric analysis of the cooling performance of vertical earth-air heat exchanger in a subtropical climate" Renewable Energy 172 (2021) 350e367; https://doi.org/10.1016/j.renene.2021.02.086.

[10] Nethra MR & Kalidasan B (2020) "Earth tube heat exchanger design for efficiency enhancement of PV panel" Materials Today: Proceeding Accepted 15 February 2020; https://doi.org/10.1016/j.matpr.2020.02.387.

[11] Andrew Zajch et al. (2020) "Influence of daily temperature behavior on earth-air heat exchangers: A case study from Aichi, Japan" City and Environment Interactions 8 (2020) 100054; https://doi.org/10.1016/j.cacint.2020.100054.

[12] H.P. Diaz-Hernandez et al. (2020) "Experimental study of an earth to air heat exchanger (EAHE) for warm humid climatic conditions" Geothermics 84 (2020) 101741; https://doi.org/10.1016/j.geothermics.2019.101741.