INVESTIGATION OF TUBE MATERIAL ALTERATIONS AND COMPUTATIONAL FLUID DYNAMICS (CFD) ANALYSIS ON SHELL AND HELICAL TUBE EXCHANGER WITH HYBRID NANO FLUIDS

BY

Medisetti Abhinay[1], Thota Viany Kumar[2], Boddu Siva Chkrababu[3], Panthadi Durga Prasad[4], K.Vijetha[5]
[1,2,3,4]B.Tech, [5]Assistant Professor, Aditya College of Engineering & Technology, Surampalem, A.P, India, 533437

ABSTRACT

Heat exchangers play a vital role in various applications ranging from miniature electronic devices to massive space shuttles, facilitating the transfer of heat between different mediums through direct or indirect contact. In industrial settings, the traditional use of copper has been superseded by more cost-effective materials like carbon steel and aluminum. This research project aims to leverage the capabilities of ANSYS Design Modeler to construct a heat exchanger model and subsequently employ ANSYS Fluent software to investigate the potential of replacing aluminum with silicon carbide (SiC) as the inner tube material. Silicon carbide exhibits favorable heat transfer characteristics and superior corrosion resistance compared to aluminum, making it an attractive alternative. Furthermore, the study delves into the heat transfer performance of various nano fluids, of Copper oxide & Titanium nano particles are taken ,as they providing insights into enhance their potential applications in heat exchanger systems. Hybrid nano fluid of various volume fractions are taken to enhance the heat transfer rate..

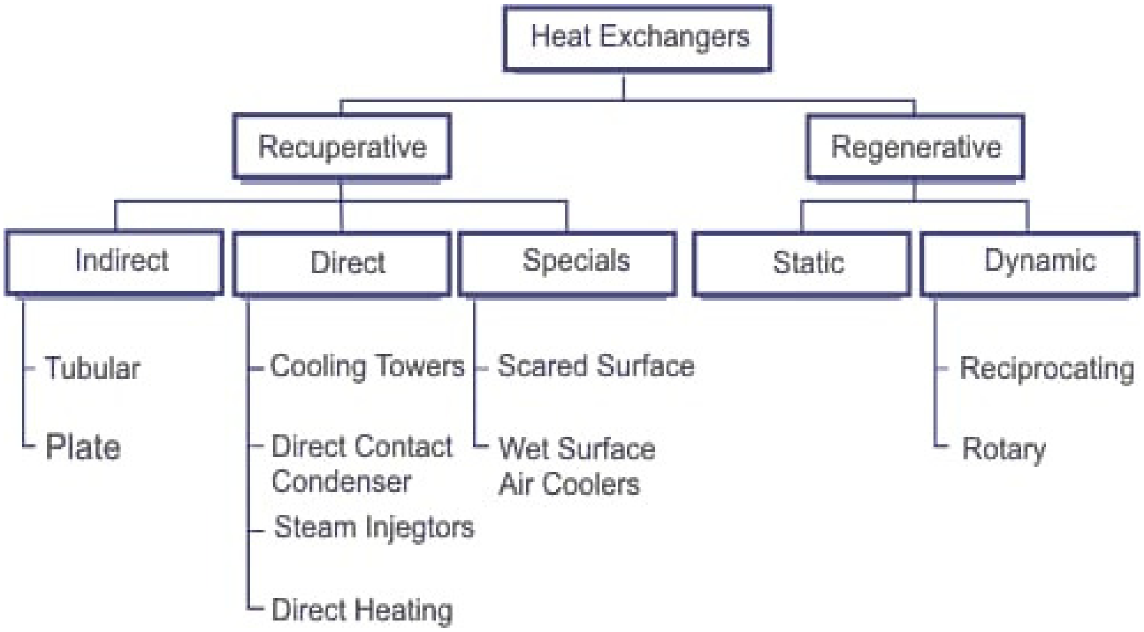
KEYWORDS: Ansys- Ansys Fluent-Hybrid Nano Fluids-Silicon Carbide–Copper Oxide -Titanium NanoParticles - Heat Transfer

1. INTRODUCTION

1. 1SHELL AND TUBE HEAT EXCHANGER

1.1.1Introduction

- Heat exchangers are like heat movers. They move heat from one thing to another with or without mixing them up. They're used in stuff like fridges to keep things cool and in big machines to make processes work better.
- Basically, they help keep things the right temperature in lots of stuff we use every day . In the realm of industrial engineering and thermal management, the design and optimization of heat exchangers stand as pivotal endeavours.
- Shell and tube heat exchangers, renowned for their versatility and widespread application across various industries, play a crucial role in transferring heat efficiently between two or more fluid streams.
- Furthermore, this project ventures into the realm of material science, where the selection of tube materials plays a pivotal role in the overall performance and longevity of heat exchangers. With a concerted exploration of alternative tube materials, we endeavour to identify materials that not only exhibit superior thermal conductivity but also align with sustainability objectives, such as recyclability and reduced environmental footprint.



1.1.2 Classification

Fig 1.1 Classifications of heat exchanger

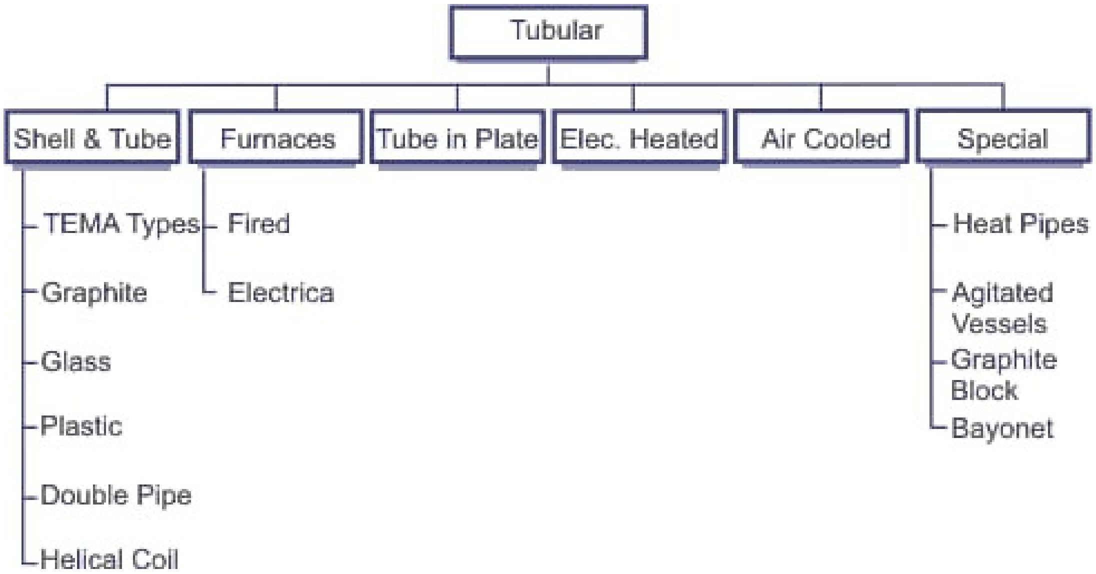


Fig 1.2 Classifications of tubular heat exchanger

1.2 COMPUTATIONAL FLUID DYNAMICS (CFD) ANALYSIS

1.2.1 Definition

- Computational fluid dynamics (CFD) is a subfield of fluid mechanics that solves and analyzes fluid flow problems using numerical techniques and algorithms..

1.2.2Advantages of CFD Analysis

- Provides insights into complex flow phenomena like turbulence, multi-phase flows, and heat transfer.
- Facilitates the thorough visualization and study of fluid flow patterns, pressure distributions, and temperature fields.
- Cuts down on development expenses and time by enabling virtual testing and optimization of designs prior to real prototyping.
- Provides the opportunity to research situations that are costly, time-consuming, or hard to replicate in an experiment.
- Enables parametric investigations with ease by adjusting geometric setups and input conditions.

1.2.3 Disadvantages of CFD Analysis

- High computational cost: CFD simulations can be computationally expensive, requiring sophisticated hardware and lengthy simulation times, especially for complicated geometries and turbulent flows.

- Numerical mistakes and uncertainties: Discreteization schemes, iterative solvers, and truncation errors can introduce numerical inaccuracies into CFD results that can affect prediction accuracy.
- Garbage in, garbage out: The precision and completeness of the input data—such as geometry, material parameters, and boundary conditions—have a significant impact on the caliber of CFD outputs.
- Limited validation data: It may be difficult or impossible to gather experimental data for validation and verification, particularly for complicated flow conditions, which makes it difficult to evaluate theaccuracy of the CFD model.
- Grid dependency: The accuracy and resolution of the computational mesh can have a significant impact on CFD results, thus attentive.

1.3 NANO FLUIDS

1.3.1 Definition

- Engineered colloidal suspensions of nanoscale particles (1–100 nm) scattered in a base fluid, like oils, ethylene glycol, or water, are known as nanofluids.
- They are thought to represent a novel class of fluids for heat transfer with improved thermal characteristics.

1.3.1.1 Why Nanofluids ?

Nanofluids are fluids containing suspended nanoparticles (typically less than 100 nm in size). The nanoparticles, made of materials like metals, metal oxides, or carbon nanotubes, have higher thermal conductivities than the base fluid. Their presence in the fluid increases the overall thermal conductivity of the nanofluid, leading to improved heat transfer capabilities. Additionally, the nanoparticles can enhance convective heat transfer by increasing the effective surface area and altering the flow characteristics. These properties make nanofluids attractive for applications requiring efficient heat transfer, such as in cooling systems, heat exchangers, and solar energy systems.

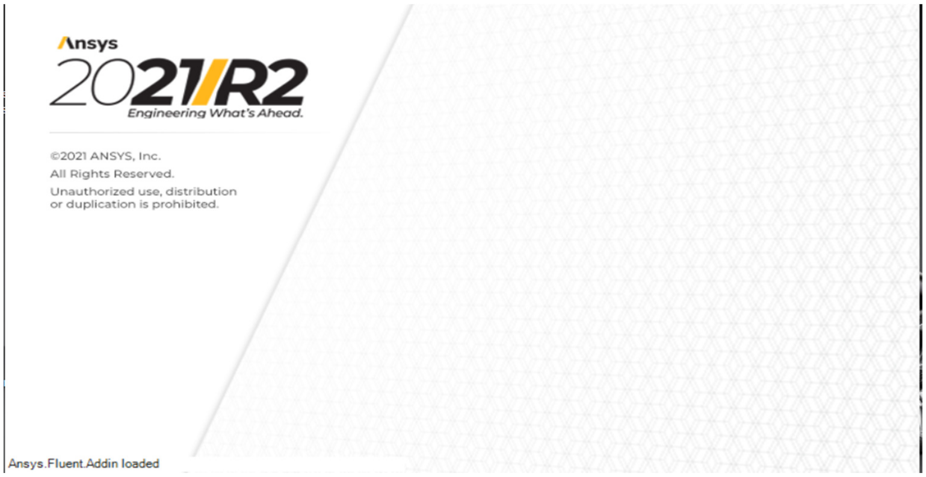
1.3.2 History and Development

- Choi and Eastman developed the concept at Argonne National Laboratory in the 1990s.
- Aimed to enhance the traditional heat transfer fluids' thermal characteristics for use in a range of industries.
- The initial studies were on oxide and metallic nanoparticles dissolved in ethylene glycol or water.

1.4 ANSYS

1.4.1 Introduction

ANSYS 2021 is a comprehensive software suite for engineering simulation, developed by ANSYS, Inc. It is a widely used tool in various industries, including automotive, aerospace, construction, and manufacturing, among others. The software provides a range of advanced simulation capabilities to analyze and optimize designs, ensuring product performance, safety, and reliability.



©2021 ANSYS, Inc.
All Rights Reserved.
Unauthorized use, distribution
or duplication is prohibited.

Ansys.Fluent.Addin loaded

Fig 1.3 Ansys Software

1.4.2 Types of Analysis

ANSYS 2021 offers a wide range of physics-based solvers and tools for various types of simulations, including:

1. Structural analysis: This module allows engineers to analyze the structural integrity, stress, deformation, and fatigue life of components and assemblies under different loading conditions.

2. Computational fluid dynamics (CFD): This module simulates the flow of fluids (liquids and gases) and associated phenomena, such as heat transfer, turbulence, and multiphase flows.

3. Electromagnetics: This module simulates electromagnetic fields, electromagnetic compatibility (EMC), and radio frequency (RF) effects in electronic devices and systems.

4. Explicit dynamics: This module is used for simulating short-duration events, such as impact, explosions, and drop tests, involving large deformations and complex material behavior.

5. Multibody dynamics: This module simulates the motion and interactions of interconnected rigid and flexible bodies, commonly used in the design of mechanical systems and mechanisms.

6. System-level simulations: ANSYS 2021 offers tools for system-level modeling and simulation, enabling the integration of multiple physics domains and the analysis of complex systems.

7. Optimization and design exploration: The software includes tools for design optimization, parametric studies, and design exploration, allowing engineers to identify the best design configurations based on specified objectives and constraints.

1.4.3 Design Modeler

DesignModeler is tightly integrated with other ANSYS products, such as ANSYS Meshing and ANSYS Mechanical, allowing for a seamless workflow from geometry creation to simulation and analysis. It is widely used in various engineering disciplines, including mechanical, aerospace, automotive, and biomedical, to create and prepare geometries for finite element analysis (FEA), computational fluid dynamics (CFD), and other simulation tasks.

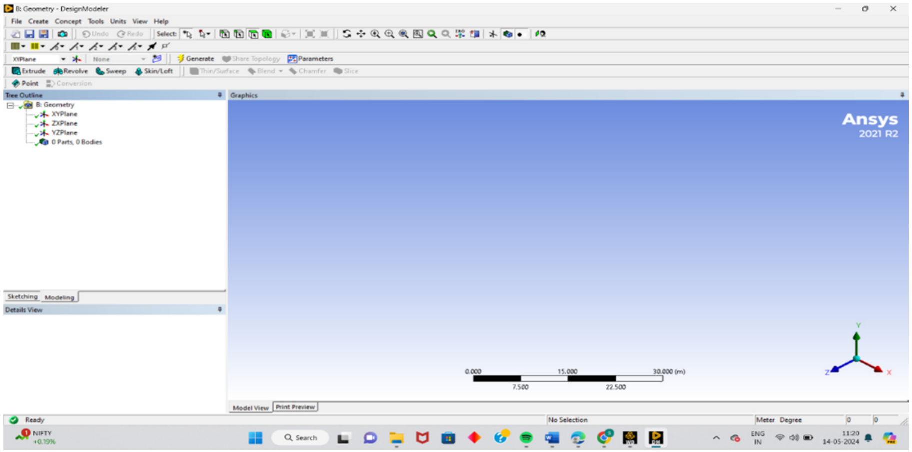


Fig 1.3 Ansys Design Modeler

In ANSYS, creating a mesh is a crucial step in the simulation process as it discretizes the geometry into smaller elements, allowing for numerical analysis. Here's a general overview of how to create a mesh using ANSYS Meshing:

1. Import or create the geometry: You can either import an existing CAD model or create the geometry within ANSYS DesignModeler.

2. Access ANSYS Meshing: Once you have the geometry, launch ANSYS Meshing from within the ANSYS Workbench environment.

3. Set up the meshing task: In ANSYS Meshing, you'll need to set up a new meshing task. This involves specifying the geometry, physics preferences (e.g., CFD, structural), and any predefined mesh settings or sizing controls.

4. Mesh setup: Here are some common steps for setting up the mesh:

   a. Assign material properties to the geometry if needed.

   b. Define mesh sizing controls, such as global or local element size, refinement regions, or sizing functions.

   c. Specify mesh methods or algorithms (e.g., sweep, Multi Zone, CFD, or automatic).

   d. Set up inflation layers for capturing boundary layers (important for CFD simulations).

   e. Define mesh disfeaturing settings to control the level of geometric detail in the mesh.

5. Mesh preview and quality checks: Before generating the mesh, you can preview the mesh and check the mesh quality metrics, such as element quality, skewness, aspect ratio, and Jacobean ratio. ANSYS provides various quality checks and criteria to ensure a high-quality mesh.

6. Generate the mesh: Once you're satisfied with the mesh setup, you can generate the mesh by executing the meshing operation. ANSYS will discretize the geometry into finite elements based on your settings.

7. Mesh refinement (optional): After generating the initial mesh, you may need to refine the mesh in specific regions or globally to improve accuracy or capture important features. ANSYS Meshing provides various refinement tools, such as face meshing, edge meshing, or mesh morphing.

8. Mesh quality checks and fixes: After generating the mesh, it's important to perform mesh quality checks again and address any issues, such as highly distorted elements, using the mesh repair tools available in ANSYS Meshing.

9. Export the mesh: Finally, you can export the mesh file in the appropriate format (e.g., CDB, ANS) for use in other ANSYS solvers, such as ANSYS Mechanical or ANSYS Fluent.

It's important to note that mesh generation is a critical step that can significantly impact the accuracy and convergence of your simulations. ANSYS provides various meshing algorithms, controls, and quality checks to help you create high-quality meshes tailored to your specific simulation requirements.

1.5 Finite Element Methods (FEM)

1.5.1 Introduction to the Finite Element Method

The Finite Element Method (FEM) is a powerful numerical technique used to solve complex engineering and mathematical problems involving irregular geometries, complex material properties, and intricate boundary conditions. It is a widely adopted method in various fields, including structural analysis, heat transfer, fluid dynamics, and electromagnetic simulations.

1.5.2 Historical Background

The origins of the FEM can be traced back to the work of mathematicians and engineers in the 1940s and 1950s, including Courant, Argyris, and Clough. However, it wasn't until the advent of digital computers that the FEM became widely adopted and practical for solving large-scale problems.

1.5.3 Discretization and Element Formulation

The fundamental idea behind the FEM is to divide the domain of interest (a solid body, fluid region, or any continuous system) into smaller, interconnected subdomains called finite elements. These finite elements are connected at specific points, known as nodes or grid points. The behavior of each element is characterized by a set of algebraic equations that approximate the governing differential equations of the problem.

The process of deriving the governing equations for each element is known as element formulation. This typically involves using variational principles, such as the principle of minimum potential energy or the principle of virtual work, to establish the element equations.

1.5.4 Types of Finite Elements

Finite elements can be classified based on their dimensionality and geometric shape:

1. One-Dimensional Elements: These include line elements, which are used for problems involving beams, trusses, and other slender structures.

2. Two-Dimensional Elements: These include triangular and quadrilateral elements, which are commonly used for plane stress/strain problems, plate and shell analysis, and heat transfer problems in two dimensions.

3. Three-Dimensional Elements: These include tetrahedral, hexahedral, and other polyhedral elements, which are used for solving problems in three-dimensional space, such as stress analysis of complex structures, fluid flow simulations, and electromagnetic field computations.

1.5.5 Steps in the Finite Element Analysis

1. Discretization: The first step is to discretize the domain into finite elements, which can be one-dimensional (lines), two-dimensional (triangles or quadrilaterals), or three-dimensional (tetrahedra, hexahedra, or other shapes). This process is known as mesh generation and is often performed using specialized software tools.

2. Element Formulation: The next step involves deriving the governing equations for each element, typically based on principles such as the principle of minimum potential energy or the principle of virtual work.

3. Assembly: Once the element equations are formulated, they are assembled into a global system of equations that represent the entire domain. This process involves enforcing continuity and equilibrium conditions at the nodes shared by adjacent elements.

4. Boundary Conditions: Appropriate boundary conditions, such as prescribed displacements, forces, or fluxes, are applied to the global system of equations.

5. Solution: The resulting system of algebraic equations is then solved numerically, typically using direct or iterative methods, to obtain the desired quantities, such as displacements, stresses, temperatures, or velocities, at the nodes.

6. Post-Processing: Finally, the nodal results are interpolated to obtain the solution within each element, and the desired quantities, such as stresses or heat fluxes, are computed and visualized.

1.5.6 Mesh Refinement and Adaptive Analysis

One of the key advantages of the FEM is the ability to refine the mesh (discretization) in regions of interest or where higher accuracy is required. This process, known as mesh refinement or adaptive analysis, involves subdividing existing elements into smaller elements, thereby increasing the resolution of the solution in those regions.

Adaptive analysis techniques can be based on error estimation strategies, where the solution error is estimated and used to guide the mesh refinement process. This approach ensures that computational resources are focused on regions with high error, leading to more accurate and efficient simulations.

1.5.7 Error Estimation and Convergence Studies

Error estimation and convergence studies are crucial aspects of finite element analysis. Error estimation techniques aim to quantify the difference between the approximate numerical solution and the exact solution of the governing equations. This information can be used to assess the accuracy of the solution and guide mesh refinement or adaptive analysis processes.

Convergence studies involve systematically refining the mesh and observing the behavior of the solution as the mesh size decreases. If the solution converges to a stable value as the mesh is refined, it provides confidence in the accuracy of the results. Convergence rates can also be analyzed to assess the performance of the numerical method and verify the theoretical convergence properties.

1.5.8 Advantages and Applications

The Finite Element Method offers several advantages, including:

- Ability to handle complex geometries and arbitrary boundary conditions
- Capability to model heterogeneous and anisotropic materials
- Flexibility in refining the mesh (discretization) in regions of interest
- Wide range of applications, including structural analysis, heat transfer, fluid dynamics, and electromagnetics

1.5.9 Applications in Various Fields

1. Structural Analysis: FEM is widely used in the analysis of structures, such as buildings, bridges, aircraft, and automotive components. It enables the calculation of stresses, deformations, and structural stability under various loading conditions.

2. Heat Transfer: The method is employed in the simulation of heat transfer problems, including conduction, convection, and radiation processes. Applications range from electronic device cooling to thermal management in power plants and aerospace systems.

3. Fluid Dynamics: FEM is used in computational fluid dynamics (CFD) to simulate fluid flow and related phenomena, such as aerodynamics, hydrodynamics, and multiphase flows.

4. Electromagnetics: The method is applied in the analysis of electromagnetic fields and wave propagation, with applications in antenna design, microwave devices, and electromagnetic compatibility studies.

5. Biomedical Engineering: FEM is utilized in the analysis of biological systems, including biomechanics, tissue engineering, and medical imaging techniques like magnetic resonance imaging (MRI) and computed tomography (CT).

1.6Problem Statement

- The majority of research efforts focus on improving the heat transfer rate of heat exchangers through methods such as incorporating inserts and altering geometries. However, investigations into alternative materials have been relatively less.
- Recent researches predominantly centers around the utilization of various nanofluids.
- While some studies have delved into material substitution, there has been a notable absence of research focusing on advanced alloy materials**.**

1.1 Objective of Work

- Conduct Computational Fluid Dynamics (CFD) analysis of shell and tube heat exchangers.
- Utilize Ansys design modeler for Heat Exchanger Model design.
- Use ANSYS Fluent software for analysis and exploration of tube material alternatives.
- Evaluate heat transfer and corrosion characteristics of Silicon Carbide. Optimize heat exchanger design for efficiency and cost-effectiveness.
- Conducted the simulation with hybrid nanofluids that are titatnium nano particles and Copper oxide in the various mixing fractious and volume fractions

2. LITERATURE REVIEW

Ashkan Alimoradiet.al [1]Explored the impact of fluid physical properties, operational variables, and geometric factors on the Nusselt numbers of both shell and tube sides. The research revealed that doubling the pitch size led to a 10% increase in the Nusselt number on the shell side, while the impact on the tube side was minimal. Additionally, the study put forward two correlations to estimate the Nusselt numbers on the coil side and shell side, respectively.**Khan A et.al[2]**This paper performed computational fluid dynamics (CFD) simulations and experimental analysis on shell and tube heat exchangers using round and hexagonal tubes for a range of flow velocities in both parallel flow and counter flow arrangements. The examination was carried out with fluid velocities ranging from 0.75 m/s to 2.75 m/s at Reynolds numbers of 10,000 to 15,000. The authors found that the rate of heat transfer is directly proportional to the number of tube turns and the fluid velocity. The optimal fluid velocity for maximum heat transfer was 2.75 m/s. Introducing hexagonal tubes increased the heat transfer rate by 13.5% and the contact surface area by 31.8% compared to round tubes. This is because the hexagonal geometry increased the surface area for heat transfer and disrupted the fluid flow, enhancing turbulence and heat transfer. The authors validated their simulation results with experiments and found good agreement, with lower uncertainty for the hexagonal tube cases (4.2%) compared to round tubes (9.2%)

Ahmet Talat İnan et.al[3]This paper presents a comparative study of conventional one-piece baffle plate and perforated baffle plate designs for shell-and-tube heat exchangers using computational fluid dynamics (CFD) analysis. The authors designed a shell-and-tube heat exchanger with a single shell and a single tube pass, and analyzed the effects of baffle plate design on pressure drop and heat transfer coefficient on the shell side. The study investigated four different mass flow rates (1.2, 1.5, 1.8, and 2.1 kg/h) for both baffle plate designs. The results showed that the heat exchanger with a perforated baffle plate exhibited lower pressure drop and heat transfer coefficient compared to the conventional one-piece baffle plate design. However, the heat transfer rate per pressure drop was found to be higher for the perforated baffle plate design, indicating improved thermohydraulic performance. The authors concluded that the heat transfer rate per pressure drop could be improved by 39% to 42% using the perforated baffle plate design compared to the conventional one-piece baffle plate design.

Roohollah Babaei-Mahani et.al[4]This study uses CFD analysis to examine a shell-and-tube condensation heat exchanger in microwave reduction processes. Variations in gas and water inlet temperatures and mass flow rates were analyzed. Key findings include increased outlet temperatures with higher gas inlet temperatures and improved heat transfer rates with increased air velocity. Additionally, higher water inlet temperatures result in elevated outlet air temperatures but reduced heat transfer rates. These results highlight the significant impact of operational parameters on heat exchanger performance in waste tire reduction processes.

Arun Kumar Tiwari et.al[5]The research aimed to improve the heat transfer performance of a triple tube heat exchanger (TTHE) by incorporating various inserts and nanofluids. Experimental tests and computational fluid dynamics (CFD) simulations were carried out to investigate the effects of these modifications. Among the different nanofluid concentrations studied, a volume fraction of 1.0% was found to be optimal. The analysis revealed that using a rib-type insert in combination with a nanofluid yielded the highest Nusselt number of 94.70 at a Reynolds number of 6200.08, indicating enhanced heat transfer. Interestingly, the friction factor exhibited a continuous decrease as the Reynolds number increased.The study also evaluated the enhancement factor, which considers both heat transfer and pressure drop. The maximum enhancement factor of 0.75 was achieved when using a rib insert with a WO3/water nanofluid at a Reynolds number of 1990.23. Comparing the experimental and CFD results, the discrepancies were within acceptable limits, with a maximum error of 4.37% for the overall heat transfer coefficient and 2.05% for the effectiveness.

Elsaid A. M. et.al[6]This paper presents a numerical analysis of a parallel flow shell and tube heat exchanger using different hybrid nanofluids as the cooling medium. The authors studied the effect of dispersing nanoparticles on the Prandtl number and heat transfer characteristics. The hybrid nanofluids investigated were Al_2O_3+MWCNT/water, Al_2O_3+BeO/water, Al_2O_3+AlN/water, and Al_2O_3+TiO_2/water with a total nanoparticle concentration of 2 vol%. The results showed that the heat transfer rate was enhanced by a maximum of 16.5% for the Al_2O_3+MWCNT/water hybrid nanofluid. Additionally, the Prandtl number was observed to decrease with the addition of nanoparticles, with a maximum reduction of 10.5% for the Al_2O_3+TiO_2/water hybrid nanofluid. The authors concluded that the Al_2O_3+MWCNT, Al_2O_3+AlN, and Al_2O_3+BeO hybrid nanofluids could be considered as better coolants for shell and tube heat exchanger applications.

Atul Bhattad et.al[7]The paper examines hybrid nanofluids, which show promise for enhancing heat transfer performance. Key findings include:

Hybrid nanofluids demonstrated significantly higher thermal conductivity compared to conventional fluids.

An Al_2O_3+MWCNT/water nanofluid exhibited around 16.5% higher heat transfer rate.

Adding nanoparticles reduced the Prandtl number, indicating better heat transfer characteristics.

However, challenges such as long-term stability, optimizing processing methods, developing suitable nanomaterial combinations, and cost need to be addressed before practical applications. Substantial further research is needed to overcome these hurdles.

Abed A. M.et.al[8]This study investigates the heat transfer and friction factor characteristics of different twisted tapes, specifically the V-cut and P-TT types, inserted into horizontal pipes with twisted ratios (TR) of 4.0 and 6.0. Through numerical analysis, several conclusions are drawn. Firstly, the use of twisted tapes leads to enhanced heat transfer, with the V-cut twisted tape exhibiting superior performance compared to the P-TT tape across all twisted ratios. Additionally, the rates of heat transfer are consistently higher in pipes with twisted tape inserts than in plain pipes, attributed to the generation of strong vortex flow by the twisted tapes. Notably, the heat transfer rate is higher for TR=4.0 than for TR=6.0. Furthermore, the friction factor obtained from pipes with twisted tape inserts is significantly higher than that of plain pipes, especially with lower twisted ratios, facilitating increased tangential contact and swirling flow. Optimal heat transfer enhancement is achieved with a Nusselt number ratio of 8.75, observed with the V-cut twisted tape at TR=4.0 and a Reynolds number of around 9000. Lastly, the highest thermal performance factor of 4.45 is attained with the V-cut twisted tape at TR=4.0 and a Reynolds number of 9000, indicating efficient heat transfer enhancement under model flow conditions.

Fetuga I. A. et.al[9]The paper presents a comprehensive analysis of the thermal performance of shell-and-tube heat exchangers (STHXs) utilized as waste heat recovery systems. Three different tube layout configurations were examined: triangular (STHX-T), rotated triangular (STHX-RT), and combined (STHX-C). The study utilized ANSYS-Fluent for a three-dimensional computational study under steady-state conditions, adopting the realizable k-epsilon turbulence model due to the possible transition from laminar to turbulence flow. The results indicated that the STHX-T layout exhibited the highest overall heat transfer coefficient, maximum pressure drop, and significant temperature changes in both the shell and tube sides, suggesting its superior performance in waste heat recovery applications compared to the other configurations. This study not only contributes to energy-saving systems but also serves as a guide for industries and future research in optimizing heat exchanger designs for effective waste heat recovery.

Kartal M. A. et.al[10]The paper presents a comprehensive analysis of the thermal performance and pressure drop (PD) characteristics of a shell-tube-heat exchanger with varying baffle spacings and flow rates. The study reveals that a 90 mm baffle spacing combined with a flow rate (FR) of 1.9 kg/h yields the highest heat transfer coefficient (HCO), indicating superior efficiency and thermal output. Conversely, the lowest PD levels are achieved with a 90 mm baffle spacing and a 0.9 kg/h FR, suggesting a more balanced thermohydraulic performance and pressure equilibrium. The findings suggest that selecting a 90 mm plate spacing can lead to a 15% improvement in system design compared to a 110 mm plate spacing.This research contributes new insights into the optimization of heat exchanger designs for enhanced thermal performance and reduced pressure drop.

Zolfalizadeh et.al[11]The paper presents a comprehensive experimental investigation on the effects of graphene nanoplate (GNP)/water nanofluids on the heat transfer performance of a shell-and-tube heat exchanger. The study reveals that the use of GNP/water nanofluids, particularly at a concentration of 0.06wt.%, significantly enhances the convective heat transfer coefficient (CVHTC) by 22.47%, resulting in a heat transfer rate improvement of 15.65% over the base fluid. However, it also notes an increase in pressure drop, which could lead to higher power consumption for the pump. The research concludes that despite the increased power demand, the nanofluid at 0.06wt.% concentration exhibits good performance, making it a promising option for improving heat exchanger efficiency. This study's novelty lies in its full experimental approach to assessing the attributes of nanofluids and its application in a newly designed shell-and-tube heat exchanger.

Fadhil N. A. et.al[12]The paper presents a comprehensive numerical investigation of heat transfer and pressure drop characteristics in a double pipe heat exchanger with corrugated tubes and rod baffles. The study explores the effects of varying corrugation depths, pitch, and rod baffle spacing on the performance of the heat exchanger across different Reynolds numbers. The results demonstrate that corrugated tubes significantly enhance heat transfer efficiency, with the average Nusselt number increasing by 25% to 55% for corrugation depths of 0.1 and 0.13, respectively. However, this enhancement comes at the cost of a higher friction factor, which is 66% to 133% greater than that of smooth tubes. The incorporation of both corrugated tubes and rod baffles leads to a thermal enhancement factor of up to 1.97 at the same pumping power. This study provides valuable insights into optimizing heat exchanger design for improved performance.

Sohrabi N. et.al[13]The paper concludes by highlighting the effectiveness of a conical spiral tube heat exchanger that utilizes Ag-HEG nanofluid and various turbulators to enhance heat transfer and manage pressure drop. The study employed numerical simulations with ANSYS Fluent software and the finite volume method. It was found that the four-blade turbulator with thirty revolutions provided the best thermal performance when compared to other models. Future research could explore the simultaneous use of non-Newtonian nanofluids and turbulators. The study also suggests that altering the number of blade revolutions can further improve the performance of the four-blade turbulator, which is already superior. One advantage of this type of converter is its cost-effectiveness and its ability to withstand high pressures and temperatures. However, a potential drawback is the additional space required for maintenance. The authors conclude that the combination of Ag-HEG nanofluids and the four-blade turbulator with thirty revolutions is optimal for improving heat transfer in conical spiral tube heat exchangers while maintaining an acceptable level of pressure drop. This combination outperforms traditional fluids and turbulators.

Jalili, P. et.al[14]The paper presents a numerical study on a shell and tube heat exchanger with non-continuous helical baffles, utilizing hybrid nanofluids composed of alumina, copper, and water. The study explores the impact of various helix angles, mass flow rates, and nanofluid concentrations on the heat transfer coefficient (HTC) and pressure drop (ΔP). The findings reveal that the optimal heat transfer occurs at a helix angle of 20° with a 2% intensity of the hybrid nanofluid and a flow rate of 1.685 kg/s. As the Reynolds number increases, the HTC also rises significantly. Conversely, the pressure drop decreases as the helix angle increases, indicating lower operating costs. The study concludes that using hybrid nanofluids can significantly enhance the heat transfer rate while simultaneously reducing the pressure drop compared to pure water. The performance evaluation factor (HTC/ΔP) shows a 6% improvement over previous studies, suggesting that hybrid nanofluids offer a promising avenue for improving heat exchanger efficiency. The research underscores the potential of hybrid nanofluids in thermal engineering applications, providing valuable insights for future advancements in heat exchanger design.

Cruz P. A. D. et.al[15]The paper presents a Computational Fluid Dynamics (CFD) analysis of CuO-water nanofluid in a shell and tube heat exchanger. The study explores the impact of varying particle loading and Reynolds number on heat transfer and pressure drop. Key findings include:

Enhanced Heat Transfer: Increasing particle loading and Reynolds number boosts heat transfer rates, with up to 48% enhancement observed at the highest particle loading.

Pressure Drop: Higher particle loading also results in a doubled pressure drop, indicating a trade-off between heat transfer improvement and pressure pressure.

In conclusion, CuO-water nanofluid shows promise for improving heat transfer in shell and tube heat exchangers, provided that the particle loading and flow rates are carefully managed to optimize performance. The study contributes valuable insights into the application of nanofluids in heat transfer systems, particularly for energy-intensive industries.

Hassan, Q. H. et.al[16]The paper concludes by highlighting the effectiveness of using nanofluids in car radiators to enhance heat transfer. The study utilized ANSYS Fluent for numerical analysis and compared the results with experimental data, finding an 8% error under similar boundary conditions. It was observed that increasing the concentration of nanoparticles in the fluid led to a higher Nusselt number, which indicates improved performance of the heat exchanger. The research also noted that the heat transfer efficiency of nanofluids is highly dependent on nanoparticle concentration and flow conditions, while being less influenced by temperature variations. This suggests that careful consideration of nanoparticle concentration and flow is crucial for optimizing thermal transfer in car radiators. The paper's findings contribute to the ongoing research on nanofluids and their potential applications in automotive cooling systems.

Khaled R. A.et.al[17]The study presented in the paper focused on the performance of double twisted circular tube heat exchangers with twisted tape inserts, exploring various twist ratios. The key findings indicate that heat exchangers with double twisted tubes outperform those with plain tubes in terms of thermal efficiency. The insertion of twisted tape within the tube enhances fluid mixing and heat transfer, especially when the inner tube is twisted. Simulations suggest that a twist ratio of 4 yields the best thermal performance. Additionally, the effectiveness of the heat exchanger increases significantly with the mass flow rate of hot water, particularly when the twist ratio is 5.The study concludes that the twist of the tubes is a crucial factor in heat transfer, especially at low mass flow rates 5.

3. METHODOLOGY

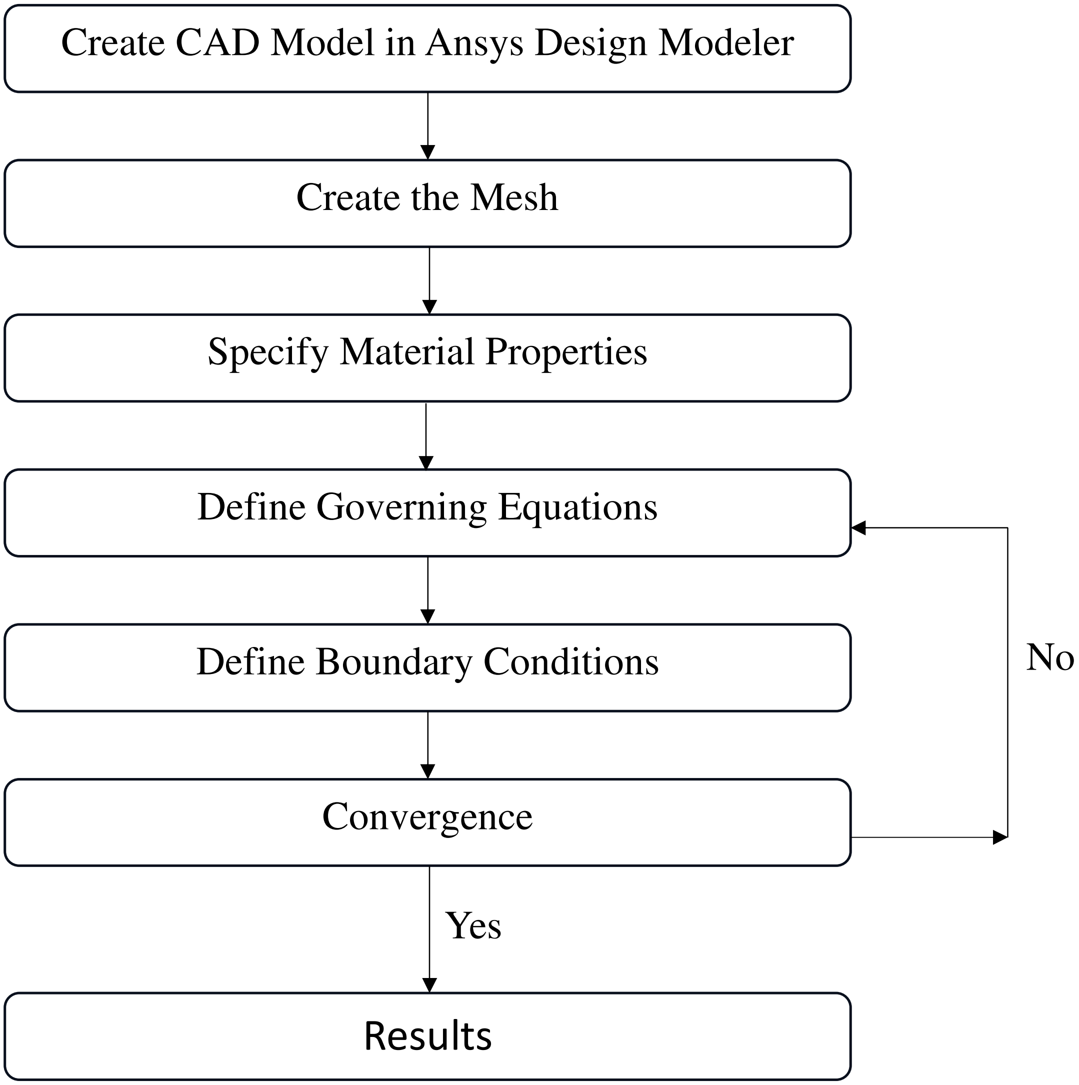


Fig 3.1 Work flow diagram of CFD analysis

3.1 Geometry and Mesh Generation

1. CAD Modeling: Create a detailed computer-aided design (CAD) model of the heat exchanger geometry, including all relevant components such as tubes, fins, headers, and inlet/outlet ports.

2. Mesh Generation: Import the CAD model into a mesh generation software or use the built-in meshing tools within the CFD software. Discretize the geometry into finite elements, typically using tetrahedral or hexahedral elements for a three-dimensional analysis.

3. Mesh Refinement: Refine the mesh in areas of interest, such as near solid-fluid interfaces, regions with high gradients, or areas where accurate results are critical. This step ensures that the numerical solution captures the relevant physics accurately.

3.2 Physics Modeling and Boundary Conditions

1. Governing Equations: Define the governing equations for the fluid flow and heat transfer processes. For a heat exchanger analysis, this typically includes the Navier-Stokes equations for fluid flow and the energy equation for heat transfer.

2. Fluid Properties: Specify the properties of the working fluids, such as density, viscosity, thermal conductivity, and specific heat capacity. These properties may be constant or temperature-dependent, depending on the problem.

3. Boundary Conditions: Apply appropriate boundary conditions to the model, including:

Inlet: Specify the inlet velocity, temperature, or mass flow rate.

Outlet: Define the outlet pressure or a pressure boundary condition.

Solid Walls: Impose no-slip and thermal boundary conditions (e.g., constant temperature or heat flux) on the solid surfaces.

Symmetry Planes (if applicable): Specify symmetry boundary conditions to reduce the computational domain and save computational resources.

3.3 Numerical Solution

1. Solution Approach: Choose the appropriate numerical method for solving the governing equations, such as the finite volume method or the finite element method. The FEM is particularly well-suited for handling complex geometries and unstructured meshes.

2. Discretization: Discretize the governing equations using the chosen numerical method, transforming the partial differential equations into a system of algebraic equations.

3. Solver Settings: Configure the solver settings, including the solution algorithm (e.g., segregated or coupled), convergence criteria, and relaxation factors.

4. Parallel Processing: Utilize parallel processing capabilities, if available, to speed up the computation time for large and complex models.

3.4 Post-Processing and Analysis

1. Convergence Check: Verify the convergence of the solution by monitoring the residuals and other relevant convergence indicators.

2. Result Visualization: Use post-processing tools to visualize the results, including velocity vectors, temperature contours, and pressure distributions.

3. Performance Evaluation: Evaluate the performance of the heat exchanger by analyzing the temperature profiles, heat transfer rates, and pressure drops.

4. Optimization: Perform parametric studies or optimization routines to improve the design and performance of the heat exchanger, considering factors such as material selection, geometry modifications, or operating conditions.

3.5 Validation and Verification

1. Experimental Data: Compare the CFD results with available experimental data or empirical correlations to validate the numerical model and assess its accuracy.

2. Mesh Independence Study: Conduct a mesh independence study by refining the mesh and ensuring that the solution converges to a stable value, independent of the mesh size.

3. Sensitivity Analysis: Perform sensitivity analyses to investigate the impact of various parameters, such as boundary conditions, turbulence models, or fluid properties, on the simulation results.

4. NUMERICAL WORK

4.1 NOMENCLATURE

A	Area (m^2)
C_p	Specific Heat (J/Kg K)
d	Diameter (m)
f	Diagonal distance between inlet and outlet valves of shell (m)
H	Height (m)
I	Turbulent Intensity
K	Thermal Conductivity W/m k
L	Coil length (m)
$\dot{m}$	Mass flow rate (Kg/s)
Nu	Nusselt number
P	Pitch (m)
Pr	Prandtl number
Q	Heat transfer (w)
Re	Reynolds number
T	Temperature (k)
u	Velocity (m/s)

Greek Symbols

ρ	Density (Kg/m^3)
μ	Dynamic viscosity (Pa.s)
Ø	Volume fraction

Subscripts

c	Coil
f	Fluid
hnf	Hybrid nanofluid
i	Inlet
in	Inner
nf	Nanofluid
o	Outlet
ot	Outer
sh	Shell
t	Tube
v	Valve
w	Water

4.2 DESIGN AND MESHING

To investigate the potential of employing Silicon Carbide material for shell and helical coil heat exchangers (SHCHEs), a comprehensive CAD model was developed. Fig4.1 depicts the geometrical configuration of the SHCHE design. The design parameters, including dimensionsof the heat exchanger and operational parameters were meticulously recorded in Table 4.1 and Table 4.2 respectively. The final model design was shown in Fig 4.2.

Parameters	**d_c**	**d_{sh}**	**H_c**	**Hsh**	**$d_{t,i}$**	**$d_{t,o}$**	**Dv**	**f**	**p**	**L**
Values	0.08	0.1	0.23	0.34	0.01	0.01	0.01	0.338	0.03	1.92

Table 4.1 Dimensions of the Heat Exchanger

Parameters	**Values**
u_c	2 – 6 m/s
u_{sh}	2 – 6 m/s
$T_{c,i}$	363 K
$T_{sh,i}$	293 K

Table 4.2 Operational parameters

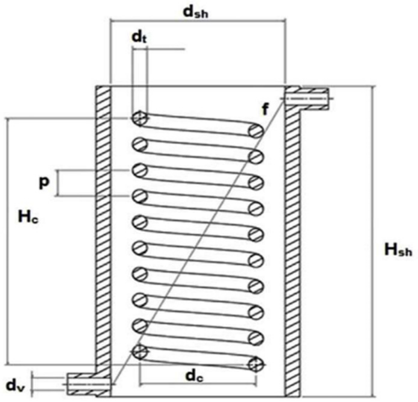


Fig 4.1 Geometrical parameters of the Heat exchanger

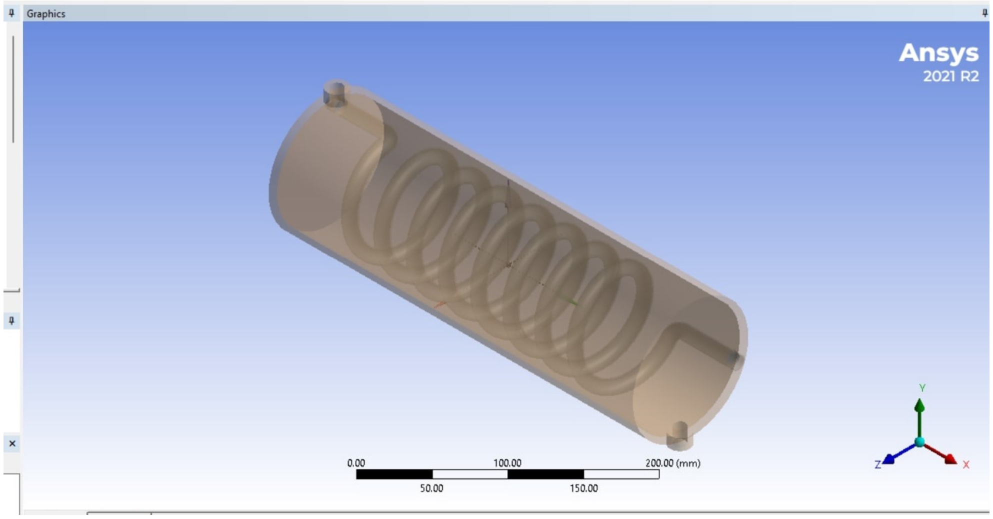
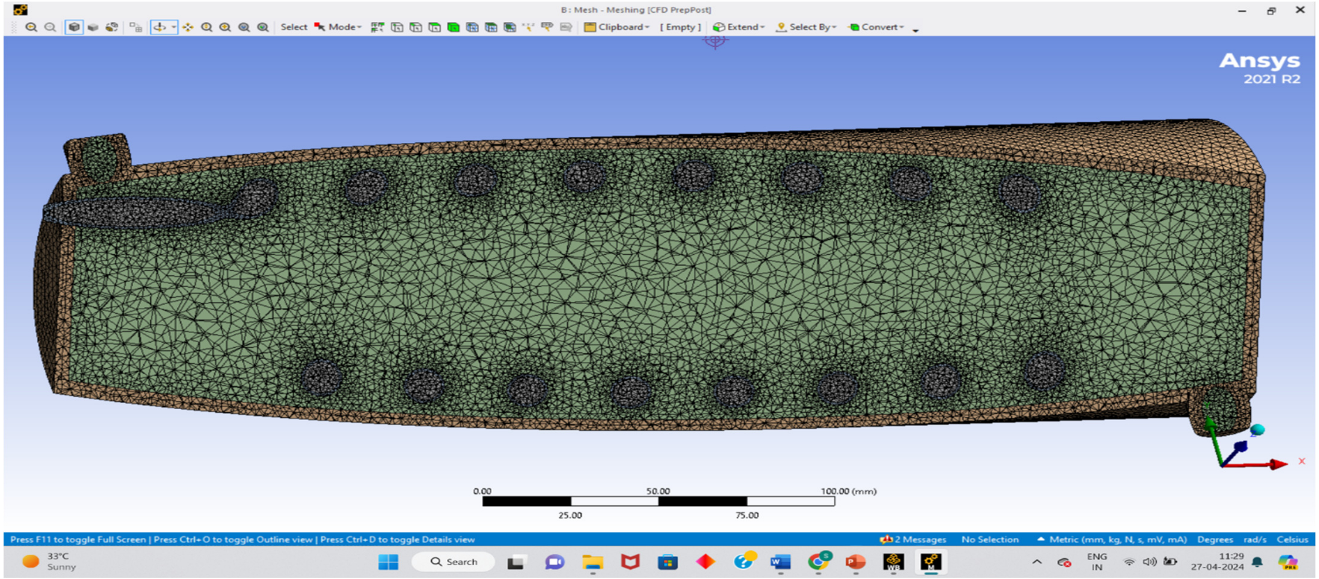


Fig 4.2 (a) Heat Exchanger CAD Model

To facilitate efficient numerical analysis, the CAD model was segmented into multiple smaller control volumes, as illustrated in Fig. 4.3. In addition to lowering computational complexity, this partitioning technique accelerated solution times by enabling parallelization of the computations. The method employed to ensure uniformity and continuity at mesh boundaries was the Patch Conforming Method.This method guarantees a smooth transition between adjacent mesh sections and removes numerical errors.The mesh building procedure made use of tetrahedral pieces. Tetrahedral meshes have several advantages, including the ability to precisely capture complex geometries and the flexibility to adjust the mesh locally in places of interest, such as areas near boundary layers or with steep gradients.

A comprehensive evaluation of the performance of the graphite flakes/copper composite material in SHCHE applications was made possible by the CAD model and the mesh creation technique, which served as the foundation for trustworthy numerical simulations.



Press F11 to toggle Full Screen | Press Ctrl+O to toggle Outline view | Press Ctrl+D to toggle Details view

Fig 4.3 (a) Mesh created on the Heat Exchanger

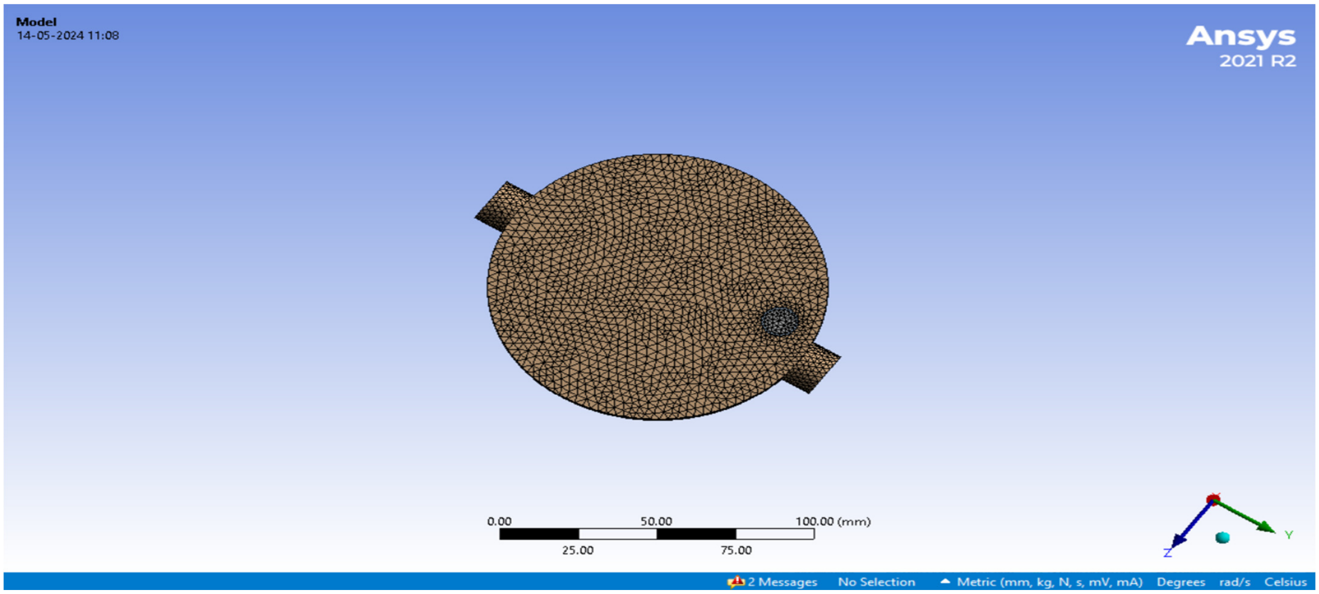


Fig 4.3 (b) Mesh created on the Heat Exchanger

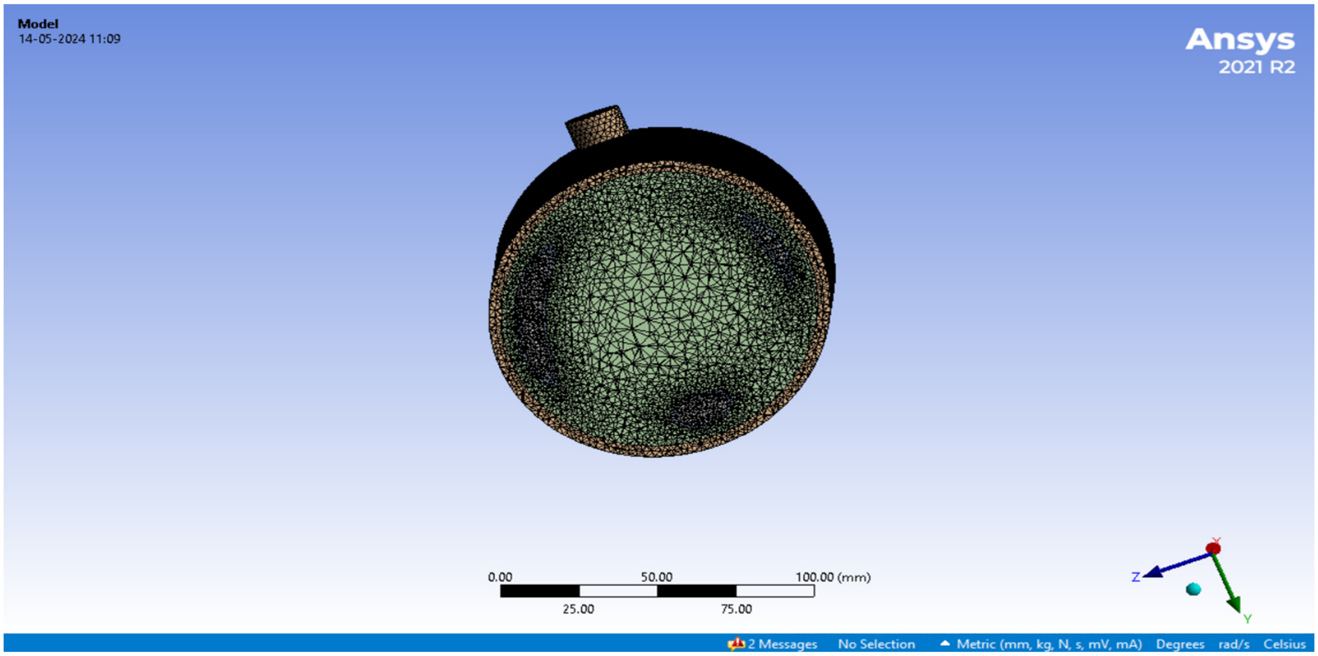


Fig 4.3 (c) Mesh created on the Heat Exchanger

Fig 4.3 (d) Mesh created on the Heat Exchanger

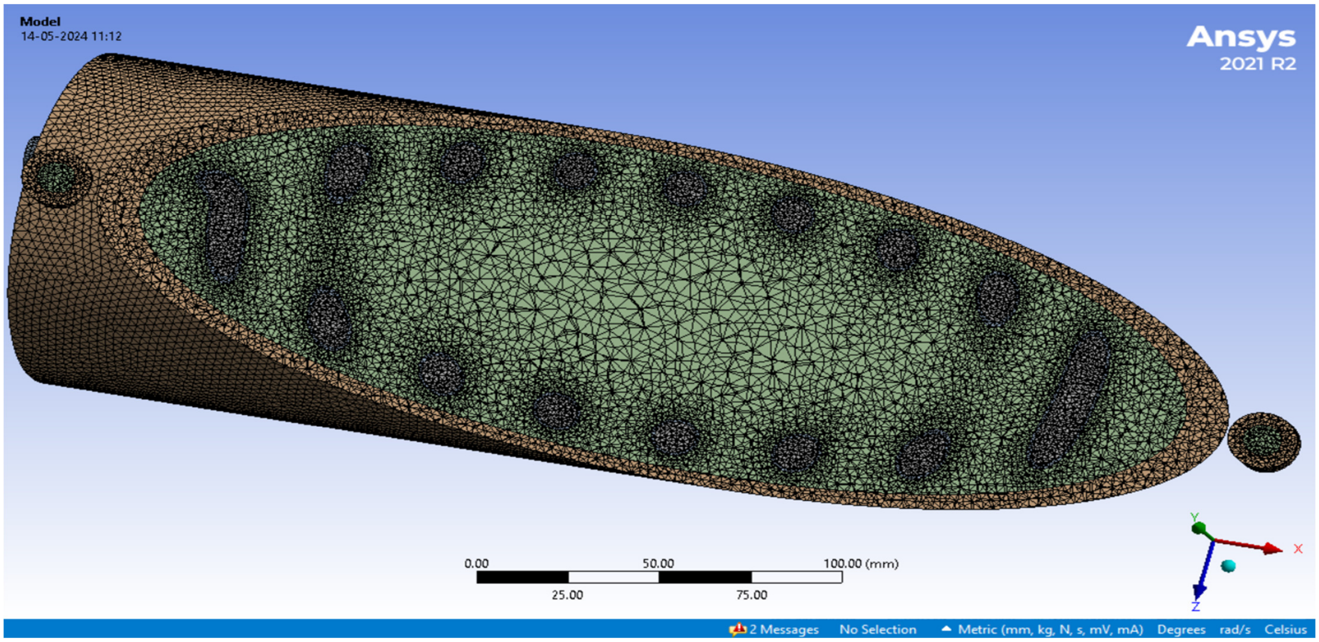


Fig 4.3 (e) Mesh created on the Heat Exchanger

4.3 ANALYSIS

ANSYS Fluent, a powerful computational fluid dynamics (CFD) tool, was used to conduct the research. A pressure-based steady-state solver was employed, which is effective for flows that are either slightly compressible or incompressible. In order to accurately capture the characteristics of the turbulent flow, standard wall functions and the Realizable K-epsilon (2 eqn) turbulence model wereselected.

Appropriate cell zones and boundary conditions were set in order to represent the geometry accurately. The coil walls were considered linked thermal conditions to promote heat transmission between the fluid and solid domains. The outside surface of the shell, assuming no heat loss to the surrounding environment, was defined as an adiabatic surface.

The linked scheme was used for the solution approach, ensuring a robust and efficient convergence. Second-order upwind spatial discretization was applied to the energy, momentum, and turbulent kinetic energy equations, improving the accuracy of the numerical solution. The solution was initialized using the hybrid initialization method, which combines the benefits of standalone and patched initialization procedures.

This approach led to faster convergence and a better first approximation of the flow field. The shell and helical coil heat exchanger (SHCHE) CFD analysis is governed by the fundamental laws of conservation of mass, momentum, and energy. These guidelines are represented mathematically by the Navier-Stokes equations, a series of partial differential equations, along with auxiliary equations for energy transport and turbulence modeling.

The equation for conservation of mass, or continuity equation, can be written as follows:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = S_m$$

The equation is the general form of the mass conservation equation and is valid for incompressible as well as compressible flows. The source S_m is the mass added to the continuous phase from the dispersed second phase (e.g., due to vaporization of liquid droplets) and any user-defined sources.

Conservation of momentum in an inertial (non-accelerating) reference frame is described by

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \nabla \cdot (\rho \vec{v}\vec{v}) = -\nabla p + \nabla \cdot (\overline{\overline{\tau}}) + \rho \vec{g} + \vec{F}$$

where p is the static pressure, $\overline{\overline{\tau}}$ is the stress tensor, and $\rho \vec{g}$ and $\vec{F}$ are the gravitational body force and external body forces (e.g., that arise from interaction with the dispersed phase), respectively. $\vec{F}$ also contains other model-dependent source terms such as porous-media and user-defined sources.

The modeled transport equations for k and ϵ in the realizable k- ϵ model are

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_j}(\rho k u_j)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+G_k+G_b-\rho\epsilon-Y_M+S_k$$

and

$$\frac{\partial}{\partial t}(\rho\epsilon)+\frac{\partial}{\partial x_j}(\rho\epsilon u_j)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\epsilon}\right)\frac{\partial \epsilon}{\partial x_j}\right]+\rho C_1 S\epsilon-\rho C_2\frac{\epsilon^2}{k+\sqrt{\nu\epsilon}}+C_{1\epsilon}\frac{\epsilon}{k}C_{3\epsilon}G_b+S_\epsilon$$

$$C_1=\max\left[0.43,\frac{\eta}{\eta+5}\right],\quad \eta=S\frac{k}{\epsilon},\quad S=\sqrt{2S_{ij}S_{ij}}$$

where

4.4 POST PROCESSING

Analyzing the system's temperature distribution—which is represented by the temperature contours—was the post-processing stage. Furthermore, important performance metrics were computed with the help of the given correlations and equations, including the Nusselt number and heat transfer rate. The unique thermophysical features of hybrid nanofluids were taken into consideration, which improved the study even further.Following the simulations, post-processing was carried out to provide temperature contours, as seen in Figure 4.4.

After simulation, the Nusselt number of the coil and shell was calculated using the correlations given by Ashkan Alimoradi, which are as follows:

$$Nu_c = 0.255Re_c^{0.685}\left(\frac{d_c}{d_{t,i}}\right)^{-0.216}\left(\frac{d_v}{d_{t,i}}\right)^{0.024}\left(\frac{d_{sh}}{d_{t,i}}\right)^{-0.012}\left(\frac{H_c}{d_{t,i}}\right)^{-0.03}\left(\frac{H_{sh}}{d_{t,i}}\right)^{-0.045}\left(\frac{f}{d_{t,i}}\right)^{0.013}\left(\frac{p}{d_{t,i}}\right)^{0.011}Pr_c^{0.315}$$

$$Nu_{sh} = 0.247Re_{sh}^{0.723}\left(\frac{d_c}{d_{t,o}}\right)^{0.378}\left(\frac{d_v}{d_{t,i}}\right)^{0.556}\left(\frac{d_{sh}}{d_{t,i}}\right)^{-0.82}\left(\frac{H_c}{d_{t,i}}\right)^{0.043}\left(\frac{H_{sh}}{d_{t,i}}\right)^{-1.03}\left(\frac{f}{d_{t,i}}\right)^{0.561}\left(\frac{p}{d_{t,i}}\right)^{0.138}Pr_{sh}^{0.717}$$

The heat transfer was then calculated using the equation $Q = \dot{m}_c * Cp_c * (t_{c,i} - t_{c,o})$ for the coil.

Nanofluids were also included in the simulation, and for the nanofluids, the values were obtained from the following equations:

Density: ρ_nf = ϕρ_p + (1-ϕ)ρ_bf

Heat capacity: (ρC_p)_nf = ϕ(ρC_p)_p + (1-ϕ)(ρC_p)_bf

Viscosity: μ_nf = μ_bf (1 - ϕ/ϕ_m)^(-η[ϕ_m])

Thermal Conductivity: k_nf / k_bf = (k_p + (n-1)k_bf - (n-1)ϕ(k_bf-k_p)) / (k_p + (n-1)k_bf + ϕ(k_bf-k_p))

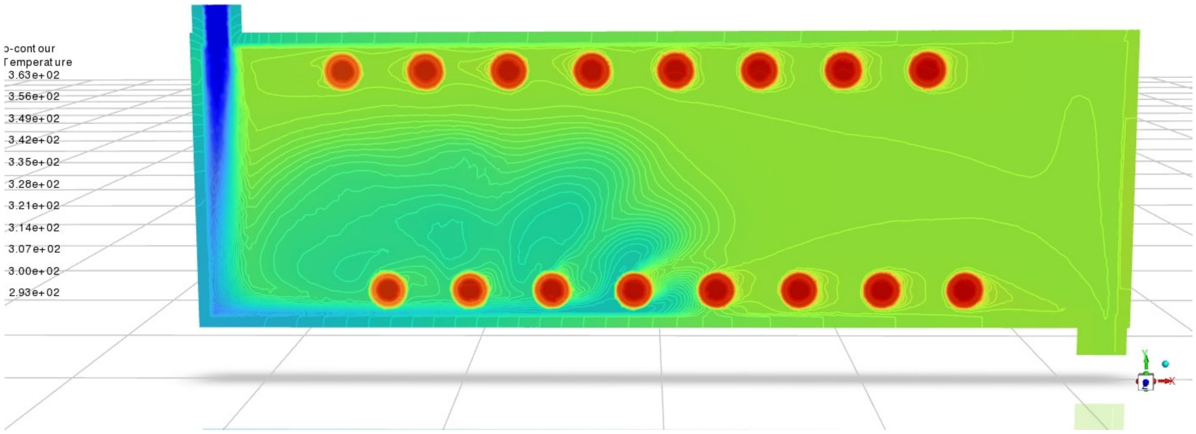


Fig 4.4(a) Temperature contours in XY direction

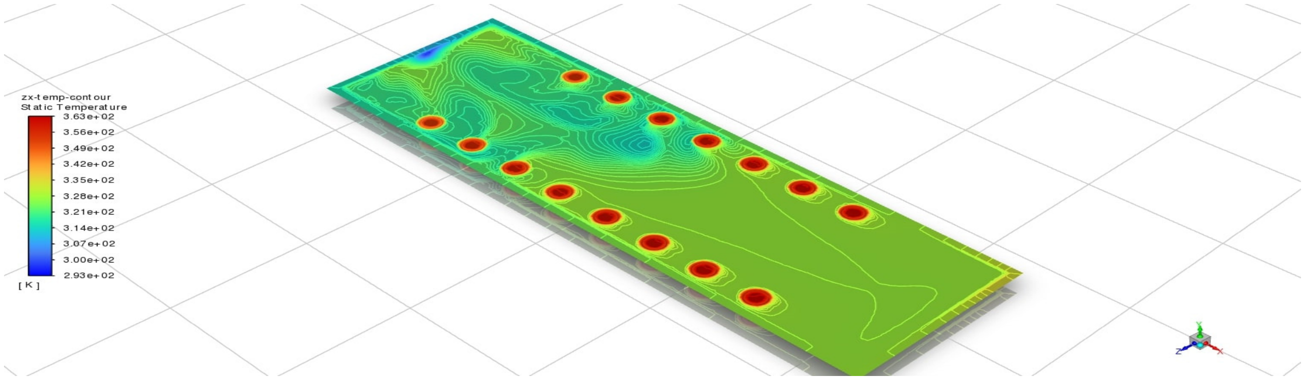


Fig 4.4(b) Temperature contours ZX direction

5.RESULTS AND DISCUSSION

5.1 DESIGN VALIDATION

The Nusselt values from the current work were compared with experimental and numerical data from the literature in order to verify the accuracy of the numerical simulations. Using correlations put forth by Ashkan Alimoradi [1], the Nusselt number—a dimensionless metric that represents the convective heat transfer coefficientwas computed.

The coil-side Nusselt number from the current investigation is compared to the experimental data [1] and numerical data [1] provided by Ashkan Alimoradi in Figure 5.1(a). The numerical model's capacity to accurately forecast the heat transport properties within the helical coil region is confirmed by the findings, which show a good degree of agreement.

In a similar vein, Fig. 5.2(b) contrasts the shell-side Nusselt number from the current investigation with the numerical data [1] and experimental data [1] from Ashkan Alimoradi's study. The near agreement between the outcomes confirms that the numerical simulations accurately represented the complex flow and heat transfer phenomena inside the heat exchanger's shell region.

A strong correlation between the coil-side and shell-side Nusselt values and the numerical method supports the basic assumptions of the simulations. These validation procedures provide as a strong basis for future research and optimizations of shell and helical coil heat exchanger designs, as well as ensuring the validity of the current study.

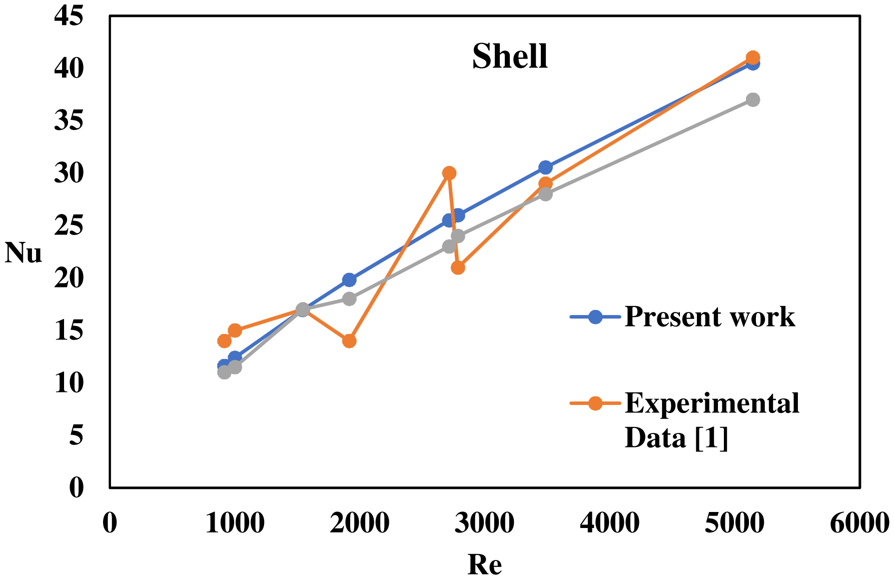


Fig 5.1 (b) Comparison betweeen present work previous work for shell

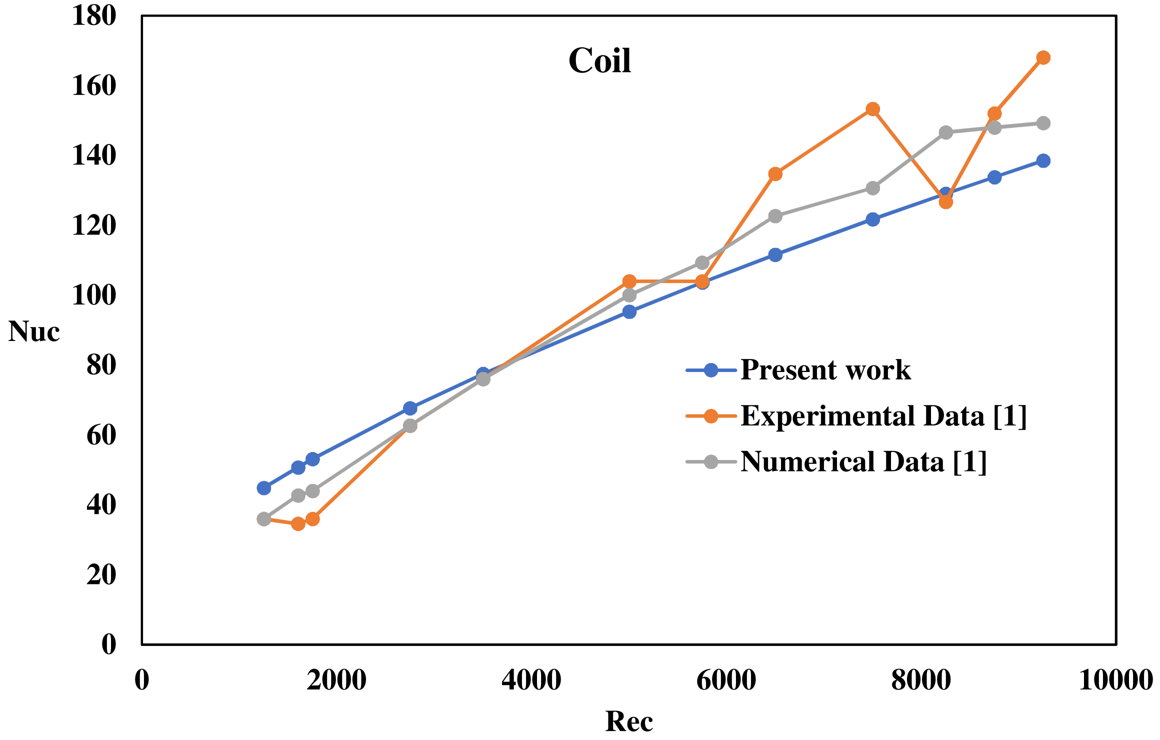


Fig 5.1 (a) Comparison betweeen present work previous work for coil

5.2 MESH INDEPENDENCY STUDY

An extensive mesh independence investigation was carried out in order to guarantee the precision and dependability of the numerical findings. The grid in Figure 4.3 illustrates the evaluation of six distinct element sizes with different cell densities. The study's findings are collected in Table 5.1.

Figure 5.2 also shows the variation in heat transport in the coil with the element sizes used. Table 5.1 shows that there is less than a 1°K variation in the expected outlet fluid temperature between the 3 mm and 2.8 mm element sizes. This slight deviation implies that increasing the mesh refinement beyond 3 mm element size would not greatly increase the accuracy of the findings.

This study led to the decision that the best mesh resolution for the ensuing simulations would be 3 mm for element size. By balancing computing efficiency and solution correctness, this decision ensures that temperature mistakes are reduced without resulting in undue computational expenses. An essential stage in guaranteeing the accuracy and convergence of numerical simulations is the mesh independence investigation. The numerical model can faithfully represent the pertinent flow and heat transfer events while preserving acceptable computation times by choosing the right mesh resolution.

Mesh Element Size (mm)	**10**	6	4	**3.5**	**3**	**2.8**
Mesh Nodes	304059	306360	315781	325545	343737	353296
Mesh Elements	1504156	1513287	1553951	1599662	1682927	1725871
$T_{sh,o}$	327.629	321.015	316.693	314.906	313.833	314.545
$T_{c,o}$	328.021	328.093	328.086	328.101	328.079	328.077

Table – 5.1 Mesh independency study

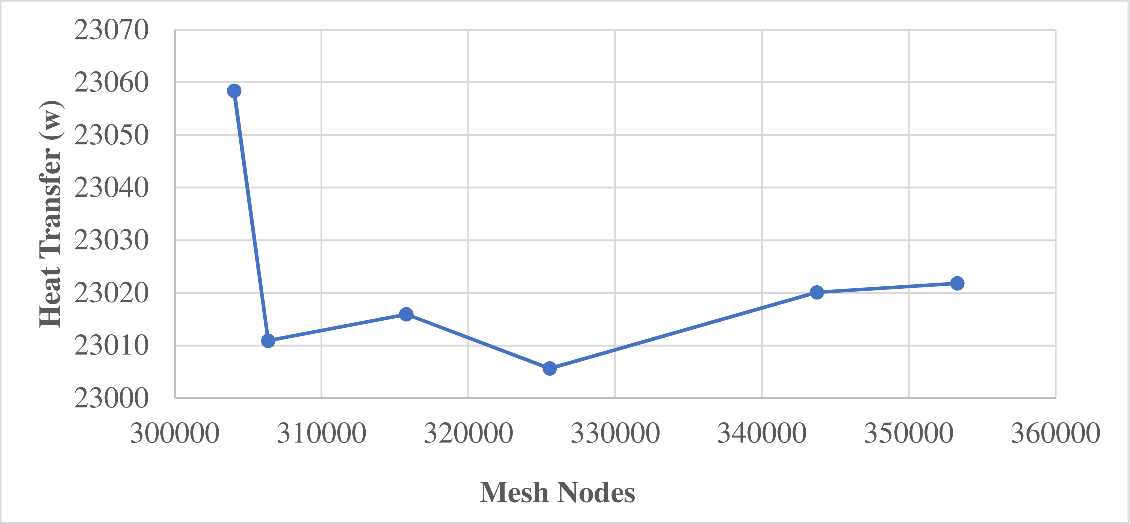


Fig 5.2 Heat transfer Vs Mesh nodes

5.3 COMPARISON BETWEEN TUBE MATERIALS

The link between the inlet velocity and the heat transfer rate for the two materials under investigation—aluminium and Silicon Carbide—is depicted in the graph (Fig. 5.3). The heat transfer rate (qc) in watts is shown on the y-axis, while the inlet velocity (m/s) is shown on the x-axis.

The graph clearly shows that all three materials show a similar trend: as the intake velocity increases, so does the heat transfer rate. This is to be expected since higher intake velocities typically improve mixing and convective heat transfer, which raise heat transfer rates.

The Silicon Carbide-Water composite has the greatest heat transfer rates than aluminium over the range of intake velocities taken into consideration.

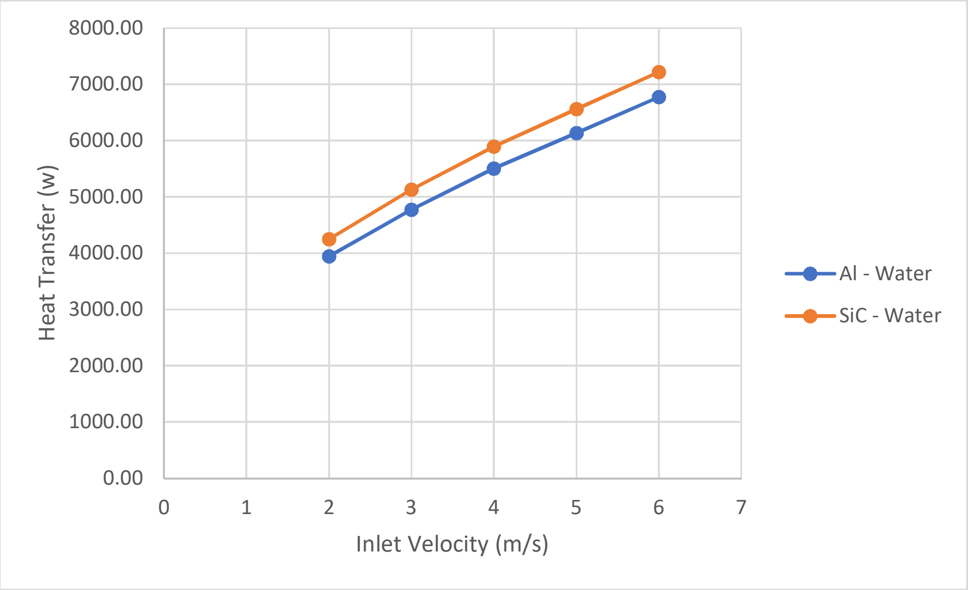


Fig 5.3 Comparison between tube materials

5.4 COMPARISON BETWEEN NANOFLUIDS

In this examination, the impact of different working fluids for Silicon carbide material on a heat exchanger system's heat transfer rate is examined. The link between intake velocity and heat transfer rate (qc) for three distinct working fluids—pure water, Titanium/water nanofluid, and CuO/water nanofluid—is depicted in the graph in Figure 5.4.Heat transfer rates were measured at different input velocities, ranging from 2 m/s to 6 m/s, during the analysis.

.

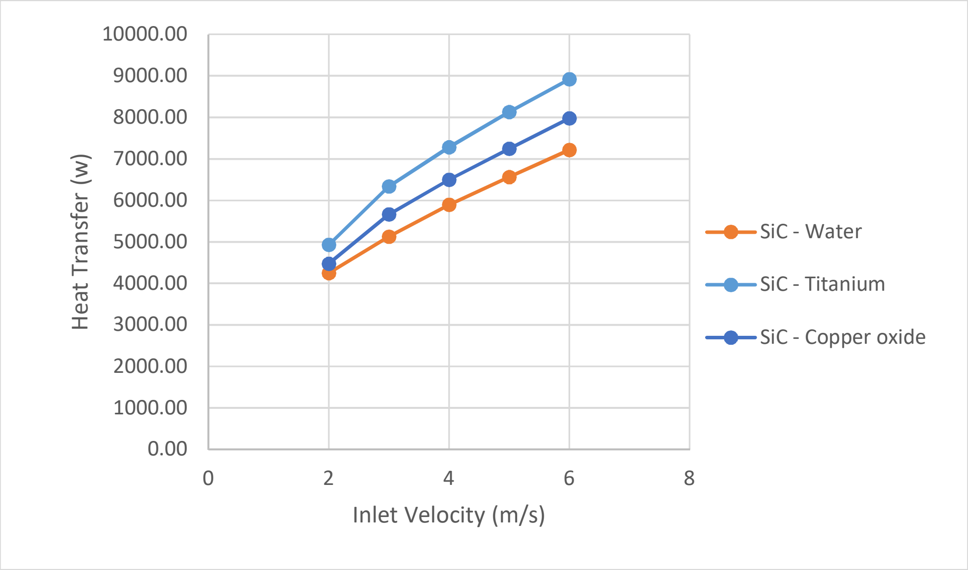


Fig 5.4 Comparison between nanofluids for Silicon Carbide

5.5 HYBRID NANOFLUIDS

In this examination, the impact of different Compositions of hybrid nano fluid for Silicon carbide material on a heat exchanger system's heat transfer rate is examined. The link between intake velocity and heat transfer rate (qc) for three different compositions — 2.5% CuO – 2.5%Ti, 1.25%CuO – 3.75%Ti,3.75%CuO–1.25%TiisdepictedinthegraphinFigure .Heat transfer rates were measured at different input velocities, ranging from2 m/s to 6m/s, during the analysis.

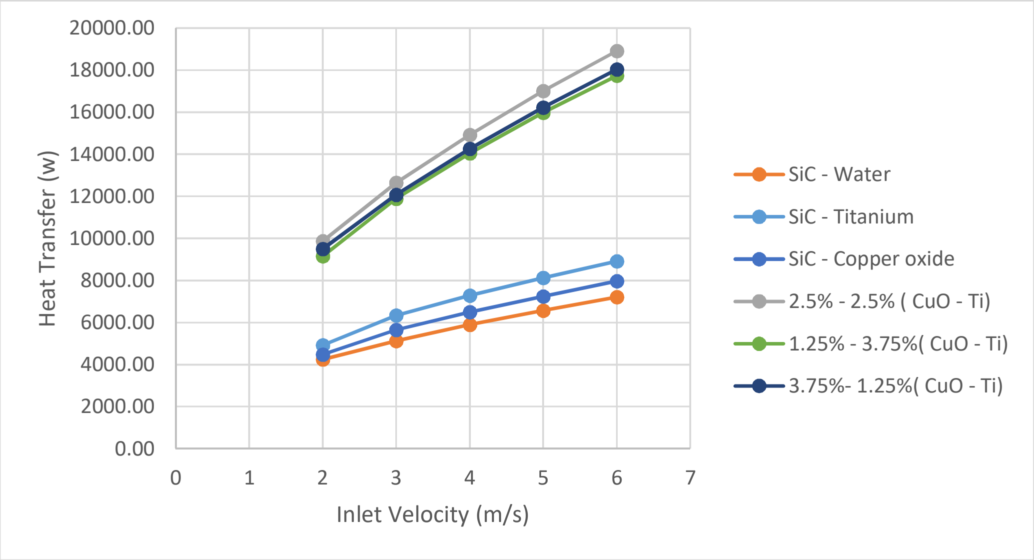


Fig 5.5(a) Comparison between hybrid nano fluids for Silicon Carbide

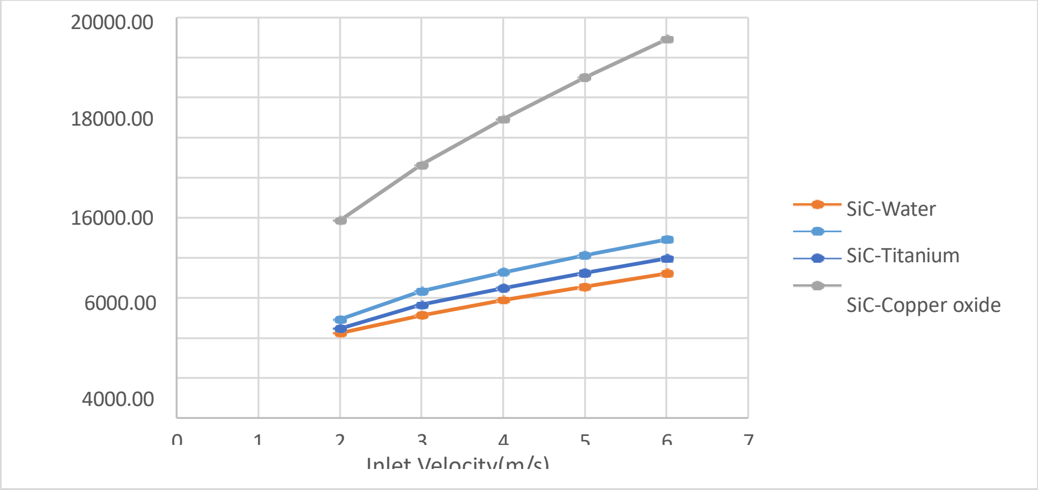


Fig 5.5(b) Comparison between hybrid nano fluids for Silicon Carbide (2.5%CuO-2.5%Ti)

6. CONCLUSIONS & FUTURE SCOPE

6.1 CONCLUSIONS

The numerical study investigated the heat transfer performance of Silicon Carbide and aluminum, as well as different working fluids such as pure water, nanofluids (CuO/water, Titanium/ water) in a helical coil heat exchanger. The key findings are summarized as follows:

- ✓ The Silicon Carbide material demonstrated superior heat transfer capabilities compared to traditional aluminum material, especially at higher inlet velocities. At 6 m/s, the heat transfer rate of the Silicon Carbide water system was 6.5% higher than Aluminum-water system.
- ✓ Among the working fluids tested, Titanium/water nanofluid exhibited the highest heat transfer rates, outperforming pure water and CuO/Water by 23.6%, 11.83% at the maximum inlet velocity of 6 m/s.
- ✓ And also CuO/water outperformed pure water in silicon carbide material by 10.5%.
- ✓ And also CuO-Ti water outperformed the Ti water in Silicon Carbide material by the 78.6%.

6.2 FUTURE SCOPE:

The project's future scope involves experimental validation of the simulated heat exchanger design using nano fluids and hybrid nano fluids for enhanced heat transfer. Further optimization studies can identify the ideal nanofluid composition and operating conditions for maximum heat transfer enhancement while considering practical factors like pressure drop. Scaling up the optimized design for industrial applications, such as power generation or HVAC systems, and conducting economic and environmental analyses are crucial for real-world implementation.

Exploring alternative nanoparticle materials, like carbon nanotubes or graphene, and integrating the optimized design with renewable energy systems like concentrated solar power plants can potentially lead to performance improvements. Investigating fouling and corrosion resistance, as well as developing advanced computational models accounting for nanoparticle behavior, can address long-term durability and performance concerns. Overall, the future scope aims to validate, optimize, and translate the project's findings into practical, sustainable, and efficient heat transfer solutions for various industrial and energy applications.

7. REFERENCES

1. Ashkan Alimoradi, Farzad Veysi, Prediction Of Heat Transfer Coefficients Of Shell And Coiled Tube Heat Exchangers Using Numerical Method And Experimental Validation, International Journal Of Thermal Sciences, Volume 107, 2016, Pages 196-208, Issn 1290-0729,https://doi.org/10.1016/j.ijthermalsci.2016.04.010

2. Khan, A.; Shah, I.; Gul, W.; Khan, T.A.; Ali, Y.; Masood, S.A. Numerical and Experimental Analysis of Shell and Tube Heat Exchanger with Round and Hexagonal Tubes. Energies 2023, 16, 880. https://doi.org/10.3390/en16020880

3. Ahmet Talat İnan, Hasan Köten, Mehmet Akif Kartal, CFD analysis and comparison of conventional type and perforated plate type shell tube heat exchangers, International Journal of Low-Carbon Technologies, Volume 17, 2022, Pages 1280–1291https://doi.org/10.1093/ijlct/ctac033

4. Roohollah Babaei-Mahani, Pouyan Talebizadeh Sardari, Salman Masoudi Soltani, Hayder I. Mohammed, Zbigniew Kuncewicz, Christine Starr, CFD simulation of a shell and multiple tubes condensing heat exchanger in a modified microwave plant applied for reprocessing End-of-Life Tires (ELTs), Environmental Quality Management, 2023,https://doi.org/10.1002/tqem.21971

5. Tiwari, A. K., Pandya, N. S., Javed, S., Oztop, H. F., Said, Z. (2021). Experimental and numerical investigation on the thermal performance of triple tube heat exchanger equipped with different inserts with WO3 nanofluid under turbulent condition1. International Journal of Thermal Sciences, 164,06861. https://doi.org/10.1016/j.ijthermalsci.2021.106861

6. Elsaid, A. M., Ammar, M., Lashin, A., & Assassa, G. M. R. (2023). Performance characteristics of shell and helically coiled tube heat exchanger under different tube cross-sections, inclination angles and nanofluids1. Case Studies in Thermal Engineering, 49, 103239. https://doi.org/10.1016/j.csite.2023.103239

7. Bhattad, A., & Sudhakar Babu, S. (2022). Thermal Analysis of Shell and Tube Type Heat Exchanger using Hybrid Nanofluid1. Trends in Sciences, 19(5), 2890. https://doi.org/10.48048/tis.2022.2890

8. Abed, A. M., Majdi, H. S., Hussein, Z., Fadhil, D., & Abdulkadhim, A. (2018). Numerical analysis of flow and heat transfer enhancement in a horizontal pipe with P-TT and V-Cut twisted tape. Case Studies in Thermal Engineering, 12, 749-758. https://doi.org/10.1016/j.csite.2018.10.004

9. Fetuga, I. A., Olakoyejo, O. T., Abolarin, S. M., Gbegudu, J. K., Onwuegbusi, A., & Adelaja, A. O. (2023). Numerical analysis of thermal performance of waste heat recovery shell and tube heat exchangers on counter-flow with different tube configurations1. Alexandria Engineering Journal, 64(859-875). https://doi.org/10.1016/j.aej.2022.09.017

10. Kartal, M. A., & Feyzioğlu, A. (2023). Numerical analysis of multipurpose shell-tube-heat exchanger with stylized geometry at different baffle gaps and various flow rates1. Case Studies in Thermal Engineering, 52, 103810. https://doi.org/10.1016/j.csite.2023.103810

11. Zolfalizadeh, M., Heris, S. Z., Mohammadpourfard, M., Pourpasha, H., & Meyer, J. P. (2023). Experimental Investigation of the Effect of Graphene/Water Nanofluid on the Heat Transfer of a Shell-and-Tube Heat Exchanger1. International Journal of Energy Research, 2023, Article ID 3477673. https://doi.org/10.1155/2023/3477673

12. Fadhil, N. A., Al-Dabagh, A. M., & Hatem, F. F. (2023). Numerical investigation of heat transfer and pressure drop characteristics in a double pipe heat exchanger with corrugated tubes and rod baffles at various Reynolds numbers. International Journal of Heat and Technology, 41(3), 591-601. https://doi.org/10.18280/ijht.410311

13. Sohrabi, N., Hammoodi, K. A., Hammoud, A., Jasim, D. J., Karouei, S. H. H., Kheyri, J., … & Nabi, H. (2024). Using different geometries on the amount of heat transfer in a shell and tube heat exchanger using the finite volume method1. Case Studies in Thermal Engineering, 55, 104037. https://doi.org/10.1016/j.csite.2024.104037

14. Jalili, P., Kazerani, K., Jalili, B., & Ganji, D. D. (2022). Investigation of thermal analysis and pressure drop in non-continuous helical baffle with different helix angles and hybrid nano-particles1. Case tudiesinThermalEngineering,36,102209.https://doi.org/10.1016/j.csite.2022.102209

15. Cruz, P. A. D., Yamat, E. J. E., Nuqui, J. P. E., & Soriano, A. N. (2022). Computational Fluid Dynamics (CFD) analysis of the heat transfer and fluid flow of copper (II) oxide-water nanofluid in a shell and tube heat exchanger. Digital Chemical Engineering, 3, 100014. https://doi.org/10.1016/j.dche.2022.100014

16. Hassan, Q. H., Afluq, S. G., & Siba, M. A. A. (2021). Numerical Investigation of Heat Transfer in Car Radiation System Using Improved Coolant1. Journal of Advanced Research in Fluid Mechanics and Thermal Sciences, 83(1), 61-69. https://doi.org/10.37934/arfmts.83.1.6169

17. Khaled, R. A., & Mushatet, K. S. (2024). A Computational Study for Evaluating the Performance of Twisted Double Tube Heat Exchangers Fitted with Twisted Tape. CFD Letters, 16(5), 21-36. https://doi.org/10.37934/cfdl.16.5.2136