Numerical simulation of magneto-hydrodynamic natural convection flow within a non-uniformly heated wavy annulus using hybrid-nanofluid

Course Code : AMTH-460
Course Name : Project and Presentation

Submitted by : Md. Masum Miah
Class Roll : FH-020-143
Reg. No. : 2017414056

Supervised by : Dr. Litan Kumar Saha
Professor
Department Of Applied Mathematics
University Of Dhaka

July 10, 2023

A project submitted to the University of Dhaka in partial fulfillment of the requirements for the degree of Bachelor of Science in Applied Mathematics.

Acknowledgements

Professor Dr. Litan Kumar Saha has provided me with essential advice, support, and mentorship throughout my whole project experience, and I would like to extend my sincere gratitude to him. He genuinely inspired me with his unwavering dedication to academic success and their enthusiasm for the subject.

I would also like to express my sincere gratitude to everyone who helped me along the way to completing my project. Their constant backing, guidance, and encouragement have been of immeasurable value to me.

Abstract

The goal of this study is to find solutions to many such fluid flow issues in curve shaped cavities containing a wavy cylinder inside. The model is examined when the bottom wall is partially heated, as well as the upper wall. COMSOL Multiphysics, MATLAB and Techplot were used in this project to develop the model. In this study, an evaluation of the influences of internal heat absorption in a closed cavity is performed. The fluid is considered laminar and the convection is generated because of the buoyancy force only. For this natural convection problem, a set of dimensionless governing equations is disclosed. Then the solution of this problem is generated by using the Galerkin weighted residual method of finite elements. A wide range of dimensionless parameter eg. Reynolds number, Hartmann numbers etc , were introduced for better understanding of the fluid flow. The model is evaluated for different values of these dimensionless parameters to understand the influence of it to the fluid flow. The study concludes that certain parameters alter and influence the flow field, transfer of heat and temperature distribution significantly. The numerical result from COMSOL is then evaluated to be displayed and better analyzed as streamlines and isotherms on a plot. It is also seen from the model that the presence of heat generation and absorption significantly impact the process of fluid flow and heat transfer. Furthermore, another inspection of the dimensionless parameter Rayleigh number (Ra) is observed for the streamlines and the isotherms changes in our model.

Contents

List of Figures

List of Abbreviations

Numenclature

C_p : Specific heat of the nanofluid at constant pressure

C_{pp} : Specific heat of the nanoparticles at constant pressure

k_{bf} : Thermal conductivity of base fluid

k_{hnf} : Thermal conductivity of hybrid nanofluid

k_{hp} : Thermal conductivity of hybrid nano-particles

Nu_h, Nu_c : Nusselt numbers

Pr : Prandtl number

ϵ : Small parameter

μ_{bf} : Dynamic viscosity of base fluid

μ_{hnf} : Dynamic viscosity of dynamic nanofluid

ρ : Density

ρ_{hnf} : Density of hybrid nanofluids

σ_{hp} : Electrical conductivity of base fluid

σ_{hnf} : Electrical conductivity of Hybrid nanofluid

σ_{bf} : Electrical conductivity of base fluid

θ : Dimensionless temperature

ϕ : Stream function

γ : Electrical conductivity ratio

k_f : Thermal conductivity of water

Nu_h^m, Nu_c^m : Mean Nusselt numbers

Ra : Rayleigh number

U_d, V_d : Dimensionless velocity components of the dusty phase

λ : Dimensionless amplitude of the lower wall

τ : Dimensionless time

ω : Vorticity of the nanofluid phase

α_{hnf} : Thermal diffusivity of the nanofluid

α_f : Thermal diffusivity of water

β : Nanofluids' thermal expansion coefficient

β_f : Thermal expansion coefficient of water

γ : Specific heat ratio

θ : Dimensionless temperature of the nanofluid phase

ξ, η : Transformed coordinates

ρ : Nanofluid density

ρ_f : Water density

ρ_p : Nanoparticles' density

ν : Kinematic viscosity of the nanofluid

ν_f : Kinematic viscosity of water

ϕ : Nanoparticles' volume fraction

ψ : Stream function of the nanofluid phase

1

Introduction

The study of fluid flow and heat transfer around complex geometries is of great interest in fluid dynamics research. The wavy cylinder and curve-shaped cavity represent geometries that deviate from traditional straight configurations. By examining the natural convection behavior in these complex geometries, researchers can gain a deeper understanding of fluid flow phenomena, such as vortex shedding, boundary layer development, and flow separation (T. Tayebi, 2017) . Such insights can contribute to the advancement of fluid dynamics knowledge and help refine existing theories and models. (Cho et al., 2013)

1.1 | Motivation

Understanding the Magnetohydrodynamic (MHD) natural convection of a hybrid nanofluid can lead to improved heat transfer performance. Hybrid nanofluids, which consist of a base fluid and nanoparticles, have shown promising enhancements in thermal conductivity. By investigating their behavior around a wavy cylinder in a curve-shaped cavity, researchers can explore the potential for further enhancing heat transfer rates, which has significant implications for various industries, such as electronics cooling, energy systems, and thermal management. (Mehryan et al., 2019)

Effective cooling is crucial for many engineering applications, including electronic devices, power plants, and aerospace systems. Investigating the natural convection of a hybrid nanofluid from a wavy cylinder in a curve-shaped cavity can provide valuable insights into the design and optimization of cooling systems. By understanding the fluid flow patterns, heat transfer characteristics, and thermal performance of such configurations, researchers can develop improved cooling strategies that maximize efficiency and minimize energy consumption. (Selimefendigil and Öztop, 2020)

Nanofluids have attracted considerable attention due to their potential in various technological applications. By investigating the behavior of hybrid nanofluids in a curved cavity with a wavy cylinder, researchers can explore the applicability of these fluids in diverse fields. This research can lead to the development of novel cooling systems, energy-efficient heat exchangers, and advanced thermal management techniques. Additionally, the findings can provide valuable data for numerical simulations and validation of computational models used in the design and optimization of nanofluid-based systems. (Jamil and Ali, 2020)

Renewable energy technologies, such as solar thermal systems and geothermal heat extraction, often rely on heat transfer processes (El Haj Assad et al., 2021) . Enhancing the heat transfer performance of these systems can significantly improve their overall efficiency and contribute to a more sustainable energy future (Osintsev et al., 2023) . Investigating the natural convection of a hybrid nanofluid from a wavy cylinder in a curve-shaped cavity can offer insights into optimizing heat transfer within renewable energy systems, leading to improved energy conversion and utilization.

1.2 | Aims and Objectives

The primary aim of this research is to understand the fluid flow and heat transfer characteristics associated with the natural convection of a hybrid nanofluid around a wavy cylinder in a curve-shaped cavity. This includes studying the velocity profiles, temperature distribution, and convective heat transfer coefficients within the system. The objective is to obtain comprehensive knowledge about the flow patterns and heat transfer mechanisms involved in order to optimize the thermal performance.

Another objective is to assess the impact of the wavy cylinder geometry on the natural convection of the hybrid nanofluid. By varying the parameters related to the wavy cylinder, such as wavelength, amplitude, and number of waves, the research aims to investigate how these geometric factors affect the fluid flow patterns, thermal boundary layer development, and convective heat transfer. Understanding the role of the wavy cylinder geometry will contribute to the design and optimization of efficient heat transfer systems. (Nakhchi, 2019)

The research aims to evaluate the thermal performance of the hybrid nanofluid under natural convection conditions. By incorporating nanoparticles into the base fluid, the objective is to assess the influence of nanoparticle concentration, size, and material on the heat transfer enhancement. Comparisons will be made between the hybrid nanofluid and the base fluid to quantify the improvements achieved in terms of convective heat transfer. This evaluation will provide insights into the potential benefits of using hybrid nanofluids in thermal management applications.

The research aims to validate numerical models and Computational fluid dynamics (CFD) simulations through experimental data. By conducting experimental measurements and comparing them with the simulation results, the objective is to verify the accuracy and reliability of the numerical models used to predict the fluid flow and heat transfer behavior. Validated models can then be employed to further explore the system's performance, conduct sensitivity analyses, and provide predictive tools for practical applications.

The ultimate objective of this research is to contribute to the existing knowledge in the field of natural convection and nanofluid heat transfer. By generating new insights, experimental data, and validated models, the research aims to advance the understanding of complex fluid flow phenomena, provide guidelines for efficient thermal management, and support the development of innovative applications in various industries.

In summary, the aims and objectives of the research on the natural convection of a hybrid nanofluid from a wavy cylinder placed in a curve-shaped cavity include investigating fluid flow and heat transfer characteristics, exploring the influence of wavy cylinder geometry, evaluating nanofluid performance, optimizing heat transfer efficiency, validating numerical models, and contributing to the existing knowledge in the field.

1.3 | Document Structure

This project report is partitioned into 5 chapters. A brief introduction to the document structure is given bellow:

Chapter 1 : This chapter provides an overview and sets the context for the study. It serves the introduction of the topic, establishes the research problem or gap, states the objectives and research questions, and outline the structure of the document.

Chapter 2 : This chapter explains some fundamental fluid mechanics concepts, with definitions and parameters who understanding is required for the study.

Chapter 3 : This chapter losses over the physical, mathematical model, dimensionless mathematical model with boundary conditions and heat transfer equation.

Chapter 4 : This chapter introduces the software used to perform the computation and simulation. It also provides a brief discussion of the methodology used.

Chapter 5 : This chapter discusses about the the and compare it with known source to varify.

Chapter 6 : This chapter concludes the overall study.

2

Background & Literature Review

This chapter explains some fundamental fluid mechanics concepts, with defi- nitions and parameters who understanding is required for the study.

2.1 | Introduction

Fluid Dynamics is a branch of applied mathematics that specializes in the study of how liquids and gases move within different environments. In terms of mathematical representation, fluid motions are expressed through Partial Differential Equations (PDEs). By exploring different variables within these equations, it becomes feasible to establish a mathematical model that describes the motion of fluids. This process enables researchers and readers to gain a deeper comprehension of the intricate dynamics involved in fluid motion. (Wikipedia)

2.2 | Fluid Motion

Fluid motion can be approached through two methods: the Statistical method and the Molecular method. In the Molecular method, fluid motion is viewed as the collective movement of molecules following the laws of dynamics and Newtonian mechanics. This approach allows for the approximation of three essential equations: the conservation of mass equation, momentum equation, and energy equation. It also considers factors such as viscosity, thermal conductivity, and the influence of forces acting on boundary surfaces and pressure points at corners.

Alternatively, the Continuum method can be employed, which treats fluids as continuous substances rather than individual molecules. It is worth noting that the conservation of mass equation, momentum equation, and energy equation remain crucial in constructing a fluid model using this method. (Oreilly)

The Statistical method is utilized when the Continuum method fails to pass the validity test. While the Statistical method is a sophisticated approach for solving differential equations related to fluid motion, it is not suitable for liquids and dense gases. (Werder et al., 2005)

Furthermore, fluid motion can be categorized based on space and time into two additional types: the Lagrangian method and the Eulerian method. (Werder et al., 2005)

2.3 | Definations & Equations

Fluid dynamics involves the study of three fundamental lines that play a crucial role in understanding fluid motion: streamline, pathline, and streakline. These lines provide valuable insights into the behavior and characteristics of fluids in motion, aiding scientists and engineers in predicting fluid flow patterns across various applications.(LiveScience)

Streamline: A streamline is an imaginary line that represents the instantaneous direction of fluid flow at any given point. It visually illustrates the flow patterns by aligning with the tangent of the velocity vector at each point. If a tiny particle were introduced into the fluid, it would follow the streamline, moving in harmony with the flow. Streamlines do not intersect, allowing for the analysis of flow direction, shape, and circulation in different scenarios. (Shaughnessy et al., 2005)

$$\frac{dx}{u} = \frac{dy}{v} = \frac{dz}{w} \tag{2.1}$$

Here, $\frac{dx}{u}$, $\frac{dy}{v}$, and $\frac{dz}{w}$ represent the fractions for the infinitesimal displacements in the x, y, and z directions, respectively. u, v, and w represent the velocity components in the x, y, and z directions, respectively.

Pathline: A pathline traces the actual trajectory followed by an individual fluid particle as it moves through the flow over time. It captures the complete history of the particle's motion, starting from its initial position and extending to its current position. Pathlines are time-dependent and can be visualized as the actual path a dye or tracer particle would take within the fluid (Johnson, 2016). By studying pathlines, researchers gain insights into the movement and interactions of individual fluid particles within the flow field.

$$\boldsymbol{x} = \boldsymbol{x}_0 + \int_{t_0}^{t} \boldsymbol{V} \tag{2.2}$$

$(\boldsymbol{x}, t)\, dt$ In this equation, $\boldsymbol{x}$ represents the pathline, $\boldsymbol{x}_0$ is the initial position, $\boldsymbol{V}(\boldsymbol{x}, t)$ is the velocity field as a function of position $\boldsymbol{x}$ and time t, and the integral represents the integration of the velocity field over time from t_0 (initial time) to t (current time).

Streakline: A streakline is formed by connecting fluid particles that have passed through a specific point in the flow field over a defined period. It represents the continuous line formed by particles that have crossed the point at different instances. Streaklines offer information on the temporal evolution of fluid flow, showcasing how fluid particles are transported through the flow field. Analyzing streaklines enables researchers to examine the mixing and spreading of fluid elements within the flow.

$$\boldsymbol{x}(t_0, t) = \int_{t_0}^{t} \boldsymbol{V}(\boldsymbol{x}_0, \tau)\, d\tau \tag{2.3}$$

In this code, $\boldsymbol{x}(t_0, t)$ represents the streakline connecting fluid particles that have passed through a specific point at time t_0 and time t. The integral represents the integration of the velocity field $\boldsymbol{V}$ as a function of the initial position $\boldsymbol{x}_0$ and the intermediate time variable τ from t_0 to t.

By understanding and analyzing these three lines, scientists and engineers gain a deeper understanding of the dynamics and behavior of fluid motion. This knowledge proves invaluable in applications such as aerodynamics, hydrodynamics, and fluid transport systems, where accurate predictions and comprehension of fluid flow patterns are essential. (Batchelor et al., 2003)

2.4 | Various Types of Fluid Flow

Steady & Unsteady Flows: When the flow quantity remains constant over time, it is referred to as steady flow. On the other hand, if the flow characteristics vary with time, it is termed as unsteady flow.

Uniform & Non-Uniform Flow: In uniform flow, the cross-sectional area of the fluid remains constant during its movement. Conversely, in non-uniform flow, the cross-sectional area changes as the fluid progresses through the conduit.

Rotational & Irrotational Flow: Rotational flow occurs when fluid particles rotate around their axis during the flow. Conversely, irrotational flow refers to fluid motion where there is no rotation of fluid particles around their axis.

Laminar, Transitional & Turbulent Flow: Laminar flow occurs when fluid moves smoothly in distinct layers without significant mixing. This type of flow is characterized by low Reynolds numbers (less than 2000) and shear stress controls the velocity distribution within the layers. Transitional flow refers to the flow that transitions between laminar and turbulent flow regimes. It is observed when the Reynolds number and viscous stress are numerically comparable (Reynolds numbers ranging from 2000 to 4000). Turbulent flow is characterized by chaotic, zig-zag motion of fluid particles resulting in the formation of eddies. It is commonly observed in rivers, lakes, and winds. Turbulent flow occurs when the Reynolds number exceeds 4000 and exhibits higher shear stress compared to laminar flow.

Compressible & Incompressible Flow: Incompressible flow refers to fluid motion in which the density remains constant throughout the fluid domain. Conversely, compressible flow occurs when the density of the fluid varies from one point to another. Most fluids, such as liquids, are considered incompressible as their density is minimally affected by changes in pressure. However, gases are highly compressible due to significant density changes resulting from pressure variations. (Saif)

2.5 | Various Types of Convection

Convection is the transfer of heat that occurs in fluids (liquids and gases) due to the movement of the fluid itself. There are three primary types of convection:

1. Natural Convection : It happens when fluid motion is driven by buoyancy forces resulting from density variations caused by temperature differences. (Roy et al., 2021) When a fluid is heated, it expands and becomes less dense, causing it to rise. As the heated fluid ascends, cooler fluid from the surroundings takes its place, creating a continuous circulation pattern. This type of convection is commonly observed in everyday scenarios, such as the rising of hot air or the circulation of water in a boiling pot.(Parvin et al., 2023)

2. Forced Convection: This type of convection occurs when fluid motion is induced by an external force, such as a fan, pump, or mechanical device. Forced convection involves making the fluid move or flow over a surface or within a conduit at a higher rate than it would under natural convection. The forced motion enhances convective heat transfer by increasing the contact between the fluid and the surface. Examples of forced convection include forced air cooling in electronic devices and the circulation of coolant in a car's radiator.(Shah and London, 1978)

3. Mixed Convection: Mixed convection combines both natural and forced convection mechanisms. In this case, fluid motion is influenced by both buoyancy-driven flow and externally induced flow. The relative importance of natural and forced convection varies depending on the specific circumstances. Mixed convection is often observed in situations where both natural and forced convection effects are present, such as in a heated room with a ceiling fan.(Merkin et al., 2022)

These distinct types of convection play a significant role in various natural and engineered systems, affecting heat transfer rates and fluid behavior. Understanding these convection mechanisms is crucial in fields such as engineering, meteorology, and fluid dynamics.

2.6 | Dimensionless Parameters

Dimensionless parameters are quantities that result from ratios or combinations of physical variables, eliminating specific units and yielding dimensionless values. These parameters are used to express relationships between variables independent of unit measurements, allowing for comparisons and analysis across different scales and systems. (PSU)

Dimensionless parameters have significant importance in scientific and engineering fields for Simplification and Generalization, Scalability and Transferability, Classification and Analysis, Design and Optimization and Standardization.

Overall, dimensionless parameters are essential tools in scientific and engineering disciplines. They provide a systematic and abstract representation of physical phenomena, simplifying analysis, aiding in the understanding of system behavior, and facilitating knowledge transfer across different scales and systems.

2.6.1 | Raynold Number

The Reynolds number is a dimensionless parameter used in fluid dynamics to characterize the flow regime of a fluid. It is named after Osborne Reynolds, a British engineer and physicist. (Wang et al., 2004) The Reynolds number is defined as the ratio of inertial forces to viscous forces within a fluid flow, given by the equation:

$$Re = \frac{\rho \cdot v \cdot L}{\mu}$$

Where:

- Re is the Reynolds number

- ρ is the density of the fluid
- v is the velocity of the fluid
- L is a characteristic length scale of the flow
- μ is the dynamic viscosity of the fluid

The Reynolds number provides information about the relative significance of inertial forces, determined by the fluid's velocity and density, compared to viscous forces, determined by the fluid's viscosity. Based on its value, the flow regime can be classified into three categories:

1. **Laminar Flow :** $Re < 2000$
2. **Transitional Flow :** $2000 < Re < 4000$
3. **Turbulent Flow :** $Re > 4000$

The Reynolds number is a valuable tool for understanding and predicting fluid flow behavior in various applications, such as pipe flow, aerodynamics, and heat transfer. It helps determine the appropriate equations, models, and design considerations based on the expected flow regime.

2.6.2 | Rayleigh Number

The Rayleigh number is a dimensionless parameter used in fluid dynamics and heat transfer to characterize the relative importance of buoyancy forces to viscous forces within a fluid. It represents the ratio of buoyancy forces to viscous forces and determines the behavior of convective heat transfer in a fluid. It is named after Lord Rayleigh, a British scientist. The Rayleigh number is defined as:

$$R_a = \frac{(g \cdot \beta \cdot \Delta T \cdot L^3)}{\nu \cdot \alpha}$$

Where:

- Ra is the Rayleigh number,
- g is the acceleration due to gravity,
- β is the coefficient of thermal expansion,
- ΔT is the temperature difference,
- L is a characteristic length scale of the flow,
- ν is the kinematic viscosity of the fluid,
- α is the thermal diffusivity of the fluid.

Based on the value of Rayleigh Number, the flow regime can be classified into three main categories:

1. Stable Regime ($Ra < Ra_c$): In the stable regime, the buoyancy forces are not strong enough to cause significant fluid motion. Heat transfer occurs mainly through conduction, and convection is suppressed. The fluid remains relatively quiescent and stratified.

2. Transitional Regime ($Ra_c < Ra < Ra_t$): In the transitional regime, the buoyancy forces start to overcome the viscous forces, leading to the initiation of convective motion. The flow becomes unstable, and convective heat transfer begins to dominate. The fluid experiences intermittent motion and the formation of thermal plumes or cells.

3. Unstable Regime ($Ra > Ra_t$): In the unstable regime, the buoyancy forces greatly exceed the viscous forces, resulting in vigorous convective motion. The flow becomes fully turbulent, characterized by the continuous formation and interaction of thermal plumes or cells. Heat transfer occurs predominantly through convective mixing.

2.6.3 | Prandtl Number

The Prandtl number is a dimensionless parameter utilized in fluid dynamics and heat transfer to quantify the relative significance of momentum diffusivity to thermal diffusivity in a fluid medium. It describes the relationship between the rates of momentum transfer and thermal transfer within the fluid It is named after Ludwig Prandtl, a German engineer and physicist. The Prandtl number, denoted as Pr, is defined as the ratio of the kinematic viscosity (ν) to the thermal diffusivity (α) of the fluid:

$$Pr = \frac{\nu}{\alpha}$$

Where,

- represents the kinematic viscosity
- represents the thermal diffusivity of the fluid

Based on its value, the Prandtl number can be classified into three primary categories:

1. Prandtl Number < 1 : When the Prandtl number is less than 1, it signifies that thermal diffusivity is significantly higher compared to momentum diffusivity. This is commonly observed in fluids with high thermal conductivity and low viscosity, such as gases. In such cases, heat conducts rapidly compared to the fluid motion, resulting in a thin thermal boundary layer and efficient heat transfer.

2. Prandtl Number $\approx$ 1 : When the Prandtl number is approximately 1, it indicates that momentum diffusivity and thermal diffusivity are of similar magnitudes. This is often observed in fluids with moderate thermal conductivity, such as water. In such scenarios, the transport of momentum and heat occurs at comparable rates, and the thicknesses of the thermal and velocity boundary layers are similar.

3. Prandtl Number > 1 : When the Prandtl number is greater than 1, it suggests that momentum diffusivity is significantly higher than thermal diffusivity. This is typical in fluids with low thermal conductivity and high viscosity, such as oils. In such situations, momentum is transported more rapidly than heat, resulting in a thicker thermal boundary layer and slower heat transfer.

2.6.4 | Froude Number

The Froude number is a dimensionless parameter utilized in fluid dynamics to quantify the relative importance of inertial forces compared to gravitational forces within a flow. epresents the ratio of inertial forces (associated with flow velocity) to gravitational forces within the flow. It is named after William Froude, a British engineer and naval architect. The Froude number is defined as:

$$Fr = \frac{V}{g \cdot L}$$

where:

- Fr is the Froude number,
- V is the velocity of the flow,
- g is the acceleration due to gravity,
- L is a characteristic length scale of the flow.

Based on its value, the Froude number can be classified into three primary categories:

1. Fr < 1 : When the Froude number is less than 1, it indicates subcritical flow or a subcritical Froude number. In this regime, gravitational forces dominate, and the flow behavior is influenced by obstacles or boundaries. Subcritical flow is characterized by smooth, uniform flow patterns, and the flow velocity is typically lower than the wave speed.

2. Fr $\approx$ 1 : When the Froude number is approximately 1, it signifies critical flow or a critical Froude number. Critical flow occurs when the flow velocity is roughly equal to the wave speed. In this regime, the flow transitions between subcritical and supercritical flow, with a balance between inertial and gravitational forces. Critical flow is associated with specific conditions, such as flow over a weir or flow through a hydraulic jump.

3. Fr > 1 : When the Froude number is greater than 1, it indicates supercritical flow or a supercritical Froude number. In this regime, inertial forces dominate, and gravitational forces have less influence. Supercritical flow is characterized by rapid flow velocities, the formation of hydraulic jumps, and upstream wave propagation. The flow patterns in this regime are highly dynamic, exhibiting significant turbulence and mixing.

2.6.5 | Grashof Number

The Grashof number is a dimensionless parameter employed in fluid dynamics and heat transfer to assess the relative influence of buoyancy forces compared to viscous forces within a fluid. It indicates the ratio of buoyancy forces (associated with temperature differences and gravity) to viscous forces within the fluid. It is named after Franz Grashof, a German engineer. The Grashof number, denoted as Gr, is determined by the equation:

$$Gr = \frac{g \cdot L^3 \cdot \beta \cdot \Delta T}{\nu^2}$$

Where:

- Gr represents the Grashof number
- g denotes the acceleration due to gravity
- L corresponds to a characteristic length scale of the flow
- β stands for the coefficient of thermal expansion
- ΔT signifies the temperature difference,
- ν denotes the kinematic viscosity of the fluid.

Depending on the value of Grashof number, the flow regime can be classified into three primary categories:

1. Gr $\ll$ 1 : When the Grashof number is significantly smaller than 1, it indicates a laminar flow regime. In this regime, viscous forces dominate over buoyancy forces, resulting in smooth and organized flow with minimal mixing and heat transfer. This situation arises when the fluid has either low temperature differences or high viscosity.

1. Gr $\approx$ 1 : When the Grashof number is approximately equal to 1, it signifies a transitional flow regime. In this regime, buoyancy forces and viscous forces have similar magnitudes. The flow behavior is intermediate between laminar and turbulent, and the transition to turbulence becomes more significant as the Grashof number increases.

1. Gr $\gg$ 1 : When the Grashof number is substantially greater than 1, it indicates a turbulent flow regime. In this regime, buoyancy forces dominate over viscous forces. The flow becomes highly turbulent, characterized by intense mixing and heat transfer. Turbulent flow occurs when the fluid experiences large temperature differences or has low viscosity.

2.6.6 | Nusselt Number

The Nusselt number, denoted as Nu, is a dimensionless parameter used in heat transfer to quantify the convective heat transfer rate between a surface and a fluid. It provides valuable information about the relative contribution of convective heat transfer to conductive heat transfer across a surface. It is named after Wilhelm Nusselt, a German engineer renowned for his contributions to the field.

$$Nu = \frac{hL}{k}$$

Based on the value of Nusselt Number, the following situations occur in heat transfer.

1. Nu < 1: A Nusselt number smaller than 1 indicates that convective heat transfer is relatively less significant compared to conductive heat transfer. This typically occurs in situations where heat transfer is primarily governed by conduction, such as with highly conductive fluids or low flow velocities.

2. Nu $\approx$ 1: When the Nusselt number is approximately equal to 1, it signifies a transitional heat transfer regime. In this regime, convective and conductive heat transfer mechanisms have comparable magnitudes. The heat transfer behavior lies between purely conductive and purely convective heat transfer phenomena.

3. Nu > 1: A Nusselt number greater than 1 indicates that convective heat transfer dominates over conductive heat transfer. This is common in scenarios where convection plays a dominant role, such as in forced convection or natural convection processes.

2.6.7 | Hartmann Number

The Hartmann number, denoted as Ha, is a dimensionless parameter used in magnetohydrodynamics (MHD) to quantify the relative significance of magnetic forces to viscous forces within a conducting fluid. It represents the ratio of magnetic forces to viscous forces in a conducting fluid subjected to a magnetic field It is named after Julius Hartmann, a German physicist. The Hartmann number is defined as:

$$Ha = B \cdot L \cdot \sqrt{\sigma / \mu}$$

Where:

- Ha is the Hartmann number,
- B is the magnetic field strength,
- L is a characteristic length scale of the flow,
- σ is the electrical conductivity of the fluid,
- μ is the dynamic viscosity of the fluid.

Depending on the value of Hartmann Number, have the following situation:

1. Ha $\ll$ 1: When the Hartmann number is much smaller than 1, it indicates a weak magnetic field or low electrical conductivity. In this regime, viscous forces dominate over magnetic forces, and the fluid flow behavior is primarily governed by viscous effects.

2. Ha $\approx$ 1: When the Hartmann number is approximately equal to 1, it signifies a transitional regime. In this regime, the magnitudes of magnetic forces and viscous forces are comparable. The flow behavior lies between purely viscous flow and flow dominated by magnetic effects.

3. Ha $\gg$ 1: When the Hartmann number is much larger than 1, it indicates a strong magnetic field or high electrical conductivity. In this regime, magnetic forces dominate over viscous forces, significantly influencing the fluid flow behavior. The fluid flow becomes more constrained and aligned with the magnetic field lines.

2.6.8 | Richardson Number

The Richardson number, named after Lewis Fry Richardson, is a dimensionless parameter used in fluid dynamics to assess the significance of buoyancy forces relative to shear forces within a flow. It represents the ratio between buoyancy forces and shear forces in fluid flow Denoted as Ri, it is defined as:

$$Ri = \frac{g \cdot \Delta\rho \cdot h}{\rho \cdot u^2}$$

Here:

- Ri represents the Richardson number.
- g is the acceleration due to gravity.
- $\Delta\rho$ is the density difference across the flow.
- h is a characteristic length scale of the flow.
- ρ is the density of the fluid.
- u is the velocity gradient (shear) within the flow.

Based on the value of Richardson Number, the flow can be categorized into different regimes:

1. Ri < 0: When the Richardson number is less than 0, it indicates an unstable flow regime. In such cases, buoyancy forces dominate over shear forces, resulting in the formation of buoyancy-driven instabilities like Kelvin-Helmholtz instabilities or Rayleigh-Bénard convection. This regime is characterized by significant mixing and turbulence.

2. Ri $\approx$ 0: When the Richardson number is approximately equal to 0, it signifies a neutral flow regime. Here, buoyancy forces and shear forces are of similar magnitudes. Flow behavior is transitional, and the development of instabilities is influenced by subtle changes in flow conditions.

3. Ri > 0: When the Richardson number is greater than 0, it indicates a stable flow regime. In this regime, shear forces dominate over buoyancy forces, promoting laminar flow and hindering the formation of instabilities. The flow is more organized and less susceptible to mixing and turbulence.

2.6.9 | Pressure Coefficient

The pressure coefficient, denoted as C_p, is a dimensionless parameter extensively employed in fluid dynamics to characterize the pressure distribution around a body or surface within a flow field. It is defined as the ratio of the pressure deviation from a reference pressure to the dynamic pressure of the flow, given by the equation:

$$C_p = \frac{P - P_ref}{0.5 \cdot \rho \cdot V^2}$$

Were:

- Cp represents the pressure coefficient,
- P denotes the pressure at a specific point on the body or surface
- Pref is the reference pressure (typically the freestream or ambient pressure)
- corresponds to the density of the fluid, and
- V represents the velocity of the flow.

1. C_p **< 0:** A negative pressure coefficient indicates a region of low pressure relative to the reference pressure. Such occurrences are typically observed on the surface of an object where the flow is accelerated, such as the upper surface of an airfoil. Negative pressure coefficients are associated with lift generation in aerodynamics.

2. C_p **= 0:** When the pressure coefficient is zero, it signifies that the pressure at a specific point equals the reference pressure. This situation commonly arises in undisturbed flow regions or areas experiencing minimal pressure variations.

3. C_p **> 0:** A positive pressure coefficient suggests a region of high pressure compared to the reference pressure. This may arise in regions where the flow is decelerated or separated, such as the lower surface of an airfoil or around obstacles. Positive pressure coefficients are associated with drag forces in aerodynamics.

2.6.10 | Darcy Number

The Darcy number, also known as the Darcy-Weisbach friction factor, is a dimensionless quantity used in fluid mechanics to characterize the relative importance of viscous effects compared to inertial effects in fluid flow through porous media or pipes. The Darcy number represents the ratio of the inertial forces to the viscous forces in the flow. It provides insight into the dominance of these forces and helps classify flow regimes. It is named after Henry Darcy and Julius Weisbach, who made significant contributions to the field.

Da = (uL) / (V)

$$Da = \frac{\rho \cdot u \cdot L}{\mu \cdot V}$$

Here:

- ρ is the density of the fluid
- u is the velocity of the fluid
- L is a characteristic length scale of the flow
- μ is the dynamic viscosity of the fluid
- V is the kinematic viscosity of the fluid

Based on the value of Darcy Number, the flow can be categorized into different regimes:

1. Di < 1: In this regime, the viscous forces dominate over the inertial forces. The flow is said to be in the "Stokes flow" or "creeping flow" regime, where the flow is slow and highly influenced by viscous effects. Examples include flows through highly porous media or very thin pipes.

2. Di $\approx$ 1: In this regime, the inertial forces and viscous forces are of comparable magnitudes. The flow is in the transition regime, where both viscous and inertial effects are significant. Accurate predictions of flow behavior may require more complex analysis or experimental data.

3. Di $\gg$ 1: In this regime, the inertial forces dominate over the viscous forces. The flow is in the "fully developed" or "turbulent" regime, where the flow is fast, and inertial effects are predominant. Examples include flows through large pipes or porous media with high flow rates.

3

Mathematical Formulation

3.1 | Geometry

2D, unsteady, laminar natural convective flow of an incompressible hybrid nanofluid in a complex curve-shaped enclosure containing a heated wavy cylinder is taken into consideration in the presence of a magnetic field. The space between the complex enclosure and wavy cylinder is filled with a Cu–Al_2O_3/water hybrid nanofluid. Figure 1 depicts the schematic diagram of the physical domain of the present problem with boundary conditions.

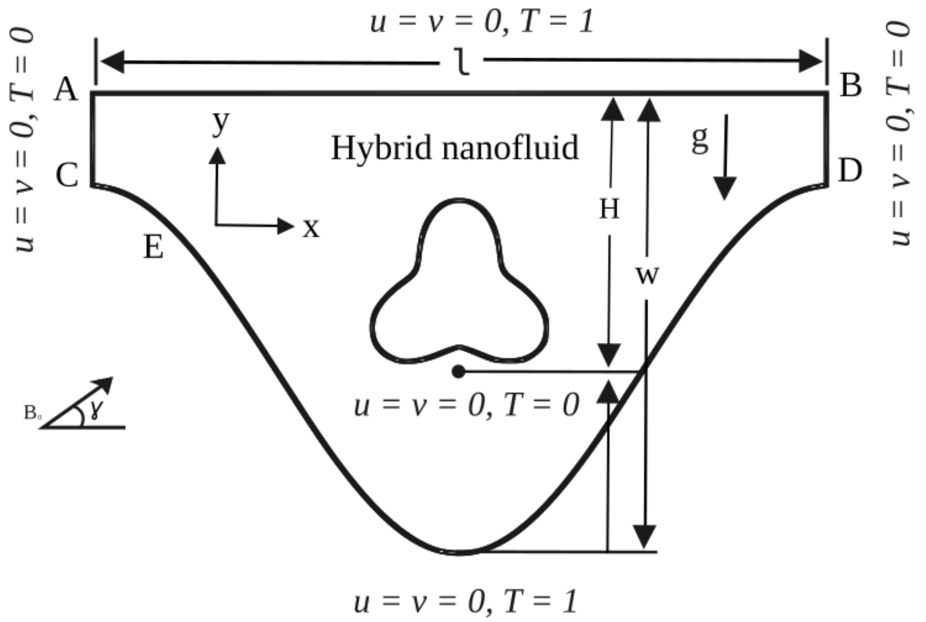


Figure 3.1: Schematic representation of the physical domain.

The lower part of the outer enclosure is deemed as sine curve-shaped, and the wavy cylinder is drawn by the dint of the following eqution.

$$\begin{aligned} r_1(\eta) &= -(r_i + A\cos(n\eta)) \cdot \sin(\eta) \\ r_2(\eta) &= (r_i + A\cos(n\eta)) \cdot \cos(\eta) \end{aligned} \tag{3.1}$$

Here,

r_i is the main radius of the inner cylinder
η is the angular position
A is the amplitude of the wave of the inner wavy cylinder
n is the number of waves = 3

From figure 3.1, l is the length of the horizontal direction of the enclosure and W is the overall vertical height and . H is the height of the upper wall AB to the lower curved wall CD. Amplitude of the lower curved wall is considered as h. The lower curved wall CD and the upper wall AB is supposed to be heated with temperature T_{hot}. Inner cylinder and the other walls of the outer enclosure is heated by a relatively low temperature T_{cold} . Natural convection occurs because of the temperature difference between the hot walls and the cold cylinder and walls. (Giwa et al., 2023)

3.2 | Governing Equations

Momentum conservation equation:

$$\begin{aligned} u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} &= \frac{-1}{\rho_{hnf}}\frac{\partial p}{\partial x} + \frac{\mu_{hnf}}{\rho_{hnf}}\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right) \\ &\quad + \frac{B^2\sigma_{hnf}}{\rho_{hnf}}\left(-u\sin^2\gamma + v\cos\gamma\sin\gamma\right) \end{aligned} \tag{3.2}$$

$$\begin{aligned} &u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} = \frac{-1}{\rho_{hnf}}\frac{\partial p}{\partial x} + \frac{\mu_{hnf}}{\rho_{hnf}}\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right) + \frac{(\rho\beta)_{hnf}}{\rho_{hnf}}g\,(T - T_c) \\ &+ \frac{B^2\sigma_{hnf}}{\rho_{hnf}}\left(-v\cos^2\gamma + u\cos\gamma\sin\gamma\right) \end{aligned} \tag{3.3}$$

Energy conservation equation:

$$u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y} = \frac{k_{hnf}}{(\rho c_p)_{hnf}}\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2}\right) \tag{3.4}$$

Here , $\mu_{hnf}, \rho_{hnf}, \alpha_{hnf}, \left(\rho C_p\right)_{hnf}, (\rho\beta)_{hnf}, k_{hnf}$ can be expressed as,

$$\mu_{hnf} = \frac{\mu_{bf}}{(1-\varphi)^{2.5}};$$
$$\alpha_{hnf} = \frac{k_{hnf}}{\left(\rho C_p\right)_{hnf}};$$
$$\rho_{hnf} = (1-\varphi)\rho_{bf} + \varphi\rho_{hp};$$
$$(\rho\beta)_{hnf} = (1-\varphi)(\rho\beta)_{bf} + \varphi(\rho\beta)_{hp};$$
$$\left(\rho C_p\right)_{hnf} = (1-\varphi)\left(\rho C_p\right)_{bf} + \varphi\left(\rho C_p\right)_{hp};$$
$$\frac{\sigma_{hnf}}{\sigma_{bf}} = 1 + \frac{3\left(\frac{\sigma_{hp}}{\sigma_{bf}} - 1\right)\varphi}{\left(\frac{\sigma_{hp}}{\sigma_{bf}} + 2\right) - \left(\frac{\sigma_{hp}}{\sigma_{bf}} - 1\right)\varphi}$$
$$k_{hnf} = \frac{k_{bf}\left[\left(k_{hp} + 2k_{bf}\right) - 2\varphi\left(k_{bf} - k_{hp}\right)\right]}{\left(k_{hp} + 2k_{bf}\right) + \varphi\left(k_{bf} - k_{hp}\right)};$$

Other physical properties can be represented as:

$$\rho_{hp} = \frac{\varphi_{cu}\rho_{cu} + \varphi_{Al_2O_3}\rho_{Al_2O_3}}{\varphi};$$
$$\beta_{hp} = \frac{\varphi_{cu}\beta_{cu} + \varphi_{Al_2O_3}\beta_{Al_2O_3}}{\varphi}$$
$$\left(C_p\right)_{hp} = \frac{\varphi_{cu}\left(C_p\right)_{cu} + \varphi_{Al_2O_3}\left(C_p\right)_{Al_2O_3}}{\varphi};$$
$$k_{hp} = \frac{\varphi_{cu}k_{cu} + \varphi_{Al_2O_3}k_{Al_2O_3}}{\varphi}$$
$$\sigma_{hp} = \frac{\varphi_{cu}\sigma_{cu} + \varphi_{Al_2O_3}\sigma_{Al_2O_3}}{\varphi}$$

To derive non dimensional form, we use

$$X = \frac{x}{L}; \quad Y = \frac{y}{L}; \quad U = \frac{uL}{\alpha_{bf}}; \quad V = \frac{vL}{\alpha_{bf}};$$
$$T' = \frac{T - T_c}{\Delta T}; \quad P = \frac{L^2 p}{\rho_{\text{hnf}}\,\alpha^2{}_{bf}}$$

Now,

$$\frac{\partial u}{\partial x} = \frac{\partial u}{\partial U}\frac{\partial U}{\partial X}\frac{\partial X}{\partial x} = \frac{1}{\frac{\partial U}{\partial u}}\frac{\partial U}{\partial X}\frac{\partial X}{\partial x} = \frac{\alpha_{bf}}{L}\frac{\partial U}{\partial X}\frac{1}{L} = \frac{\alpha_{bf}}{L^2}\frac{\partial U}{\partial X}$$
$$\frac{\partial u}{\partial y} = \frac{\partial u}{\partial U}\frac{\partial U}{\partial Y}\frac{\partial Y}{\partial y} = \frac{1}{\frac{\partial U}{\partial u}}\frac{\partial U}{\partial Y}\frac{\partial Y}{\partial y} = \frac{\alpha_{bf}}{L}\frac{\partial U}{\partial Y}\frac{1}{L} = \frac{\alpha_{bf}}{L^2}\frac{\partial U}{\partial Y}$$

similarly, we can write,

$$
\begin{aligned}
u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} &= \frac{\alpha^2{}_{bf}}{L^3}\left(U\frac{\partial U}{\partial X} + V\frac{\partial U}{\partial Y}\right)\\
u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} &= \frac{\alpha^2 bf}{L^3}\left(U\frac{\partial V}{\partial X} + V\frac{\partial V}{\partial Y}\right)
\end{aligned}
\tag{3.5}
$$

$$
u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} = \frac{\alpha^2 bf}{L^3}\left(U\frac{\partial V}{\partial X} + V\frac{\partial V}{\partial Y}\right)
\tag{3.6}
$$

$$
\frac{\partial^2 u}{\partial x^2} = \frac{\partial}{\partial x}\left(\frac{\partial u}{\partial x}\right) = \frac{1}{L}\frac{\alpha_{bf}}{L^2}\frac{\partial^2 U}{\partial X^2} = \frac{\alpha_{bf}}{L^3}\frac{\partial^2 U}{\partial X^2}
\tag{3.7}
$$

$$
\frac{\partial^2 u}{\partial y^2} = \frac{\partial}{\partial y}\left(\frac{\partial u}{\partial y}\right) = \frac{1}{L}\frac{\alpha_{bf}}{L^2}\frac{\partial^2 U}{\partial Y^2} = \frac{\alpha_{bf}}{L^3}\frac{\partial^2 U}{\partial Y^2}
\tag{3.8}
$$

so,

$$
\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} = \frac{\alpha_{bf}}{L^3}\left(\frac{\partial^2 U}{\partial X^2} + \frac{\partial^2 U}{\partial Y^2}\right)
\tag{3.9}
$$

similarly,

$$
\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} = \frac{\alpha_{bf}}{L^3}\left(\frac{\partial^2 V}{\partial X^2} + \frac{\partial^2 V}{\partial Y^2}\right)
$$

Again,

$$
\begin{aligned}
\frac{\partial p}{\partial x} &= \frac{\partial p}{\partial P}\frac{\partial P}{\partial X}\frac{\partial X}{\partial x} = \frac{\rho_{hnf}\alpha^2{}_{bf}}{L^2}\frac{\partial P}{\partial X}\frac{1}{L} = \frac{\rho_{hnf}\alpha^2{}_{bf}}{L^3}\frac{\partial P}{\partial X}\\
\frac{\partial p}{\partial y} &= \frac{\partial p}{\partial P}\frac{\partial P}{\partial Y}\frac{\partial Y}{\partial y} = \frac{\rho_{hnf}\alpha^2{}_{bf}}{L^2}\frac{\partial P}{\partial Y}\frac{1}{L} = \frac{\rho_{hnf}\alpha^2{}_{bf}}{L^3}\frac{\partial P}{\partial Y}
\end{aligned}
$$

Here,

$$
\begin{aligned}
\frac{\partial T}{\partial x} &= \frac{\partial}{\partial x}\left(T'\Delta T + T_c\right) = \Delta T\frac{\partial T'}{\partial x} = \Delta T\frac{\partial T'}{\partial X}\frac{\partial X}{\partial x} = \frac{\Delta T}{L}\frac{\partial T'}{\partial X}\\
\frac{\partial T}{\partial y} &= \frac{\partial}{\partial y}\left(T'\Delta T + T_c\right) = \Delta T\frac{\partial T'}{\partial y} = \Delta T\frac{\partial T'}{\partial Y}\frac{\partial Y}{\partial y} = \frac{\Delta T}{L}\frac{\partial T'}{\partial Y}
\end{aligned}
$$

So,

$$
u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y} = \frac{\alpha_{bf}\Delta T}{L^2}\left(U\frac{\partial T'}{\partial X} + V\frac{\partial T'}{\partial Y}\right)
\tag{3.10}
$$

Again,

$$
\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} = \frac{\partial}{\partial x}\left(\frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(\frac{\partial T}{\partial y}\right) = \frac{\Delta T}{L^2}\left(\frac{\partial^2 T'}{\partial X^2} + \frac{\partial^2 T'}{\partial Y^2}\right)
\tag{3.11}
$$

Now, the continuity equation is formed as: $\frac{\partial U}{\partial X} + \frac{\partial V}{\partial Y} = 0$

Momentum conservation equation is formed by using (eq. 3.5 to 3.9):

X-momentum:

$$U\frac{\partial U}{\partial X}+V\frac{\partial U}{\partial Y}=-\frac{\partial P}{\partial X}+\frac{\mu_{hnf}}{\rho_{hnf}\alpha_{bf}}\left(\frac{\partial^2 U}{\partial X^2}+\frac{\partial^2 U}{\partial Y^2}\right)+\frac{\sigma_{hnf}\rho_{bf}Ha^2Pr}{\sigma_{bf}\rho_{hnf}}\left(-U\sin^2\gamma+V\cos\gamma\sin\gamma\right) \tag{3.12}$$

Y-momentum:

$$\Longrightarrow U\frac{\partial V}{\partial X}+V\frac{\partial V}{\partial Y}=-\frac{\partial P}{\partial X}+\frac{\mu_{hnf}}{\rho_{hnf}\alpha_{bf}}\left(\frac{\partial^2 V}{\partial X^2}+\frac{\partial^2 V}{\partial Y^2}\right)+ \\ \frac{(\rho\beta)_{hnf}}{\rho_{hnf}\beta_{bf}}PrRaT'+\frac{\sigma_{hnf}\rho_{bf}Ha^2Pr}{\sigma_{bf}\rho_{hnf}}\left(-V\cos^2\gamma+U\cos\gamma\sin\gamma\right) \tag{3.13}$$

Energy conservation equation is formed by using (eqn. 3.10 to 3.11) :

$$U\frac{\partial T'}{\partial X}+V\frac{\partial T'}{\partial Y}=\frac{\alpha_{hnf}}{\alpha_{bf}}\left(\frac{\partial^2 T'}{\partial X^2}+\frac{\partial^2 T'}{\partial Y^2}\right) \tag{3.14}$$

Where,

$$\alpha_{hnf}=\frac{k_{hnf}}{\rho C_{p_{hnf}}},\mathrm{Pr}=\frac{\mu_{hnf}}{\rho_{hnf}\alpha_{hnf}},Ra=\frac{g\beta\rho_{hnf}L^3\Delta T}{\alpha_{hnf}\mu_{hnf}},Ha=BH\sqrt{\frac{\sigma}{\mu}}$$

Physical quantities are appeared as:

$$\frac{\mu_{hnf}}{\rho_{hnf}\alpha_{bf}}=\frac{Pr}{(1-\varphi)^{2.5}\left(1-\varphi+\varphi_b\right)}$$
$$\frac{\alpha_{hnf}}{\alpha_{bf}}=\frac{(s+2)-2\varphi(1-s)}{(s+2)+\varphi(1-s)(1-\varphi)+\varphi_{br}}$$
$$\frac{(\rho\beta)_{hnf}}{\rho_{\text{hnf}}\beta_{bf}}=\frac{(1-\varphi)+\varphi_{bm}}{(1-\varphi)+\varphi_b}$$

Where,

$$b=\frac{\rho_{hp}}{\rho_{bf}},m=\frac{\beta_{hp}}{\beta_{bf}},s=\frac{k_{hp}}{k_{bf}},r=\frac{(C_p)_{hp}}{(C_p)_{bf}}$$

And Nusselt Number, Nu = -$\frac{k_{hnf}}{k_{bf}}\cdot\frac{\delta T}{\delta\bar{n}}$

3.3 | Boundary Conditions

The following is a list of the relevant boundary conditions:

$$
\begin{aligned}
\text{Entire domain } &: u = v = 0, \quad T = 0 \\
\text{upper horizontal wall AB} &: u = v = 0, \quad T = 1 \\
\text{Right side wall BC} &: u = v = 0, \quad T = 0 \\
\text{Lower curve wall CD} &: u = v = 0, \quad T = 1 \\
\text{Left side wall AD} &: u = v = 0, \quad T = 0 \\
\text{Inner cylinder } &: u = v = 0, \quad T = 0
\end{aligned}
$$

4

Methodology

4.1 | Applications Used

Two applications are used to solve and model this complex phenomenon. The applications are COMSOL multiphysics and MATLAB. COMSOL is used to solve the physical problem that is encounted and MATLAB is used to do further manipulations to get a better view of the deleveloped model. Using these two applications the complex problem is first solved then modeled accordingly.

4.1.1 | COMSOL

COMSOL Multiphysics is a software extensively utilized by engineers and scientists across various domains of engineering, manufacturing, and scientific research. It offers a simulation platform that enables the modeling and simulation of designs, devices, and processes. With COMSOL Multiphysics, users can achieve fully coupled multiphysics and single-physics modeling capabilities. It's Model Builder encompasses defining geometries, material properties, and the physics governing specific phenomena, as well as conducting computations and analyzing the results obtained. (Comsol-Web)

One Very interesting feature of COMSOL multiphysics application is that it uses the Finite Element method to solve partial differential equations numerically (Dickinson et al., 2014a) . This applicaton has the ability to expand any motion related problem and solve the problem using its various components and modules. The features of COMSOL makes it very convenient for the user to design and understand complicated models. This is why, this tool has become the leading product design software for researchers, and engineers for any preliminary stage of machinery development project.

4.1.2 | MATLAB

MATLAB is a programming and numeric computing platform used to analyze data, develop algorithms, and create models. (MATLAB-Web) It integrates computation, visualization, and programming in an easy-to-use environment where problems and solutions are expressed in familiar mathematical notation(WISC)

MATLAB's versatility, extensive library of functions and toolboxes, and its ease of use make it a popular choice for professionals and researchers in various fields for numerical analysis, data processing, and algorithm development.

4.2 | Finite Element Method

The finite element method (FEM) is a widely used numerical technique for solving complex engineering and mathematical problems in various fields. It divides a complex problem domain into smaller regions called finite elements and represents each element with mathematical equations that approximate the system's behavior. By assembling these equations, the entire problem domain is modeled as a system of algebraic equations. (Zienkiewicz, 1989)

The Finite element method (FEM) utilizes a variational formulation to find approximate solutions by minimizing a suitable functional, often based on energy principles. This results in a system of algebraic equations that can be solved numerically.(Dickinson et al., 2014b) The most important advantage of the FEM is that it can handle complex geometries and boundary conditions. The domain is be divided into elements of various shapes and sizes, enabling accurate representation of irregular geometries. Moreover, different types of boundary conditions, such as fixed displacements, prescribed forces, and temperature gradients, can be easily incorporated into the analysis. (Baccouch, 2021)

The FEM provides the advantage of being able to effectively address various physical phenomena. If the governing equations and constitutive relationships for each element is chosen correctly, the FEM can accurately model a wide range of phenomena, including structural deformations, heat conduction, fluid flow, and electromagnetic fields. This versatility makes it an invaluable tool for analyzing and simulating complex real-world problems. (Ashcroft and Mubashar, 2011)

Nevertheless, the FEM has limitations. The computational cost increases as the number of elements grows, and inaccurate results can occur if the mesh has distorted or irregular elements (Eymard et al., 2014) . Proper selection of approximation functions is essential to capture the system's behavior accurately.

Despite these limitations, the finite element method remains widely used in engineering and scientific applications. Its ability to handle complex geometries, diverse physical phenomena, and its overall versatility make it an indispensable tool for solving a wide range of engineering problems and advancing scientific understanding. (Kumar and Kumar)

4.3 | COMSOL Workflow: Simulating Designs and Processes

In COMSOL Multiphysics, solving the problem involves several steps. Firstly, the appropriate component and nature of the model are selected based on the given equations. In this case, a two-dimensional laminar model with possible heat transfer is chosen. Next, a geometry is constructed to meet the specified requirements, comprising a plain upper wall and a curved bottom wall. A wavy shaped cylinder is placed inside of the upper and lower walls to observe the effects of this cylinder in fluid flow.

After defining the geometry, the model is converted into a solid representation. Parameters such as Reynolds number (Re), Grashof number (Gr), Prandtl number (Pr), Hartmann number (Ha), Richardson number (RI), and heat absorption/generation coefficient (A) are set. The focus is on laminar single-phase flow for the fluid properties. Density, dynamic viscosities, initial values, and volume forces including magnetic effects are specified.

Boundary conditions for heat transfer are then established. The upper wall and the curved bottom wall is assumed to be heated and all the other walls are maintained at low temperatures. Temperature values of 0 (indicating lower temperature) and 1 (indicating higher temperature) are assigned accordingly. A heat absorption or generation function is applied to account for any relevant phenomena.

Additional adjustments are made to the mesh distribution and parameter variations to provide the user with the flexibility to choose specific values. Finally, the model is sent for computation, and the obtained results are thoroughly examined.

5

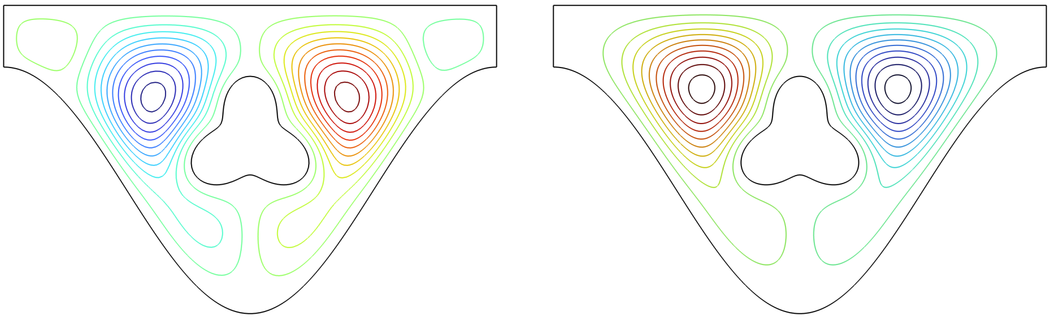
Results and Discussion

In this project, the flow and heat transfer of hybrid nanofluid in a 2D enclosure that has a wavy cylinder inside. The results of this project is represented graphically in the figure of isotherms, velocity distribution.

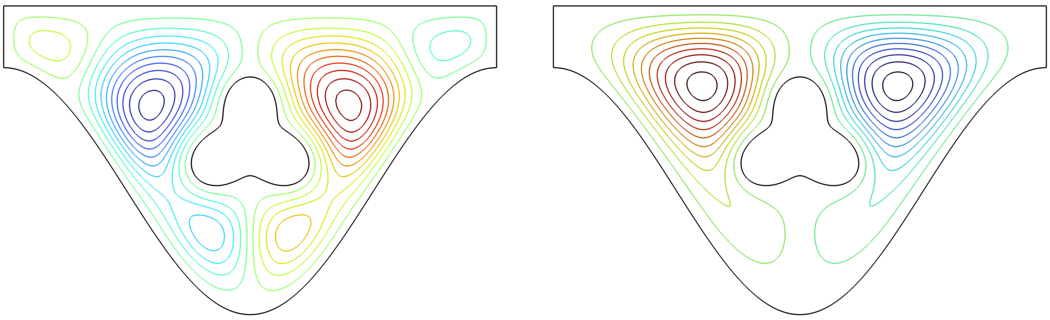
To validate the numerical solution of this project, an examination of the results from Saha et al. is conducted. The study by Saha et al. focused on the "Natural convection of a hybrid nanofluid from a sinusoidal wavy cylinder placed in a curved cavity." While there are differences in the heat source and conditions compared to the current study's geometry, most of the other criteria remain consistent, as evidenced by this comparison.

In Saha et al. (2021), the inner wavy cylinder is solely heated to a higher temperature than the other walls of the enclosure. In contrast, in this project, a higher temperature is applied to the middle portion of the lower curve, while the remaining portion experiences zero heat flux. The other walls, including the inner wavy cylinder, are relatively cooler.

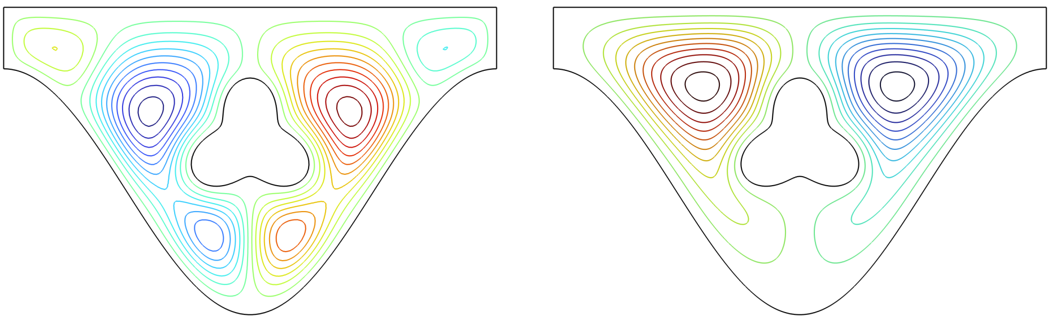
Numerical and graphical results obtained for different parametric values such as Reynolds, Grashof, Richardson, Hartmann, and heat production or absorption coefficients are shown and described in this section. The study focuses on analyzing the key features of laminar flow and heat transmission using streamline contours and isotherms. Furthermore, a comprehensive investigation of the Nusselt point and average Nusselt number is conducted, considering the variation in the Richardson Number. In this whole project we have taken the Prandtl number(Pr) equals to 6.2.



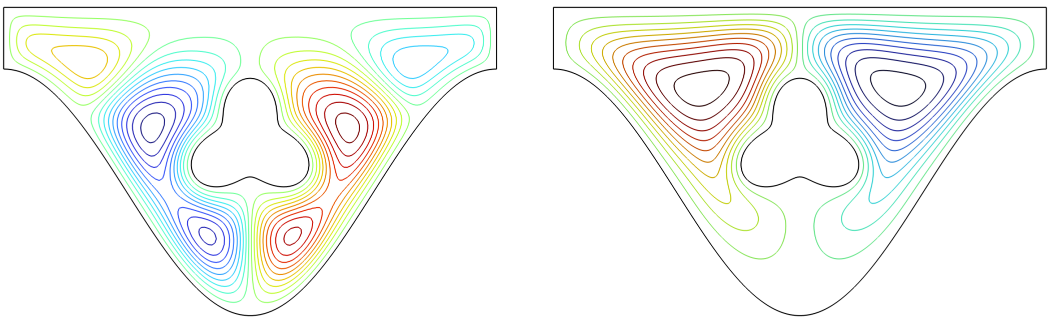
[a]



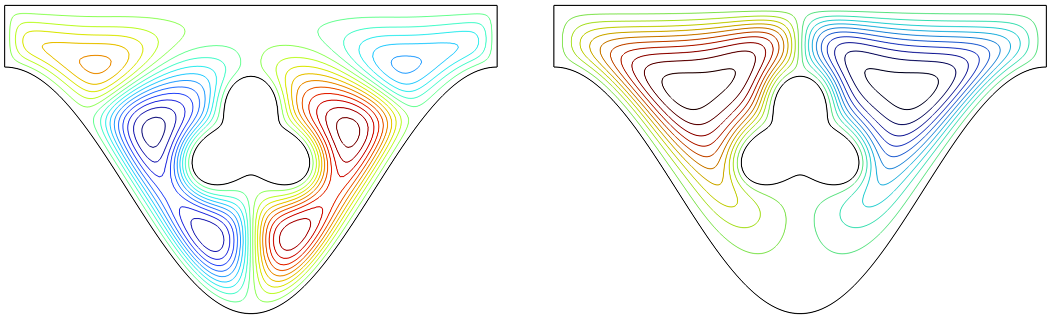
[b]



[c]

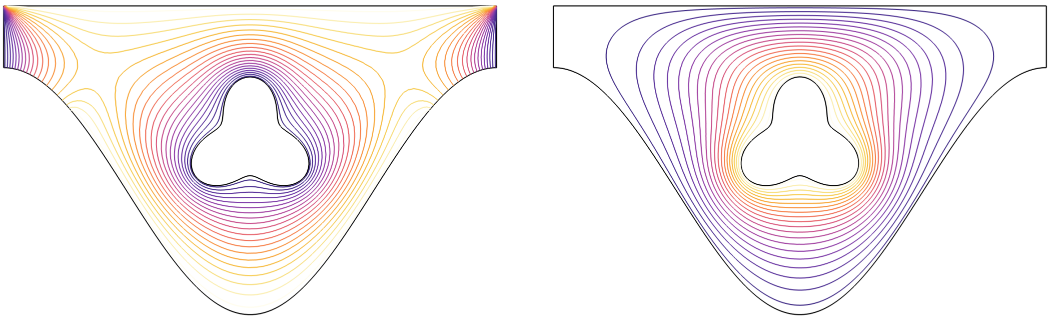


[d]

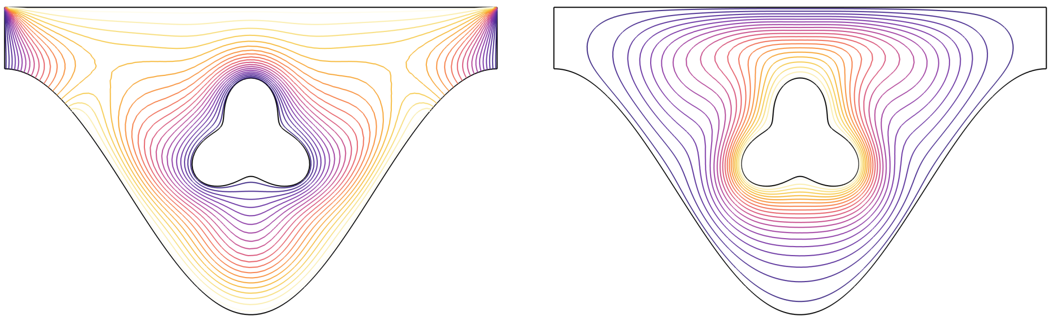


[e]

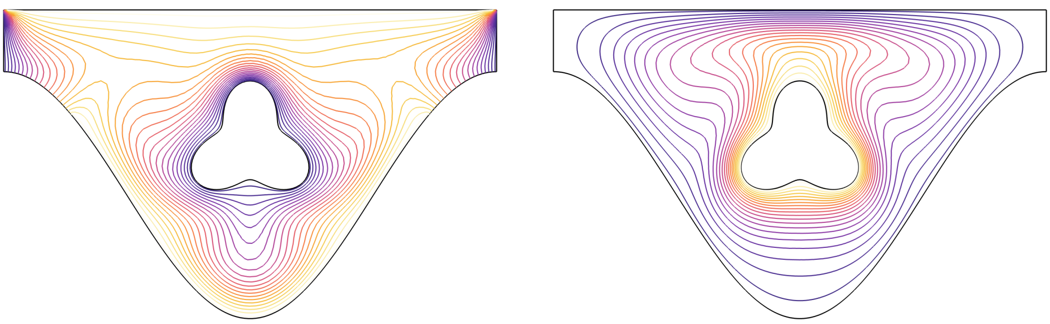
Figure 5.1: Comparison of Velocity Distribution obtained by Saha et al. (2021) (right) and present code (left) for different Rayleigh numbers: (a) Ra = 10^3 , (b) Ra = 5 $\times 10^3$, (c) Ra = 10^4 , and (d) Ra =5 $\times$ 10^4 (e) Ra =10^5



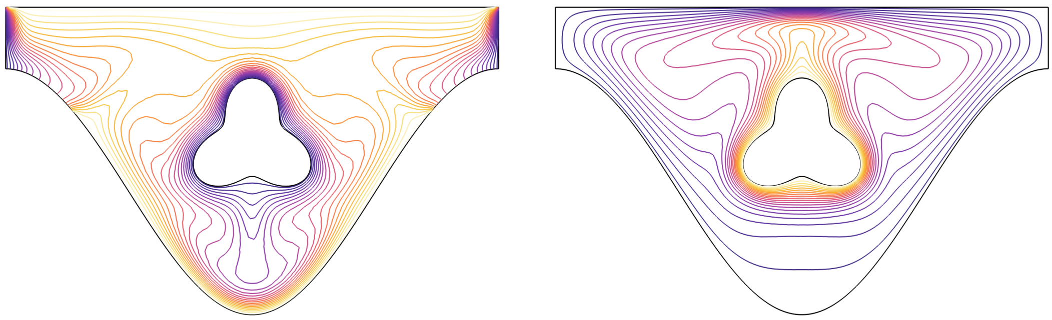
[a]



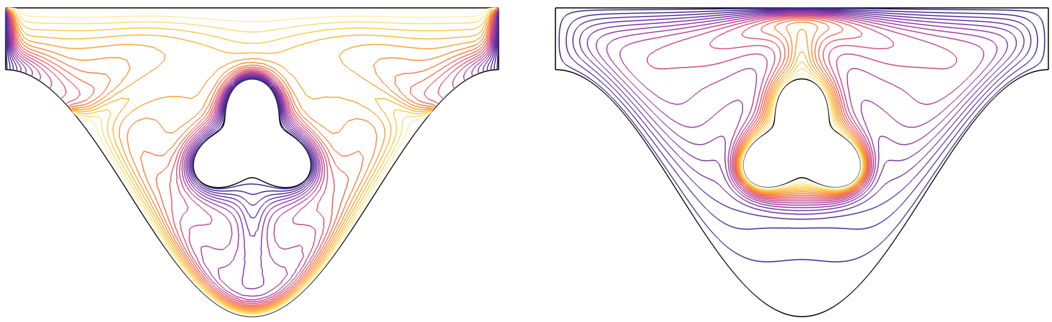
[b]



[c]



[d]



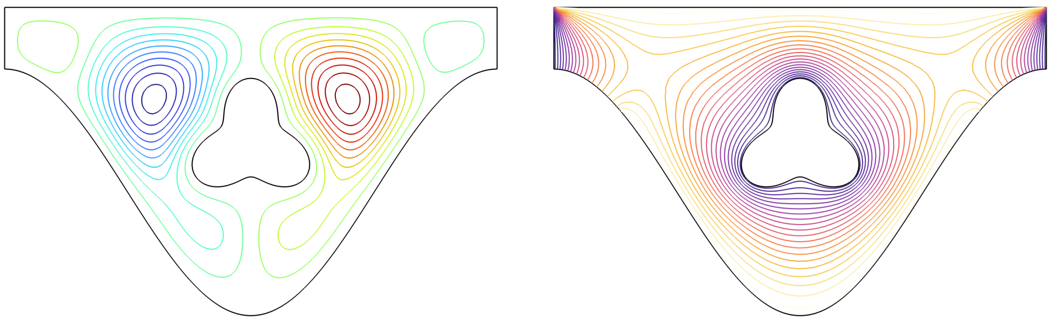
[e]

Figure 5.2: Comparison of Isotherms obtained by Saha et al. (2021)(right) and present code (left) for different Rayleigh numbers: (a) Ra = 10^3 , (b) Ra = 5 $\times 10^3$, (c) Ra = 10^4 , and (d) Ra =5 $\times$ 10^4 (e) Ra =10^5

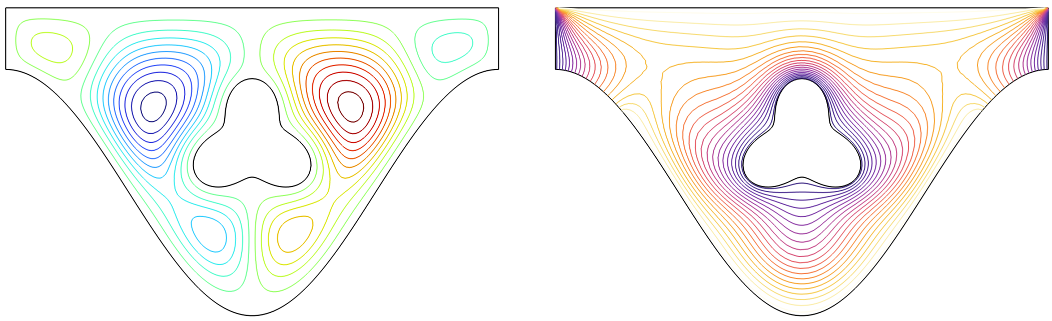
5.1 | Effect of Rayleigh number

To investigate the influence of buoyancy-driven flow and heat transfer in a system, a model was executed with varying values of the Rayleigh number. The Rayleigh number, a dimensionless quantity that compares buoyancy forces to viscous forces in a fluid, provides valuable insights into fluid flow and heat transfer characteristics under diverse conditions.

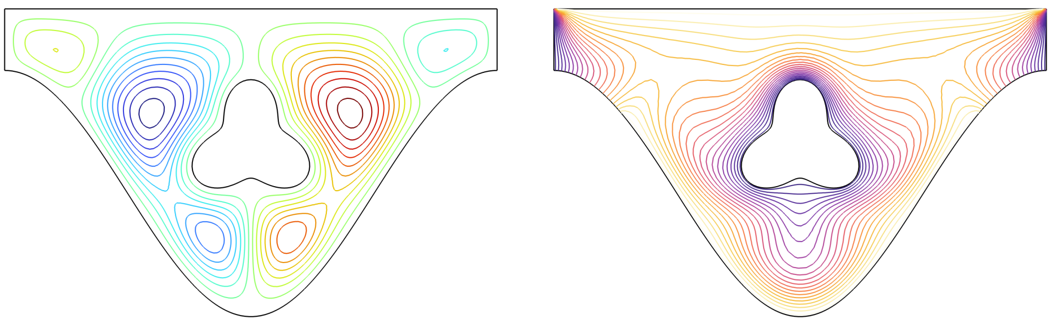
Analyzing the model for different Rayleigh numbers enables a comprehensive examination of how alterations in the driving forces affect flow patterns, heat distribution, and overall system behavior. It aids in comprehending the transition between laminar and turbulent flow regimes, the development of convective cells or plumes, and the efficiency of heat transfer within the system.



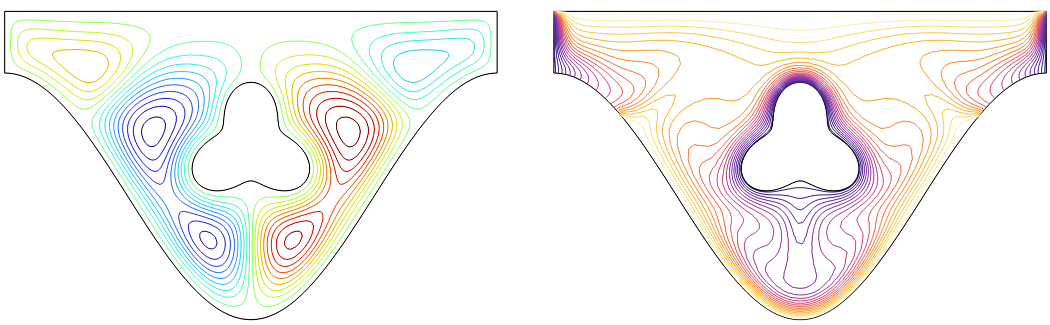
[a]



[b]



[c]



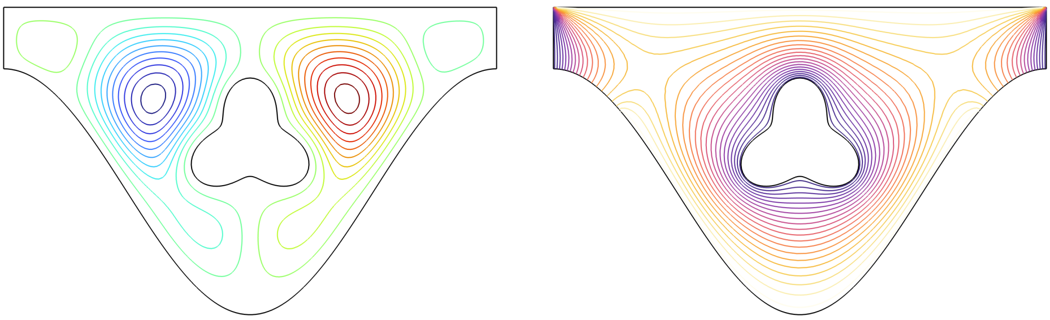
[d]

Figure 5.3: Velocity distribution and Isotherms for different Rayleigh numbers: (a) Ra = 10^3 , (b) Ra = 5 $\times 10^3$, (c) Ra = 10^4 , and (d) Ra =5 $\times$ 10^4

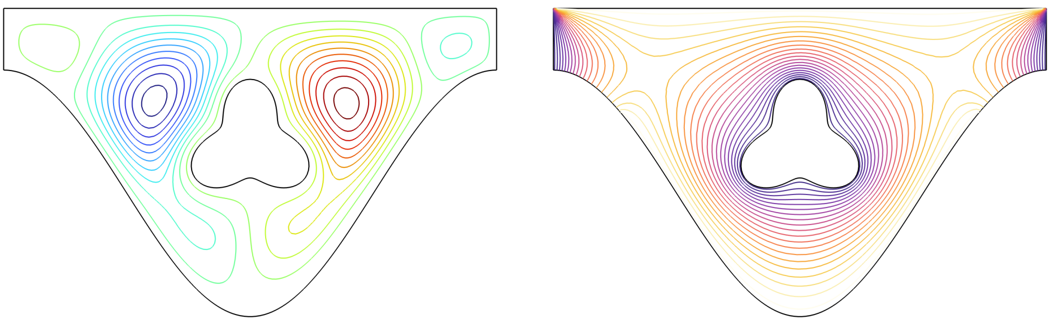
5.2 | Effect of Hartmann number

This model was run for different values of the Hartmann number in order to analyze its impact on the system under study. The Hartmann number (Ha) is a dimensionless parameter used to characterize the behavior of a conducting fluid in the presence of a magnetic field. It quantifies the relative importance of magnetic forces compared to viscous forces within the fluid.

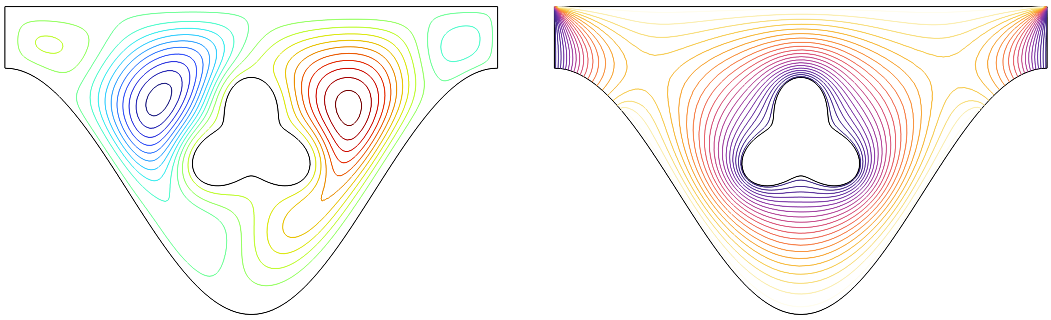
By varying the Hartmann number, the study aimed to observe how changes in magnetic field strength would affect the flow and heat transfer characteristics of the system. This analysis helps in understanding the influence of magnetic forces on fluid behavior and the overall performance of the system. It provides valuable insights into the optimal operating conditions and potential limitations associated with different Hartmann number values.



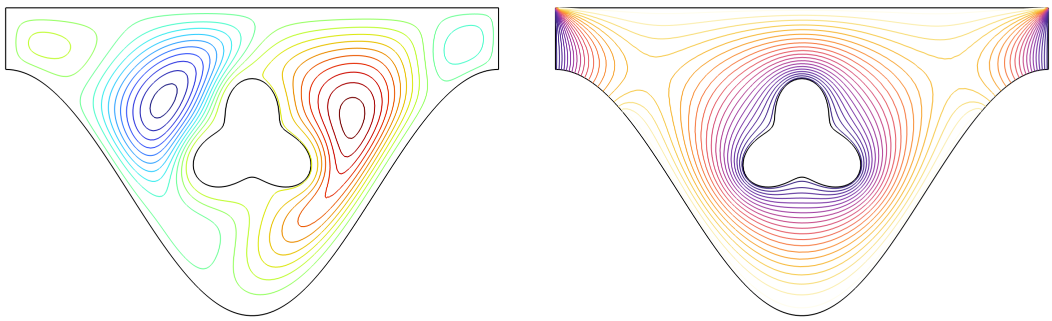
[a]



[b]

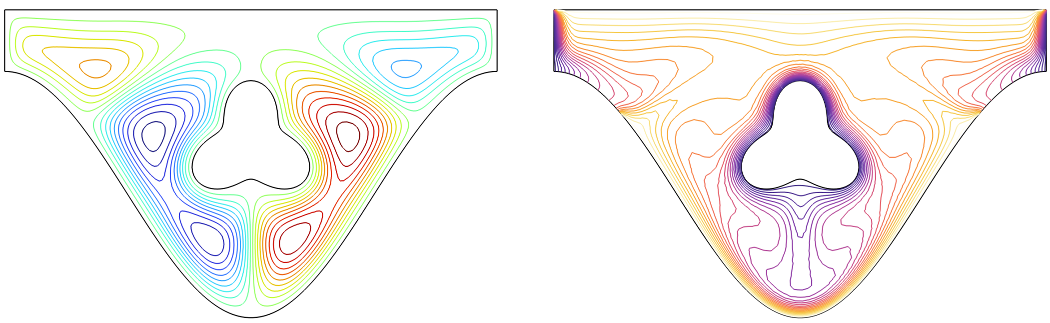


[c]

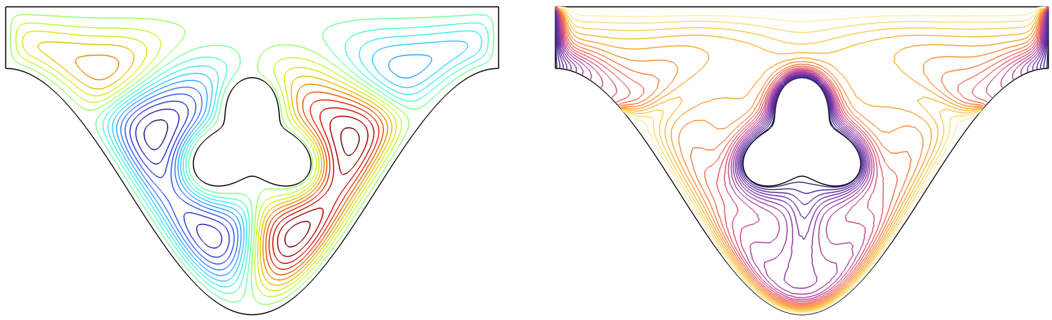


[d]

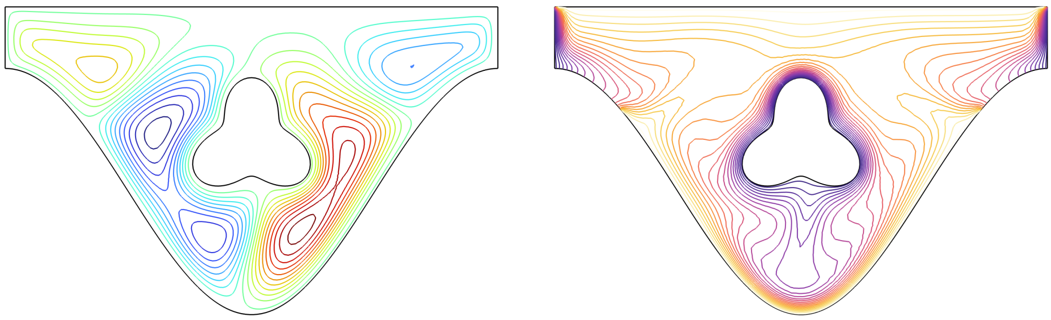
Figure 5.4: Velocity distribution and Isotherm for different Hartmann numbers: (a) Ha = 0 , (b) Ha = 5 , (c) Ha = 10 , and (d) Ha =15 when Ra = 10^3



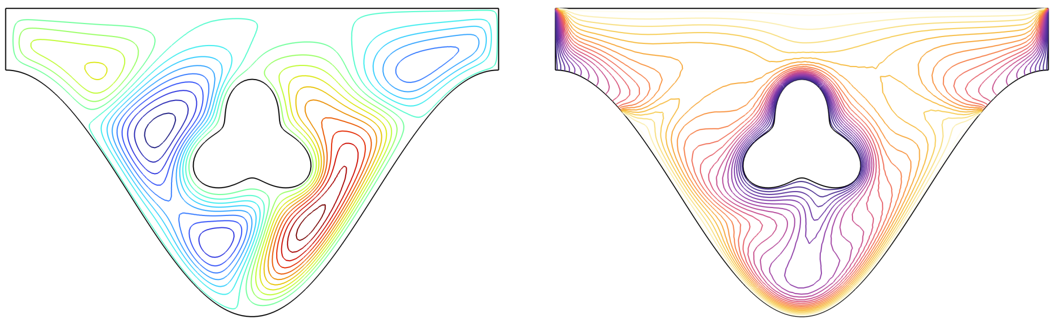
[a]



[b]



[c]



[d]

Figure 5.5: Velocity distribution and Isotherm for different Hartmann numbers: (a) Ha = 0 , (b) Ha = 5 , (c) Ha = 10 , and (d) Ha =15 when Ra = 10^5

5.3 | Effect of Volume Fraction

The model was run for different values of volume fraction to examine the impact of varying particle concentrations on the flow behavior. By adjusting the volume fraction, which represents the ratio of the volume occupied by particles to the total volume of the mixture, we can investigate how the presence of particles affects fluid flow characteristics.

Running the model for different volume fractions allows us to observe changes in

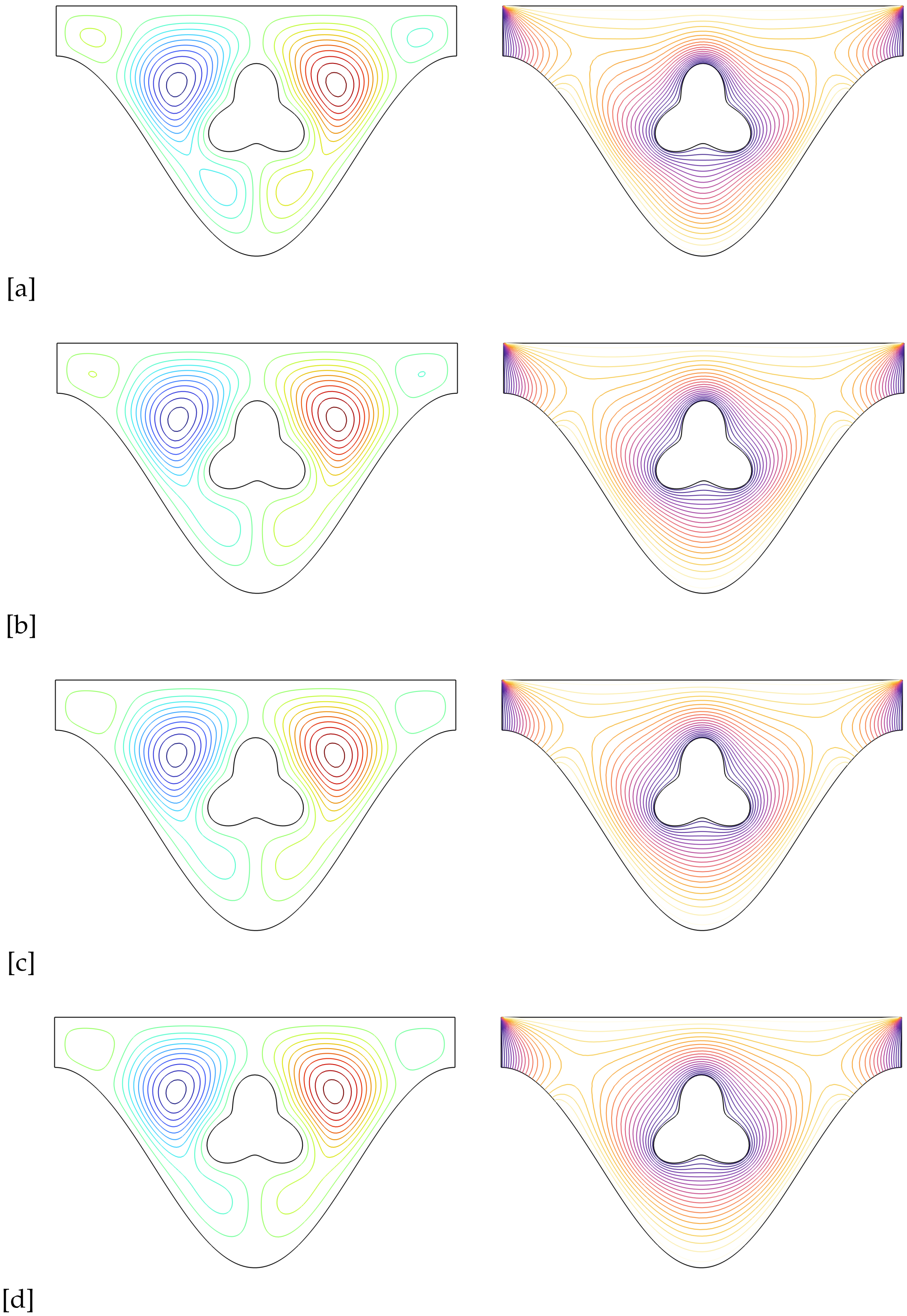


Figure 5.6: Velocity distribution and Isotherm for different Volume Fractions: (a) ϕ = 0.01 , (b) ϕ = 0.03 , (c) ϕ = 0.06 , and (d) ϕ = 0.09 when Ra = 10^3

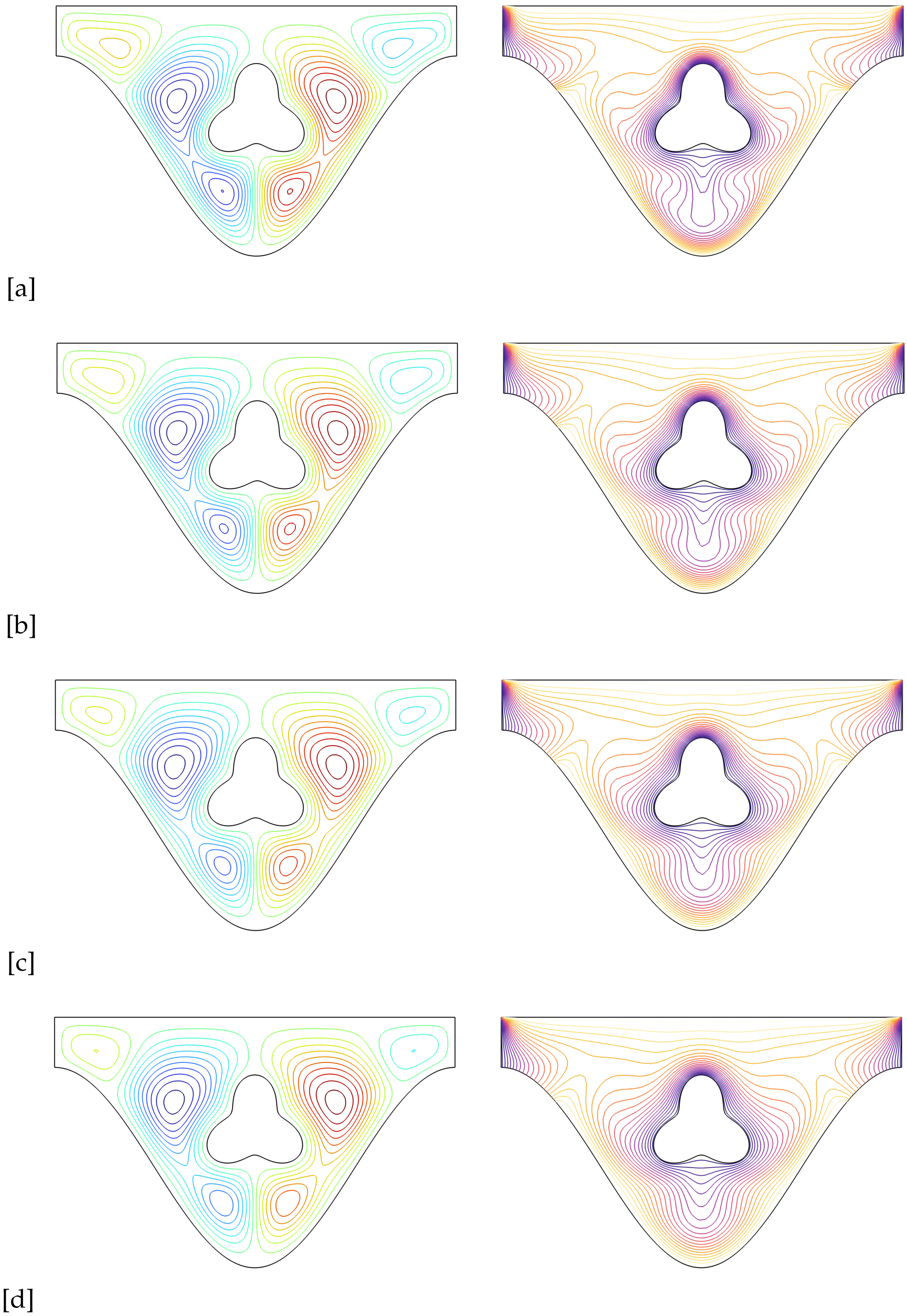


Figure 5.7: Velocity distribution and Isotherm for different Volume Fractions: (a) ϕ = 0.01 , (b) ϕ = 0.03 , (c) ϕ = 0.06 , and (d) ϕ = 0.09 when Ra = 10^4

velocity distribution, isotherms and the pressure distribution. It helps in understanding the behavior of multiphase flows, where solid particles or droplets are dispersed within a fluid medium.

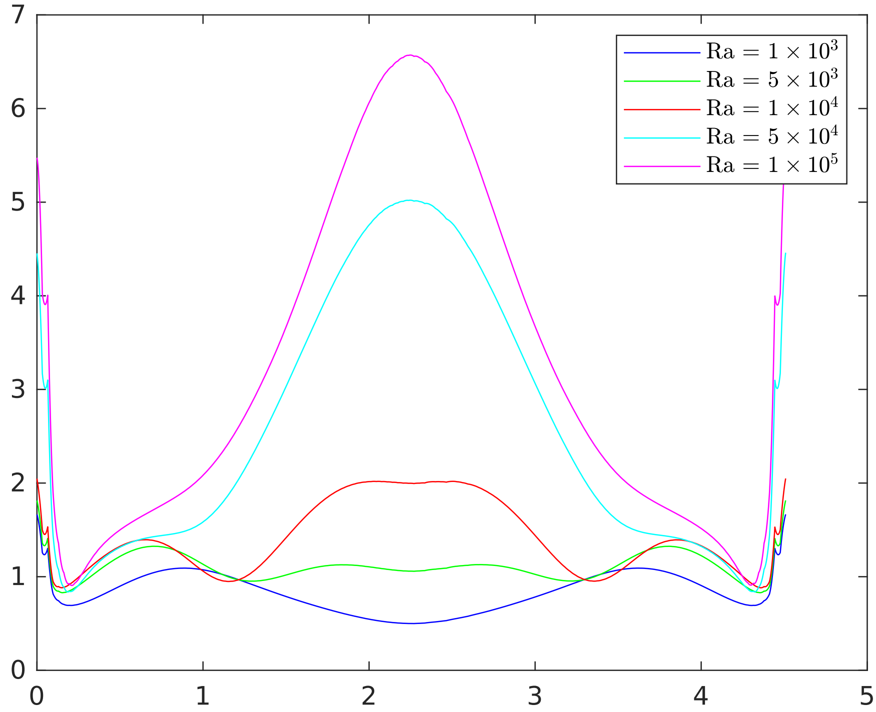


Figure 5.8: Nusselt Number Comparison when $\phi = 0.09$, $Pr = 6.2$

6

Conclusions

The project aims to numerically analyze the effects of a wavy cylinder placed inside a curved shaped cavity that is partially heated filled with hybrid nanofluid in effect of magnetic field. This causes a natural convection of heat transfer in the annular region. The top wall and the curved bottom wall of the cavity is partially heated while the inner wavy cylinder is remained in the cold temparature. The governing equations for this complex phenomenon are solved using the simulation program COMSOL Multiphysics and further investigated into MATLAB.

The obtained results are compared with previously published research on similar scenarios, and overall agreement is observed. The study presents and discusses illustrative findings, including velocity streamlines and isotherm contours, for various parametric configurations. The Rayleigh number, Hartmann number, Volume fraction are identified as key factors influencing the heat transfer processes and flow characteristics within the cavity, where MHD driven natural convection dominates.

Changes in the Hartmann number and the Rayleligh number impact the isotherms but the more changes occurs in the velocity streamlines. The changes in the velocity distributio is also noticeable when the volume fraction of the nanofluid is changed. Overall. The magnetic field also contrubutes an important rule in this heat transfer of nanofluid inside of wavy cavity with curved shaped cylinder inside.

Overall the findings state that the higher value of Rayleigh number increses the heat and fluid tranfer noticeably. Increment of Rayleigh number and decrement of Hartmann number causes an intensification to take place in the fluid and heat transfer and adding the hybrid nanoparticles into the base fluid improves the heat transfer.

References

I. A. Ashcroft and A. Mubashar. *Numerical Approach: Finite Element Analysis*, pages 629–660. Springer Berlin Heidelberg, Berlin, Heidelberg, 2011. ISBN 978-3-642-01169-6. doi: 10.1007/978-3-642-01169-6_25. URL https://doi.org/10.1007/978-3-642-01169-6_25.

M. Baccouch. *Finite Element Methods and Their Applications*. BoD – Books on Demand, 11 2021.

G. K. Batchelor, H. K. Moffatt, and M. G. Worster. *Perspectives in Fluid Dynamics*. Cambridge University Press, 1 2003.

C.-C. Cho, C.-L. Chen, and C.-K. Chen. Natural convection heat transfer and entropy generation in wavy-wall enclosure containing water-based nanofluid. *International Journal of Heat and Mass Transfer*, 61:749–758, 06 2013. doi: 10.1016/j.ijheatmasstransfer.2013.02.044.

Comsol-Web. Comsol multiphysics simulation software understand, predict, and optimize real-world designs, devices, and processes with simulation. URL https://www.comsol.com/comsol-multiphysics.

E. J. F. Dickinson, H. Ekström, and E. Fontes. COMSOL Multiphysics®: Finite element software for electrochemical analysis. A mini-review. *Electrochemistry Communications*, 40:71–74, 3 2014a. doi: 10.1016/j.elecom.2013.12.020. URL https://doi.org/10.1016/j.elecom.2013.12.020.

E. J. F. Dickinson, H. Ekström, and E. Fontes. COMSOL Multiphysics®: Finite element software for electrochemical analysis. A mini-review. *Electrochemistry Communications*, 40:71–74, 3 2014b. doi: 10.1016/j.elecom.2013.12.020. URL https://doi.org/10.1016/j.elecom.2013.12.020.

M. El Haj Assad, M. Alhuyi Nazari, and M. A. Rosen. Chapter 1 - applications of renewable energy sources. In M. E. H. Assad and M. A. Rosen, editors, *Design and Performance Optimization of Renewable Energy Systems*, pages 1–15. Academic Press, 2021. ISBN 978-0-12-821602-6. doi: https://doi.org/10.1016/B978-0-12-821602-6.00001-8. URL https://www.sciencedirect.com/science/article/pii/B9780128216026000018.

R. Eymard, T. R. Gallouët, and R. Herbin. The Finite Volume Method Handbook of Numerical Analysis. 40:713–1020, 3 2014. doi: 10.1016/S1570-8659(00)07005-8.

S. O. Giwa, A. E. Adeleke, M. Sharifpur, and J. P. Meyer. Chapter 3 - from 2007 to 2021: present state of research on hybrid nanofluids as advanced thermal fluids. In S. Vijay, B. Solomon, and J. Meyer, editors, *Materials for Advanced Heat Transfer Systems*, Woodhead Publishing Series in Energy, pages 63–174. Woodhead Publishing, 2023. ISBN 978-0-323-90498-8. doi: https://doi.org/10.

1016/B978-0-323-90498-8.00001-4. URL https://www.sciencedirect.com/science/article/pii/B9780323904988000014.

F. Jamil and H. Ali. *Applications of hybrid nanofluids in different fields*, pages 215–254. 01 2020. ISBN 9780128192801. doi: 10.1016/B978-0-12-819280-1.00006-9.

R. W. Johnson. *Handbook of Fluid Dynamics*. CRC Press, 4 2016.

M. Kumar and P. Kumar. A Finite Element Approach for Finding Positive Solutions of Semilinear Elliptic Dirichlet Problems. 25:1119–1128, 3 . doi: 10.1002/num.20390.

Live-Science. What is fluid dynamics? URL https://www.livescience.com/47446-fluid-dynamics.html.

MATLAB-Web. Matlab : Math. graphics. programming. URL https://www.mathworks.com/products/matlab.html.

S. Mehryan, M. Sheremet, M. Soltani, and M. Izadi. Natural convection of magnetic hybrid nanofluid inside a double-porous medium using two-equation energy model. *Journal of Molecular Liquids*, 03 2019. doi: 10.1016/j.molliq.2018.12.147.

J. H. Merkin, I. Pop, Y. Y. Lok, and T. Grosan. Chapter 6 - mixed convection flow in porous medium. In J. H. Merkin, I. Pop, Y. Y. Lok, and T. Grosan, editors, *Similarity Solutions for the Boundary Layer Flow and Heat Transfer of Viscous Fluids, Nanofluids, Porous Media, and Micropolar Fluids*, pages 163–203. Academic Press, 2022. ISBN 978-0-12-821188-5. doi: https://doi.org/10.1016/B978-0-12-821188-5.00003-5. URL https://www.sciencedirect.com/science/article/pii/B9780128211885000035.

M. Nakhchi. Experimental optimization of geometrical parameters on heat transfer and pressure drop inside sinusoidal wavy channels. *Thermal Science and Engineering Progress*, 9:121–131, 2019. ISSN 2451-9049. doi: https://doi.org/10.1016/j.tsep.2018.11.006. URL https://www.sciencedirect.com/science/article/pii/S2451904918302427.

Oreilly. Continuum fluid mechanics and the navier-stokes equations. URL https://www.oreilly.com/library/view/understanding-aerodynamics-arguing/9781119967514/12_chap03.html.

K. Osintsev, S. Aliukov, S. Kuskarbekova, T. Tarasova, A. Karelin, V. Konchakov, and O. Kornyakova. Increasing thermal efficiency: Methods, case studies, and integration of heat exchangers with renewable energy sources and heat pumps for desalination. *Energies*, 16(13), 2023. ISSN 1996-1073. doi: 10.3390/en16134930. URL https://www.mdpi.com/1996-1073/16/13/4930.

S. Parvin, N. C. Roy, and L. K. Saha. Natural convective non-newtonian nanofluid flow in a wavy-shaped enclosure with a heated elliptic obstacle. *Heliyon*, 9(6):e16579, 2023. ISSN 2405-8440. doi: https://doi.org/10.1016/j.heliyon.2023.e16579. URL https://www.sciencedirect.com/science/article/pii/S2405844023037866.

PSU. Dimensional analysis and similarity. URL https://www.me.psu.edu/cimbala/learning/fluid/Dim_anal/dim_anal.htm.

N. C. Roy, L. K. Saha, and S. Siddiqa. Electrohydrodynamics and thermal radiation effects on natural convection flow in an enclosed domain. *International Communications in Heat and Mass Transfer*, 126: 105437, 2021. ISSN 0735-1933. doi: https://doi.org/10.1016/j.icheatmasstransfer.2021.105437. URL https://www.sciencedirect.com/science/article/pii/S0735193321003304.

L. K. Saha, S. Parvin, and N. C. Roy. Magnetohydrodynamic natural convection of a hybrid nanofluid from a sinusoidal wavy cylinder placed in a curve-shaped cavity. *AIP publishing*, pages 11–11, 2021.

M. Saif. Fluid flow: And types of fluid flow [fluid mechanic]. URL `https://www.theengineerspost.com/types-of-fluid-flow/`.

F. Selimefendigil and H. F. Öztop. Control of natural convection in a cnt-water nanofluid filled 3d cavity by using an inner t-shaped obstacle and thermoelectric cooler. *International Journal of Mechanical Sciences*, 169:105104, 2020. ISSN 0020-7403. doi: https://doi.org/10.1016/j.ijmecsci.2019.105104. URL `https://www.sciencedirect.com/science/article/pii/S0020740319324051`.

R. Shah and A. London. Chapter i - introduction. In R. Shah and A. London, editors, *Laminar Flow Forced Convection in Ducts*, pages 1–4. Academic Press, 1978. ISBN 978-0-12-020051-1. doi: https://doi.org/10.1016/B978-0-12-020051-1.50006-1. URL `https://www.sciencedirect.com/science/article/pii/B9780120200511500061`.

E. J. Shaughnessy, I. M. Katz, and J. P. Schaffer. *Introduction to Fluid Mechanics*. Oxford University Press, USA, 1 2005.

A. J. C. T. Tayebi. Natural convection enhancement in an eccentric horizontal cylindrical annulus using hybrid nanofluids," numer. heat transfer. *Numer. Heat Transfer Part A*, 71:1159–1173, 2017.

Z. Wang, J. M. Birch, and M. H. Dickinson. Unsteady forces and flows in low Reynolds number hovering flight:two-dimensional computations<i>vs</i>robotic wing experiments. *The Journal of Experimental Biology*, 207(3):449–460, 1 2004. doi: 10.1242/jeb.00739. URL `https://doi.org/10.1242/jeb.00739`.

T. Werder, J. Walther, and P. Koumoutsakos. Hybrid atomistic–continuum method for the simulation of dense fluid flows. *Journal of Computational Physics*, 205:373–390, 05 2005. doi: 10.1016/j.jcp.2004.11.019.

Wikipedia. Fluid dynamics. URL `https://en.wikipedia.org/wiki/Fluid_dynamics`.

WISC. What is matlab. URL `https://cimss.ssec.wisc.edu/wxwise/class/aos340/spr00/whatismatlab.htm`.

O. C. Zienkiewicz. *The finite element method*. Number 1. 1 1989. URL `http://cds.cern.ch/record/258841`.