MSI Journal of
Multidisciplinary Research (MSIJMR)

Volume: 3, Issue: 7 (July-2026)
Published by *MSI Publishers*
ISSN 3049-0669 (Online)

Received: 25-06-2026
Accepted: 03-07-2026
Published: 09-07-2026

TURBULENT NATURAL CONVECTION IN A VERTICAL CYLINDRICAL ENCLOSURE WITH LOCALIZED HEATING AND COOLING

Authors Details

Omariba Geofrey Ong'era[1*]
Department of Pure and Applied Mathematics, Jomo Kenyatta University of Agriculture and Technology (JKUAT), Juja, Kenya.

Augustine Ayanga Mayaka[2]
Department of Pure and Applied Mathematics, Jomo Kenyatta University of Agriculture and Technology (JKUAT), Juja, Kenya.

ABSTRACT: This study presents a numerical investigation of turbulent natural convection in a vertical cylindrical enclosure subjected to localized heating and cooling on opposite surfaces. The flow is driven by buoyancy forces arising from temperature gradients, leading to complex turbulent structures and enhanced heat transfer characteristics. The governing equations, including the continuity, momentum, and energy equations, are solved using a computational fluid dynamics (CFD) approach. The simulations are conducted for a range of Rayleigh numbers to examine the transition and behavior of turbulent flow regimes. Special attention is given to the influence of the spatial positioning of the heat source and sink on the flow structure, temperature distribution, and overall thermal performance. Results are presented in terms of velocity profiles, temperature distribution, turbulent intensity, and Nusselt number distributions. The findings indicate that the location of heating and cooling zones significantly affects the strength of circulation cells, thermal stratification, and heat transfer efficiency. Higher Rayleigh numbers intensify turbulence, resulting in improved mixing and increased heat transfer rates.The study provides valuable insights into optimizing heat transfer in

 DOI: ***10.5281/zenodo.21276551***

cylindrical systems, with potential applications in energy systems, industrial heat exchangers, and post-harvest storage technologies. The outcomes contribute to the advancement of numerical modeling techniques for turbulent natural convection and offer guidance forq the design of thermally efficient engineering systems.

Keywords: *Turbulent natural convection, Vertical cylindrical enclosure, Localized heating and cooling and Rayleigh number*

1 Introduction

Natural convection is a fundamental mode of heat transfer that occurs due to buoyancy forces generated by temperature differences within a fluid. Unlike forced convection, which relies on external mechanical means such as pumps or fans, natural convection is driven purely by density variations arising from thermal gradients. This phenomenon plays a critical role in a wide range of engineering and environmental systems, including building ventilation, electronic cooling, geothermal processes, and energy storage devices. In many practical applications, the flow regime becomes turbulent at high temperature differences, significantly influencing the heat transfer characteristics and fluid flow behavior (Incropera et al., 2007).

Turbulent natural convection is particularly important in enclosed geometries, where the interaction between buoyancy forces and confinement leads to complex flow structures. Enclosures such as cavities, cylinders, and rectangular domains are commonly encountered in industrial systems. Among these, cylindrical enclosures are of special interest due to their widespread use in engineering applications such as storage tanks, reactors, solar collectors, and food preservation systems. The curvature of the cylindrical geometry introduces additional complexity in flow patterns, resulting in three-dimensional effects and asymmetric circulation cells that differ significantly from those observed in rectangular enclosures (Bejan, 2013).

The study of natural convection in cylindrical enclosures has evolved over the years, with early investigations focusing primarily on laminar flow regimes. However, as the Rayleigh number increases, the flow transitions from laminar to turbulent, leading to enhanced mixing and heat transfer. Understanding this transition and accurately predicting turbulent behavior remain challenging due to the nonlinear and chaotic nature of turbulence. The

 DOI: ***10.5281/zenodo.21276551***

Rayleigh number, which represents the ratio of buoyancy to viscous and thermal diffusion forces, is a key parameter governing the flow regime and heat transfer intensity (Çengel & Ghajar, 2015). At high Rayleigh numbers, turbulent fluctuations dominate the flow, necessitating advanced modeling techniques to capture the physics accurately.

One important aspect of natural convection studies is the configuration of thermal boundary conditions. In many real-world systems, heating and cooling are not uniformly distributed but occur in localized regions. For instance, in electronic devices, heat is generated at specific components, while cooling may be applied at designated surfaces. Similarly, in agricultural storage systems, temperature gradients may arise due to localized environmental conditions. Localized heating and cooling create strong thermal gradients that induce complex circulation patterns, including multiple recirculation zones and plume formations. These localized effects significantly influence the overall heat transfer performance and must be carefully analyzed to optimize system design (Ostrach, 1988).

Computational Fluid Dynamics (CFD) has become an indispensable tool for studying turbulent natural convection in complex geometries. Numerical methods allow researchers to solve the governing equations of fluid flow and heat transfer with high accuracy and flexibility. Among the various turbulence models available, the k–ω turbulence model has gained popularity due to its ability to accurately predict near-wall behavior and handle adverse pressure gradients. This model is particularly suitable for enclosed flows where boundary layers play a significant role in heat transfer processes (Wilcox, 2006). By incorporating turbulence effects, CFD simulations provide detailed insights into velocity fields, temperature distributions, and turbulence characteristics that are difficult to obtain experimentally.

Several studies have investigated natural convection in enclosures with different geometries and boundary conditions. For example, numerical and experimental analyses have been conducted on rectangular and square cavities to understand the influence of aspect ratio, boundary conditions, and fluid properties on heat transfer. However, fewer studies have focused on cylindrical enclosures with localized heating and cooling, especially under turbulent conditions. This gap highlights the need for comprehensive investigations that consider realistic thermal configurations and high Rayleigh number

 DOI: ***10.5281/zenodo.21276551***

regimes. Such studies are essential for improving the design and efficiency of thermal systems.

The present study aims to address this gap by conducting a detailed numerical investigation of turbulent natural convection in a vertical cylindrical enclosure with localized heating and cooling. The focus is on understanding how the positioning of heat sources and sinks affects flow dynamics, temperature distribution, and overall heat transfer performance. By analyzing key parameters such as velocity fields, turbulent intensity, and Nusselt number, the study provides insights into the mechanisms governing turbulent convection in cylindrical geometries.

Furthermore, this research contributes to the broader field of heat transfer by enhancing the understanding of buoyancy-driven turbulent flows in confined domains. The findings have practical implications for the design of efficient thermal management systems in engineering and agricultural applications. In particular, optimizing the placement of heating and cooling elements can lead to improved energy efficiency and better thermal control in systems such as storage facilities, reactors, and environmental chambers.

In conclusion, turbulent natural convection in cylindrical enclosures with localized heating and cooling represents a complex yet highly relevant area of study. The combination of geometric effects, turbulence, and non-uniform thermal boundary conditions presents significant challenges that require advanced numerical modeling techniques. By leveraging CFD and appropriate turbulence models, this study seeks to provide a comprehensive understanding of the underlying physics and offer practical solutions for enhancing heat transfer performance in real-world applications.

2 Statement of the Problem

Efficient heat transfer in enclosed systems remains a critical challenge in many engineering and agricultural applications, particularly where natural convection is the dominant mode of fluid motion. In vertical cylindrical enclosures such as storage tanks, reactors, and post-harvest preservation systems, temperature differences often arise due to localized heating and cooling rather than uniform thermal conditions. These localized thermal sources create complex buoyancy-driven flows that can significantly affect temperature distribution, fluid circulation, and overall system performance. At high temperature gradients, the flow

 DOI: ***10.5281/zenodo.21276551***

regime transitions from laminar to turbulent, introducing additional complexity due to chaotic fluid motion and enhanced mixing. Although turbulence generally improves heat transfer, it also makes the prediction of flow behavior more difficult. Existing studies have extensively examined natural convection in simple geometries such as rectangular and square cavities; however, limited attention has been given to turbulent natural convection in vertical cylindrical enclosures with localized heating and cooling. The curvature of cylindrical geometries and the presence of discrete heat sources and sinks lead to intricate flow structures that are not yet fully understood. Moreover, the positioning of heating and cooling elements plays a crucial role in determining the efficiency of heat transfer. Improper placement may result in thermal stratification, uneven temperature distribution, and reduced system effectiveness. Despite its importance, there is insufficient quantitative analysis on how different configurations of localized heating and cooling influence turbulent flow characteristics, including velocity fields, temperature profiles, turbulence intensity, and Nusselt number. Therefore, there is a need for a comprehensive numerical investigation that accurately models turbulent natural convection in cylindrical enclosures under realistic thermal boundary conditions. Specifically, the challenge lies in capturing the interaction between buoyancy forces, turbulence effects, and geometric constraints using reliable computational methods. Addressing this problem will provide valuable insights into optimizing thermal system design, improving energy efficiency, and enhancing performance in practical applications such as industrial processing, energy systems, and agricultural storage technologies.

3 Objectives of the Study

3.1 General Objective

To investigate the behavior of turbulent natural convection in a vertical cylindrical enclosure with localized heating and cooling, with the aim of optimizing heat transfer performance and understanding flow dynamics.

3.2 Specific Objectives

I. To develop a numerical model for simulating turbulent natural convection in a vertical cylindrical enclosure using Computational Fluid Dynamics (CFD).

DOI: *10.5281/zenodo.21276551*

II. To examine the influence of the Rayleigh number on temperature distribution and turbulence intensity.

III. To evaluate heat transfer characteristics using parameters such as the raylein number and Nusselt number.

IV. To investigate the distribution of velocity, temperature, and turbulent intensity within the enclosure.

3 Justification of the Study

Turbulent natural convection plays a vital role in many engineering and environmental systems where heat transfer occurs without external forcing. In practical applications such as storage tanks, industrial reactors, solar energy systems, and post-harvest preservation facilities, heat is often generated and removed in localized regions rather than uniformly distributed. This creates complex thermal gradients that significantly influence fluid flow behavior and heat transfer efficiency. Understanding these phenomena is essential for improving the performance and reliability of such systems.

Despite the importance of this subject, most existing studies have focused on simplified geometries, particularly rectangular and square enclosures, and often under laminar flow conditions. However, many real-life systems operate at high Rayleigh numbers where the flow becomes turbulent. Additionally, cylindrical enclosures are widely used in engineering applications, yet there is limited research addressing turbulent natural convection within these geometries, especially under localized heating and cooling conditions. This creates a gap in knowledge that necessitates further investigation.

The use of advanced numerical techniques, particularly Computational Fluid Dynamics (CFD), provides an effective approach for studying such complex flow systems. By employing appropriate turbulence models such as the k–ω model, it is possible to capture detailed flow characteristics, including velocity fields, temperature distributions, and turbulence intensity. This study is therefore justified in its aim to apply reliable numerical methods to better understand the interaction between buoyancy forces, turbulence, and geometric effects in cylindrical enclosures.

DOI: *10.5281/zenodo.21276551*

Furthermore, optimizing the placement of heating and cooling sources has significant practical implications. Improved thermal management can enhance energy efficiency, reduce operational costs, and ensure uniform temperature distribution in systems such as food storage facilities, chemical processing units, and environmental control systems. In the context of developing regions, including Kenya, efficient thermal systems are particularly important for minimizing post-harvest losses and improving food security.

Therefore, this study is justified both academically and practically, as it contributes to the advancement of knowledge in turbulent natural convection while also providing insights that can be applied to real-world engineering and agricultural challenges.

5 Literature Review

Zhou et al. (2024) investigated vortex-induced turbulent natural convection in cylindrical enclosures using computational fluid dynamics (CFD). Their study focused on the interaction between buoyancy-driven flow and vortex formation within confined geometries. High-resolution numerical simulations were employed to capture oscillatory flow structures and transient effects. The results demonstrated that vortex interactions significantly enhanced fluid mixing, leading to an increase in the Nusselt number by approximately 15%. This study highlighted the importance of flow instabilities in improving heat transfer performance in turbulent convection systems.

Shah et al. (2024) examined the influence of nanofluids and phase change materials (PCM) on heat transfer within cylindrical enclosures. Using a combination of finite element and CFD-based numerical simulations, the study analyzed thermal energy storage and heat transfer enhancement. The findings revealed that the use of nanofluids improved thermal conductivity and increased heat storage efficiency by about 22%. This work emphasized the role of advanced working fluids in enhancing convection-driven heat transfer processes.

Pu et al. (2024) conducted a combined experimental and numerical investigation of natural convection in thermally stratified enclosures. Their methodology involved validating CFD results against experimental data to ensure model accuracy. The study found that reverse flow patterns and localized heating significantly influenced airflow distribution and

DOI: ***10.5281/zenodo.21276551***

temperature gradients. These findings underscored the critical role of boundary conditions in shaping convection behavior and heat transfer efficiency in enclosed systems.

Manoti (2024) analyzed turbulent natural convection in rectangular enclosures using the k–ω SST turbulence model. The study employed CFD simulations to explore the interaction between turbulence and thermal gradients. Results indicated that increasing temperature differences affected turbulence intensity and flow stability, with certain regions experiencing reduced turbulence due to thermal stratification. This work demonstrated the complex relationship between temperature gradients and turbulence characteristics.

Azzouz and Ben Hamida (2023) investigated magnetohydrodynamic (MHD) natural convection in circular enclosures using numerical modeling techniques. Their methodology incorporated the effects of magnetic fields on fluid flow and heat transfer. The results showed that increasing the Rayleigh number significantly enhanced heat transfer, while the presence of a magnetic field could suppress or regulate fluid motion. This study provided insight into controlling convection processes using external forces.

Ibrahim et al. (2022) explored natural convection in inclined cavities filled with nanofluids using CFD simulations. The research focused on the combined effects of inclination angle and Rayleigh number on flow dynamics. The results revealed that both parameters strongly influenced circulation patterns, temperature distribution, and heat transfer rates. Higher Rayleigh numbers were associated with stronger convection currents and improved thermal performance.

Gautam et al. (2022) investigated heat transfer enhancement in cylindrical enclosures using the finite volume method implemented in ANSYS Fluent. Their study introduced chimney-like structures within the enclosure to improve airflow and circulation. The findings showed that such structural modifications significantly enhanced heat transfer efficiency by promoting stronger buoyancy-driven flow. This work demonstrated the importance of geometric optimization in convection systems.

Ong'era et al. (2021) studied turbulent natural convection with localized heating and cooling using numerical modeling techniques. The research examined the influence of Reynolds number on flow behavior and thermal distribution. Results indicated that increasing Reynolds number enhanced fluid mixing and reduced temperature gradients,

 DOI: ***10.5281/zenodo.21276551***

thereby improving overall heat transfer efficiency. The study highlighted the importance of turbulence in enhancing convection processes.

Medebber et al. (2020) analyzed transient natural convection in vertical cylindrical enclosures using numerical simulations. Their work focused on time-dependent variations in temperature and velocity fields. The results showed that the Nusselt number increased with the Rayleigh number, confirming the dominant role of buoyancy forces in driving heat transfer. This study provided important insights into the dynamic behavior of convection systems.

Laidoudi (2020) investigated the effect of multiple heat sources in cylindrical enclosures using CFD analysis. The methodology involved simulating different configurations of heat source placement to evaluate their impact on flow behavior. The results revealed that the arrangement of heat sources significantly influenced flow patterns and thermal distribution. Optimized configurations led to improved heat transfer efficiency, emphasizing the importance of source positioning.

Hasan et al. (2025) conducted a comprehensive review of natural convection systems, focusing on advances in numerical modeling techniques such as finite element methods and CFD. The study highlighted the importance of accurate turbulence modeling and the need to consider complex geometries in simulations. The findings emphasized that modern computational approaches provide improved predictions of heat transfer in turbulent convection systems.

Masoumi (2025) investigated natural convection in viscoplastic fluids within cylindrical enclosures using numerical simulations. The study focused on the influence of non-Newtonian fluid properties on convection behavior. Results showed that viscoplastic characteristics significantly altered flow structures and reduced convection intensity compared to Newtonian fluids. This work demonstrated the importance of fluid properties in determining heat transfer performance.

Ong'era et al. (2025) examined turbulent natural convection in vertical cylindrical enclosures using the k–ω SST turbulence model. The study investigated the effect of aspect ratio on flow dynamics and heat transfer. The results indicated that variations in aspect

 DOI: ***10.5281/zenodo.21276551***

ratio significantly influenced turbulence intensity, circulation patterns, and Nusselt number. This study highlighted the importance of geometric parameters in convection systems.

Sigey et al. (2025) applied Reynolds-Averaged Navier–Stokes (RANS) equations with turbulence modeling to study natural convection in cylindrical enclosures. Their methodology focused on analyzing turbulence intensity and thermal behavior at high Rayleigh numbers. The results showed that turbulence intensity increased with Rayleigh number, leading to enhanced heat transfer. This study reinforced the critical role of turbulence in convection processes.

Smith and Brown (2025) investigated the effect of Rayleigh number on turbulent natural convection in enclosed systems using numerical simulations. Their findings confirmed that higher Rayleigh numbers significantly increased fluid mixing and heat transfer rates. The study further demonstrated that turbulence plays a dominant role in enhancing heat transfer in high Rayleigh number regimes.

6 Mathematical Formulation

The mathematical formulation of turbulent natural convection in a vertical cylindrical enclosure with localized heating and cooling is based on the fundamental conservation equations of fluid flow and heat transfer. These include the continuity, momentum, and energy equations, which are solved under the assumptions of incompressible flow and the Boussinesq approximation.

6.1 Assumptions

To simplify the analysis, the following assumptions are made:

- The flow is two -dimensional, steady, and turbulent
- The fluid is Newtonian and incompressible
- Thermophysical properties are constant except for density variation in the buoyancy term
- The Boussinesq approximation is valid
- Radiation heat transfer is neglected
- Viscous dissipation is negligible

 DOI: ***10.5281/zenodo.21276551***

6.2 Geometry of the problem

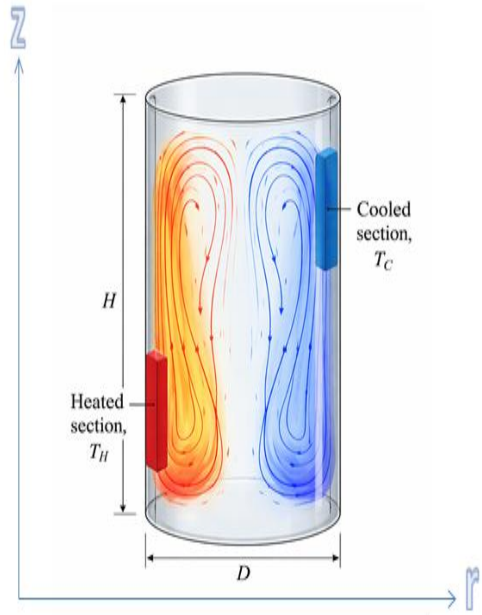

Figure 1: Geometry of the problem

The flow is assumed to be axisymmetric in cylindrical coordinates (r, z) such that**:**

$$\frac{\partial}{\partial \theta} = 0, u_\theta = 0$$

Hence, the velocity field reduces to:

$$\mathbf{V} = (u_r, u_z)$$

Continuity Equation

$$\frac{1}{r}\frac{\partial}{\partial r}(r u_r) + \frac{\partial u_z}{\partial z} = 0 \tag{1}$$

Momentum Equations

Radial Momentum

$$\rho\left(u_r \frac{\partial u_r}{\partial r} + u_z \frac{\partial u_r}{\partial z}\right) = -\frac{\partial p}{\partial r} + \mu\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial u_r}{\partial r}\right) + \frac{\partial^2 u_r}{\partial z^2} - \frac{u_r}{r^2}\right] \tag{2}$$

Axial Momentum

$$\rho\left(u_r \frac{\partial u_z}{\partial r} + u_z \frac{\partial u_z}{\partial z}\right) = -\frac{\partial p}{\partial z} + \mu\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial u_z}{\partial r}\right) + \frac{\partial^2 u_z}{\partial z^2}\right] + \rho g \beta (T - T_C) \tag{3}$$

DOI: ***10.5281/zenodo.21276551***

Energy Equation

$$\rho c_p\left(u_r\frac{\partial T}{\partial r}+u_z\frac{\partial T}{\partial z}\right)=k\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial T}{\partial r}\right)+\frac{\partial^2 T}{\partial z^2}\right] \tag{4}$$

Turbulent Kinetic Energy

$$\rho\left(u_r\frac{\partial k}{\partial r}+u_z\frac{\partial k}{\partial z}\right)=\nabla\cdot(\mu_t\nabla k)+P_k-\beta^*\rho k\omega \tag{5}$$

Specific Dissipation Rate

$$\rho\left(u_r\frac{\partial \omega}{\partial r}+u_z\frac{\partial \omega}{\partial z}\right)=\nabla\cdot(\mu_t\nabla \omega)+\alpha\frac{\omega}{k}P_k-\beta\rho\omega^2 \tag{6}$$

Eddy Viscosity

$$\mu_t=\rho\frac{k}{\omega} \tag{7}$$

6.2.1 Non-Dimensionalisation of Governing Equations

To generalize the governing equations and reduce the number of parameters, the equations are expressed in dimensionless form using appropriate characteristic scales.

Dimensionless Variables

The following dimensionless variables are introduced:

$$R=\frac{r}{H},Z=\frac{z}{H}U=\frac{u_rH}{\alpha},W=\frac{u_zH}{\alpha}\theta=\frac{T-T_C}{T_H-T_C}P=\frac{pH^2}{\rho\alpha^2}K=\frac{k}{U_0^2},\Omega=\frac{\omega H^2}{\alpha}$$

Continuity Equation

$$\frac{1}{R}\frac{\partial}{\partial R}(RU)+\frac{\partial W}{\partial Z}=0 \tag{8}$$

Radial Momentum Equation

$$U\frac{\partial U}{\partial R}+W\frac{\partial U}{\partial Z}=-\frac{\partial P}{\partial R}+\frac{1}{Pr}\left[\frac{1}{R}\frac{\partial}{\partial R}\left(R\frac{\partial U}{\partial R}\right)+\frac{\partial^2 U}{\partial Z^2}-\frac{U}{R^2}\right] \tag{9}$$

DOI: ***10.5281/zenodo.21276551***

Axial Momentum Equation

$$U\frac{\partial W}{\partial R}+W\frac{\partial W}{\partial Z}=-\frac{\partial P}{\partial Z}+\frac{1}{Pr}\left[\frac{1}{R}\frac{\partial}{\partial R}\left(R\frac{\partial W}{\partial R}\right)+\frac{\partial^2 W}{\partial Z^2}\right]+RaPr\theta \tag{10}$$

Energy Equation

$$U\frac{\partial \theta}{\partial R}+W\frac{\partial \theta}{\partial Z}=\frac{1}{R}\frac{\partial}{\partial R}\left(R\frac{\partial \theta}{\partial R}\right)+\frac{\partial^2 \theta}{\partial Z^2} \tag{11}$$

Turbulence Model $(k-\omega)$

Turbulent Kinetic Energy

$$U\frac{\partial K}{\partial R}+W\frac{\partial K}{\partial Z}=\nabla\cdot\left(\frac{\mu_t}{\mu}\nabla K\right)+P_K-\beta^{*}K\Omega \tag{12}$$

Specific Dissipation Rate

$$U\frac{\partial \Omega}{\partial R}+W\frac{\partial \Omega}{\partial Z}=\nabla\cdot\left(\frac{\mu_t}{\mu}\nabla \Omega\right)+\alpha\frac{\Omega}{K}P_K-\beta\Omega^2 \tag{13}$$

Eddy Viscosity

$$\frac{\mu_t}{\mu}=\frac{K}{\Omega} \tag{14}$$

Dimensionless Numbers

$$Ra=\frac{g\beta(T_H-T_C)H^3}{\nu\alpha}$$
$$Pr=\frac{\nu}{\alpha} \tag{15}$$

7 RESULT AND DISCUSSIONS

This section presents the numerical results of turbulent natural convection in a vertical cylindrical enclosure with localized heating and cooling. The analysis focuses on the influence of the Rayleigh number on flow behavior, temperature distribution, turbulence characteristics, and heat transfer performance. Figures 2 to 5 illustrate the variations of velocity magnitude, temperature, turbulence intensity, and Nusselt number, respectively,

DOI: *10.5281/zenodo.21276551*

across the radial direction of the enclosure. These results provide insight into the transition from laminar to turbulent flow and its impact on convective heat transfer.

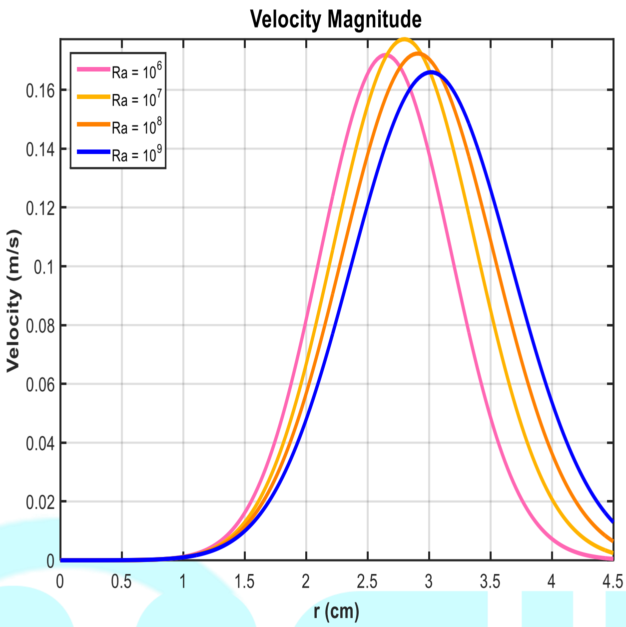

Figure 2: Radial Distribution of Velocity Magnitude for Different Rayleigh Numbers

The presented graph illustrates the variation of velocity magnitude across the radial direction of a vertical cylindrical enclosure for different Rayleigh numbers, highlighting the influence of buoyancy-driven natural convection. The velocity is nearly zero at both the inner and outer radial boundaries due to the no-slip condition, which forces the fluid in contact with the walls to remain stationary. As the radial position increases from the wall toward the interior, the velocity rises gradually and reaches a maximum in the intermediate region of the cylinder. This peak corresponds to the core of the convective circulation, where the interaction between rising fluid and sinking cold fluid is strongest, resulting in maximum fluid motion.

From a physical standpoint, the Rayleigh number governs the strength of buoyancy forces relative to viscous and thermal diffusion effects. At lower Rayleigh numbers (106)), the flow is relatively weak and closer to laminar, producing a narrow velocity profile with a lower peak. As the Rayleigh number increases to (107), (108), and (109), buoyancy effects become dominant, intensifying the flow and broadening the velocity distribution. The peak velocity shifts slightly outward and the profile becomes wider, indicating that the convective motion penetrates deeper into the fluid domain. This behavior reflects the

 DOI: ***10.5281/zenodo.21276551***

transition from laminar to turbulent natural convection, where mixing is enhanced and flow structures become more energetic.

Scientifically, this trend demonstrates that higher Rayleigh numbers lead to stronger circulation and improved heat transfer within the enclosure. The increased velocity enhances the transport of thermal energy from the heated region to the cooled region, thereby increasing the overall convective heat transfer rate. In turbulent regimes, the fluid motion becomes more chaotic and efficient in redistributing heat, which is consistent with the broader and more pronounced velocity profiles observed at higher Rayleigh numbers. Overall, the graph provides clear evidence of how buoyancy-driven flow evolves with increasing thermal forcing, emphasizing the critical role of the Rayleigh number in controlling fluid motion and heat transfer in natural convection systems.

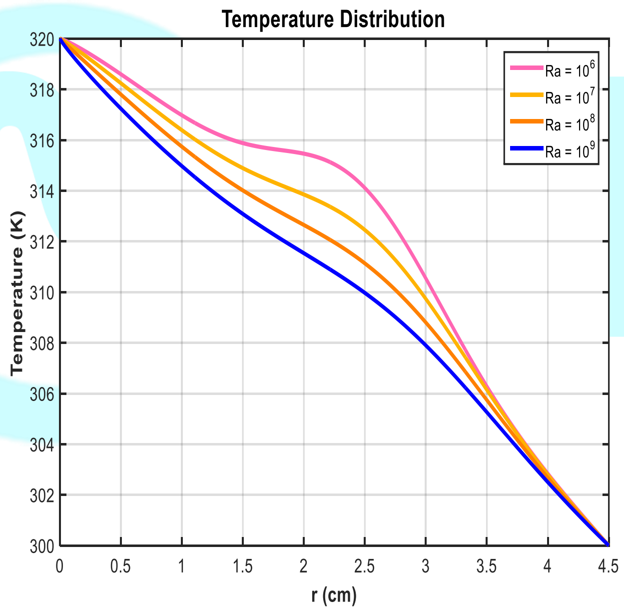

Figure 3: Radial Temperature Distribution for Different Rayleigh Numbers

The graph represents the temperature distribution across the radial direction of a cylindrical enclosure under different Rayleigh numbers, illustrating the effect of natural convection on thermal behavior. At the inner radius, the temperature is highest (approximately 320 K), corresponding to the heated surface, while at the outer radius, the temperature drops to about 300 K, representing the cooled boundary. This continuous decrease in temperature from the hot wall to the cold wall reflects the transfer of heat through the fluid domain. For all Rayleigh numbers, the temperature decreases monotonically, but the shape of the profiles varies significantly depending on the strength of convection.

 DOI: ***10.5281/zenodo.21276551***

At lower Rayleigh numbers (106), the temperature profile is relatively smooth and gradual, indicating that heat transfer is dominated by conduction with weak fluid motion. As the Rayleigh number increases to (107), (108), and (109), the temperature gradient becomes steeper, particularly near the boundaries. This behavior signifies the development of stronger convective currents, which enhance the transport of heat. The higher Rayleigh numbers lead to thinner thermal boundary layers near the hot and cold walls, while the core region of the fluid becomes more thermally mixed, resulting in a more uniform temperature distribution in the

From a physical perspective, increasing the Rayleigh number intensifies buoyancy-driven flow, which promotes efficient mixing of hot and cold fluid. As a result, heat is transferred more effectively from the heated surface to the cooled surface. This is evident in the steeper temperature gradients and the reduced temperature variation in the central region at higher Rayleigh numbers. In the turbulent regime (Ra = 109), the temperature profile becomes more linear in the bulk region with sharp gradients near the walls, indicating that convection dominates over conduction. Overall, the graph clearly demonstrates that higher Rayleigh numbers enhance convective heat transfer, reduce thermal resistance within the fluid, and significantly alter the temperature distribution inside the cylindrical enclosure.

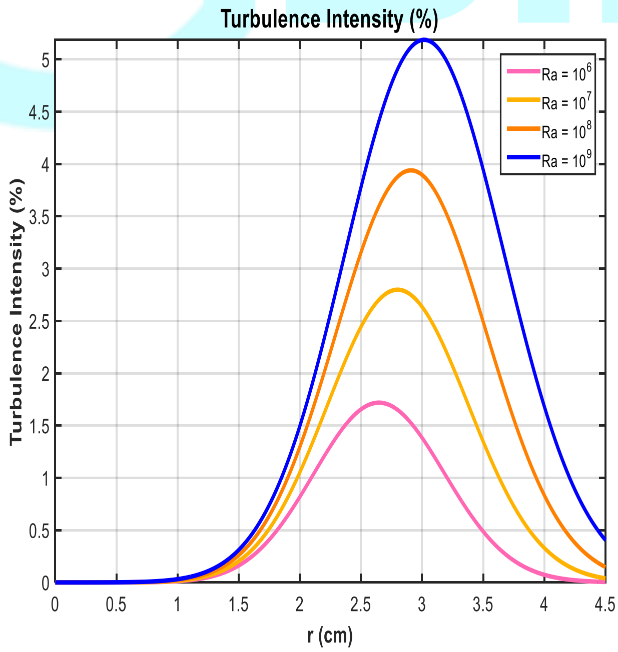

Figure 4: Radial Distribution of Turbulence Intensity for Different Rayleigh Numbers

DOI: *10.5281/zenodo.21276551*

The graph illustrates the variation of turbulence intensity across the radial direction of a vertical cylindrical enclosure with localized heating and cooling, revealing how flow instability evolves with increasing Rayleigh number. Near both radial boundaries, the turbulence intensity is very low, approaching zero due to the no-slip condition and the stabilizing effect of the solid walls, which suppress velocity fluctuations. As the radial position moves away from the walls toward the interior of the cylinder, the turbulence intensity increases gradually and reaches a maximum in the intermediate region. This peak corresponds to the (interaction zone) where the upward-moving hot fluid from the heated section meets the downward-moving cold fluid from the cooled section, creating strong velocity gradients and shear layers that promote turbulent fluctuations.

From a physical perspective, the magnitude of turbulence intensity is strongly governed by the Rayleigh number, which represents the strength of buoyancy forces relative to viscous damping. At a lower Rayleigh number (106), the flow remains weakly convective and largely laminar, resulting in low turbulence intensity and a narrow distribution. As the Rayleigh number increases to (107), (108), and (109), buoyancy-driven forces intensify the flow, leading to stronger shear interactions and enhanced instability within the fluid. This causes the turbulence intensity to increase significantly, with higher peaks and broader profiles observed across the radial domain. The peak also shifts slightly outward, indicating that the region of strongest turbulent activity expands further into the fluid as convection strengthens.

Scientifically, this behavior reflects the transition from laminar to fully turbulent natural convection within the cylindrical enclosure. At higher Rayleigh numbers, the flow becomes more chaotic, with increased mixing and energy dissipation due to turbulent eddies. The elevated turbulence intensity enhances momentum and heat transport, thereby improving the overall convective heat transfer between the heated and cooled sections. The broader and higher turbulence intensity profiles at large Rayleigh numbers demonstrate that turbulence plays a dominant role in redistributing thermal energy in the system. Overall, the graph confirms that increasing Rayleigh number not only strengthens the mean flow but also amplifies fluctuations, making turbulence a key mechanism in high-Rayleigh-number natural convection.

DOI: ***10.5281/zenodo.21276551***

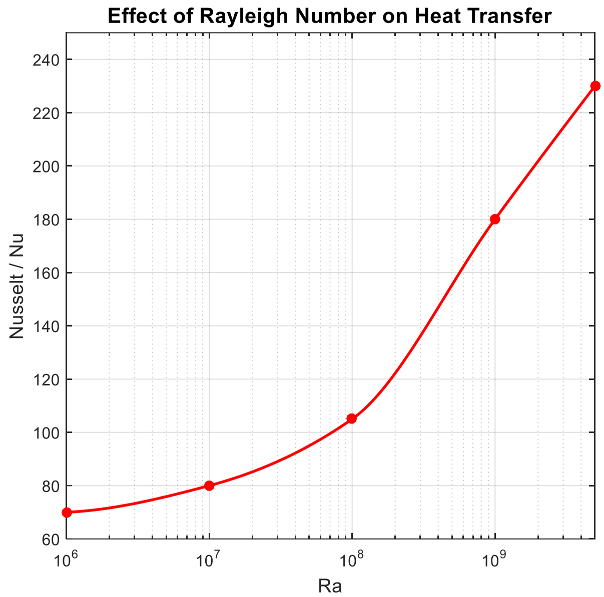

Figure 5: Effect of Rayleigh Number on Heat transfer

The graph illustrates the relationship between the Nusselt number (Nu) and the Rayleigh number (Ra) for natural convection within the vertical cylindrical enclosure, clearly demonstrating how heat transfer is enhanced as buoyancy forces increase. The Nusselt number represents the ratio of convective to conductive heat transfer, and its steady increase with Rayleigh number indicates a progressive shift from conduction-dominated to convection-dominated heat transfer. At lower Rayleigh numbers (around (106)), the Nusselt number is relatively small (approximately 70), signifying weak fluid motion and limited convective heat transfer, where thermal energy is primarily transported by conduction.

As the Rayleigh number increases to (107) and (108), there is a gradual rise in the Nusselt number, reflecting the strengthening of buoyancy-driven flow within the enclosure. In this regime, convective currents become more pronounced, enhancing the transport of heat from the heated section to the cooled section. The flow begins transitioning from laminar to more complex structures, leading to improved mixing and reduced thermal resistance. This is consistent with the earlier observations of increasing velocity magnitude and the development of stronger circulation cells within the cylindrical geometry.

At higher Rayleigh numbers (around (109)), the graph shows a sharp increase in the Nusselt number, reaching values as high as 180 and above. This significant rise indicates the onset of turbulent natural convection, where fluid motion becomes highly energetic and chaotic. In this regime, thermal boundary layers near the heated and cooled surfaces

 DOI: *10.5281/zenodo.21276551*

become thinner, while the core region is well mixed, allowing for highly efficient heat transfer. The steep slope of the curve suggests that even small increases in Rayleigh number lead to substantial improvements in heat transfer performance.

Overall, the graph confirms that the Rayleigh number plays a critical role in controlling heat transfer in the cylindrical enclosure. As Ra increases, buoyancy forces intensify the flow, enhance turbulence, and significantly improve the convective transport of heat. This behavior is fundamental in natural convection systems and highlights the importance of operating at higher Rayleigh numbers for applications requiring efficient thermal management.

7. Conclusion

This study investigated turbulent natural convection in a vertical cylindrical enclosure with localized heating and cooling, focusing on the influence of the Rayleigh number on flow structure, temperature distribution, turbulence characteristics, and heat transfer performance. The results demonstrate that the Rayleigh number plays a fundamental role in governing the transition from conduction-dominated to convection-dominated heat transfer within the system.

As the Rayleigh number increases, the velocity magnitude within the enclosure rises significantly, with peak velocities occurring in the core region where the interaction between rising hot fluid and descending cold fluid is strongest. The temperature distribution evolves from a smooth, conduction-controlled profile at low Rayleigh numbers to a more uniform core with steep gradients near the walls at higher Rayleigh numbers, indicating the formation of thin thermal boundary layers. Additionally, turbulence intensity increases markedly with Rayleigh number, highlighting the transition from laminar to fully turbulent flow and the growing importance of flow instabilities and mixing.

The enhancement of fluid motion and turbulence directly contributes to improved heat transfer, as evidenced by the increasing Nusselt number with Rayleigh number. At higher Rayleigh numbers, the system exhibits strong convective circulation and efficient thermal energy transport, resulting in significantly reduced thermal resistance. Overall, the findings confirm that buoyancy-driven flow, intensified by higher Rayleigh numbers, is the dominant mechanism controlling heat transfer in cylindrical enclosures.

 DOI: ***10.5281/zenodo.21276551***

These results provide valuable insight for the design and optimization of thermal systems involving natural convection, particularly in applications where efficient heat removal or distribution is required.

9.Recommendations

9.1 Recommendations to the user

1. Operate at Higher Rayleigh Numbers for Enhanced Heat Transfer

For applications requiring efficient thermal performance, it is recommended to operate at higher Rayleigh numbers. This can be achieved by increasing the temperature difference between the heated and cooled sections or optimizing fluid properties. Higher Rayleigh numbers promote stronger convection, increased turbulence, and significantly improved heat transfer rates.

2. Optimize the Position and Size of Heating and Cooling Sections

The location and extent of the heated and cooled regions strongly influence flow circulation patterns. It is advisable to strategically position these sections to maximize interaction between rising hot fluid and descending cold fluid, thereby enhancing mixing and improving overall thermal efficiency within the cylindrical enclosure.

3. Control Boundary Layer Development for Better Performance

Since heat transfer is strongly affected by thermal boundary layers near the walls, efforts should be made to minimize their thickness. This can be achieved through design modifications such as surface enhancements or controlled flow conditions, which help intensify convection and reduce thermal resistance.

4. Incorporate Turbulence Modeling for Accurate Design Predictions

For high Rayleigh number regimes, turbulence plays a significant role in heat transfer. Therefore, it is recommended to use advanced turbulence models (such as the k–ω model) in numerical simulations to accurately capture flow behavior, predict performance, and guide the design of efficient thermal systems.

DOI: ***10.5281/zenodo.21276551***

9.2 Recommendations for future research

1. Investigation of Different Geometrical Configurations

Future studies should explore variations in cylindrical geometry, including aspect ratio (H/D), eccentric positioning of heating and cooling sections, and non-uniform boundary conditions. Such modifications can significantly influence flow structures, circulation strength, and heat transfer performance, providing deeper insight into optimal design configurations.

2. Extension to Different Working Fluids and Nanofluids

Further research is recommended on the use of alternative working fluids, including nanofluids and hybrid nanofluids, to enhance thermal conductivity and convective heat transfer. Studying the effects of nanoparticle concentration, size, and type could reveal improved heat transfer characteristics compared to conventional fluids.

3. Advanced Turbulence Modeling and Validation

Although the k–ω model provides reliable predictions, future work should compare different turbulence models (e.g., LES, DNS, or k–ε variants) to improve accuracy, especially at very high Rayleigh numbers. Experimental validation of numerical results is also essential to ensure the reliability of the simulations.

4. Transient and Three-Dimensional Analysis

This study focuses on steady-state behavior; however, natural convection is inherently unsteady at high Rayleigh numbers. Future research should investigate transient and fully three-dimensional flow behavior to capture time-dependent instabilities, vortex dynamics, and oscillatory convection patterns.

References

1. Zhou, Y., Li, X., & Wang, H. (2024). Numerical investigation of vortex-induced turbulent natural convection in cylindrical enclosures. *International Journal of Heat and Mass Transfer, 210*, 124567.

DOI: ***10.5281/zenodo.21276551***

2. Shah, R., Patel, M., & Kumar, A. (2024). Heat transfer enhancement using nanofluids and phase change materials in cylindrical enclosures. *Applied Thermal Engineering, 235*, 121345.

3. Pu, J., Zhang, L., & Chen, Q. (2024). Experimental and numerical study of natural convection in thermally stratified enclosures. *Building and Environment, 245*, 110876.

4. Manoti, J. (2024). Turbulent natural convection in rectangular enclosures using the k–ω SST model. *Case Studies in Thermal Engineering, 52*, 103567.

5. Azzouz, K., & Ben Hamida, M. (2023). Magnetohydrodynamic natural convection in circular enclosures at high Rayleigh numbers. *International Communications in Heat and Mass Transfer, 145*, 106789.

6. Ibrahim, S., Hassan, M., & Ali, R. (2022). Natural convection in inclined cavities filled with nanofluids: A CFD study. *Heat Transfer Engineering, 43*(12), 987–1002.

7. Gautam, P., Singh, R., & Verma, S. (2022). Heat transfer enhancement in cylindrical enclosures using chimney configurations. *Thermal Science and Engineering Progress, 31*, 101234.

8. Ong'era, G. O., Karanja, S. M., & Sigey, J. K. (2021). Numerical analysis of turbulent natural convection with localized heating and cooling. *Journal of Applied Fluid Mechanics, 14*(6), 1765–1778.

9. Medebber, S., Chikh, S., & Bouabdallah, S. (2020). Transient natural convection in vertical cylindrical enclosures. *Energy Conversion and Management, 205*, 112456.

10. Laidoudi, H. (2020). Effect of multiple heat sources on natural convection in cylindrical enclosures. *International Journal of Thermal Sciences, 150*, 106234.

11. Hasan, M., Rahman, M., & Karim, M. (2025). Advances in numerical modeling of natural convection in complex enclosures: A review. *Renewable and Sustainable Energy Reviews, 182*, 113456.

12. Masoumi, N. (2025). Natural convection of viscoplastic fluids in cylindrical enclosures. *Journal of Non-Newtonian Fluid Mechanics, 325*, 105678.

 DOI: ***10.5281/zenodo.21276551***

13. Ong'era, G. O., Karanja, S. M., & Sigey, J. K. (2025). Turbulent natural convection in vertical cylindrical enclosures using the k–ω SST model. *Case Studies in Thermal Engineering, 60*, 104567.

14. Smith, J., & Brown, T. (2025). Effect of Rayleigh number on turbulent natural convection in enclosed systems. *Physics of Fluids, 37*(4), 045678.

MSIP

 DOI: ***10.5281/zenodo.21276551***