

MHD Mixed Convection Control in an Actuated Vertically Rectangular Vented Cavity with a Rotating Core Cylinder

Pushpa Akhtar

Ph.D. Fellow, Department of Mathematics, Khulna University of Engineering & Technology, Khulna-9203, Bangladesh <https://orcid.org/0009-0003-6972-6299>

M. M. Touhid Hossain

mthosain@math.kuet.ac.bd

Department of Mathematics, Khulna University of Engineering & Technology, Khulna-9203, Bangladesh
<https://orcid.org/0000-0001-5068-9844>

Md. Abdul Alim

Department of Mathematics, Bangladesh University of Engineering and Technology, Dhaka-1000, Bangladesh <https://orcid.org/0000-0001-9706-2221>

Mohammad Mokaddes Ali

Department of Mathematics, Mawlana Bhashani Science and Technology University, Santosh, Tangail-1902, Bangladesh

Research Article

Keywords: Mixed convection, Lorentz's force, nanofluid (CuO), rotating cylinder, finite element method, and vertical vented rectangular enclosure

Posted Date: July 13th, 2026

DOI: <https://doi.org/10.21203/rs.3.rs-10311369/v1>

License:  This work is licensed under a Creative Commons Attribution 4.0 International License.

[Read Full License](#)

Additional Declarations: The authors declare no competing interests.

MHD Mixed Convection Control in an Actuated Vertically Rectangular Vented Cavity with a Rotating Core Cylinder

Pushpa Akhtar¹[0009-0003-6972-6299], M. M. Touhid Hossain²[0000-0001-5068-9844], *, Md. Abdul Alim³[0000-0001-9706-2221] and Mohammad Mokaddes Ali⁴

¹ Ph.D. Fellow, Department of Mathematics, Khulna University of Engineering & Technology, Khulna-9203, Bangladesh, email: pushpaakhter92@gmail.com

² Department of Mathematics, Khulna University of Engineering & Technology, Khulna-9203, Bangladesh, email: mthosain@math.kuet.ac.bd

³ Department of Mathematics, Bangladesh University of Engineering and Technology, Dhaka-1000, Bangladesh, email: a0alim@gmail.com

⁴ Department of Mathematics, Mawlana Bhashani Science and Technology University, Santosh, Tangail-1902, Bangladesh, email: mmali309@gmail.com

*Corresponding author

Abstract:

This study employs the Galerkin FE approach to assess the impact of magnetic field (Lorentz force) on the MHD mixed convection dynamics of CuO-water nanofluid flow within a vented vertical cavity. The system incorporates a centrally positioned rotating cylinder with an entry port at the lower-left corner and a discharge port at the upper-right boundary. A comprehensive parametric investigation was conducted by systematically varying key dimensionless parameters: the Hartmann number ($0 \leq Ha \leq 100$), the Reynolds number ($10 \leq Re \leq 500$), the cylinder's rotational speed ($\omega = 0$ to 50), and the nanoparticle volume fractions ($0\% \leq \phi \leq 6\%$). The resulting fluid behaviour and thermal characteristics were evaluated through detailed visualizations of streamlines and isotherms, alongside quantitative tracking of average temperature and the overall Nusselt number (Nu). The computational results demonstrate that higher Re significantly intensifies heat transfer rates especially when the cylinder undergoes counter-clockwise rotation. Furthermore, adjusting Ha and ϕ proves vital for optimizing thermal efficiency. Notably, severe magnetic field suppression ($Ha = 100$) induces approximately 47.90% reduction in heat transfer compared to the non-magnetic baseline for $Re = 500$. Conversely, introducing an anticlockwise rotation boosts localized heat transfer near the bottom ends by nearly 13.09% relative to the stationary configuration ($\omega = 0$). For clockwise cylinder rotation ($\omega = 10$) yields a massive 195.74% increase in the local Nusselt number as Re escalates from 10 to 500. These insights establish a robust design framework for high-efficiency thermal management systems, with direct implications for advanced compact radiators, electronic thermal controllers, and next-generation energy conversion technologies.

Keywords: Mixed convection, Lorentz's force, nanofluid (CuO), rotating cylinder, finite element method, and vertical vented rectangular enclosure.

1. Introduction

Mixed convection describes the situation where both forced convection and natural convection act together during heat transfer in fluid flow systems. In recent years, this phenomenon has attracted significant interest because of its broad applications in modern engineering systems, including enclosures, cavities, and channels [1] – [3]. [4] explored turbulent mixed-convection dynamics in a vented chamber through numerical simulations. Their work established that altering the inlet and outlet configurations substantially impacts how effectively the space is ventilated. Efficient thermal management remains a critical challenge in many industrial and thermal engineering systems. Traditional fluids like water, ethylene glycol, and engine oil have comparatively

low thermal conductivity, which restricts the improvement of heat transfer efficiency in compact engineering devices. To overcome these limitations, researchers have introduced various advanced techniques. Among them, nanotechnology has emerged as an effective solution through the development of nanofluids. A nanofluid is a stable suspension of high thermal conductivity nanoparticles dispersed uniformly within a convectational base fluid. The presence of nanoparticles substantially improves the thermal conductivity and heat transfer characteristics of the base fluid, thereby enhancing overall system performance. Research specified that the thermal properties of traditional base liquids can remarkably be improved by suspending various metallic or non-metallic particles at the nanoscale [5-7]. The performance of Al_2O_3 -water based nano liquids during mixed convection inside an open-ended square chamber has been analysed through computational modelling [8]. This particular study emphasized the optimization of the outlet port position to achieve superior thermal results. Findings indicated that locating the outlet at the mid-height of the vertical boundaries resulted in a diminished heat transfer rate. Furthermore, the dynamics of buoyancy-induced flow and heat transport within a differentially heated vertical chamber containing Al_2O_3 -water nanofluid were thoroughly investigated by [9]. By employing the Galerkin-based finite element method, the researchers examined how the geometric aspect ratio, Prandtl number, and nanoparticle volume fraction collectively influence the system. Their analysis revealed that nanoparticles can either promote or inhibit heat transfer, depending stringently on the governing physical parameters. A numerical exploration of laminar flow and combined convection within a square geometry was reported by [10], where nanoparticles were used to enhance fluid motion driven by a sliding wall. Their mathematical framework utilized Buongiorno's nanofluid formulation, which was subsequently solved using a second-order finite difference technique. The vertical boundaries were assumed to be adiabatic to eliminate heat leakage, whereas the top and bottom moving walls were subjected to uniform nanoparticle distribution and fixed thermal settings. Their findings indicated a strong correlation between the governing dimensionless parameters and the resulting transport phenomena, as evidenced by the variations in streamlines, thermal isotherms, and particle distribution contours. [11] employed numerical simulations to explore the combined convection behavior of Al_2O_3 -water suspensions inside a rectangular chamber featuring vents. In their results configuration, a uniform high temperature was applied to the right vertical wall, while cooling was achieved through suction or injection of fluid to illustrate the mixed convection mechanism. Their results indicated that raising the nanoparticle concentration enhances the heat transfer performance within the cavity.

Moreover, extracting fluid from the upper left corner of the enclosure was found to be an effective strategy for maximizing thermal performance. Magnetohydrodynamic (MHD) mixed convection has attracted considerable research interest because of its extensive applications in electronic cooling, heat exchangers, nuclear reactor systems, solar collectors, Micro-Electro-Mechanical Systems (MEMS), galvanization processes, crystal growth, and various material, chemical, and food processing operations. When a conducting fluid interacts with an applied magnetic field, a resistive force referred to as the Lorentz force arises, that affects the fluid motion by acting opposite to the flow direction. Consequently, the interaction between magnetic fields and mixed convection in nanofluid-filled enclosures and channels has become an increasingly important research area [12]–[14]. The use of magnetic fields is recognized as an effective technique for controlling and optimizing convective heat transfer processes. [15] performed a numerical analysis of MHD convective heat transfer for CuO -water nanofluid employing the Lattice Boltzmann method. The results of this research indicated that the Lorentz

force limits fluid motion, reducing convective transport while increasing the contribution of heat conduction. [16] analyzed how MHD interacts with mixed convection in a three-dimensional micro-passage using water-based Al_2O_3 nanofluids. Their findings demonstrated that enhancing the magnetic field intensity and increasing the solid volume fraction leads to a higher Nusselt number, thereby boosting the overall heat exchange rate. [17] examined mixed magnetohydrodynamics convective flow within a ventilated square cavity containing Cu-water nanofluid, considering different inlet and outlet port arrangements. Their objective was to determine the optimal magnetic and flow conditions for enhanced thermal performance. The study suggested that a Hartmann number of approximately 30 yields favourable heat transfer characteristics. Furthermore, the results indicated that a decline in the average Nusselt number occurs with higher Richardson and Hartmann numbers, as well as an increase in the inlet port's vertical placement. In cavity flow systems, convective heat transfer can be effectively modified by incorporating stationary or rotating shafts, rotary heat exchangers, and drilling operations in oil extraction systems. The introduction of fixed or rotating cylindrical elements substantially alters the fluid dynamics and associated heat transfer mechanisms. [18] numerically analyzed steady, laminar, incompressible combined convection inside a ventilated rectangular enclosure containing a centrally positioned heat-conducting cylindrical obstacle. The interaction between forced convection and buoyancy-driven flow was examined through streamline and isotherm distributions. Their findings revealed that the flow behavior, thermal patterns, average Nusselt number, mean fluid temperature are highly sensitive to the Richardson number (the governing mixed convection parameter) and the cavity aspect ratio. [19] numerically analyzed heat transfer behavior in a square cavity containing water-copper nanofluid, where a heated circular obstacle acted as an internal heat source and an outlet port was incorporated in the enclosure. Their study emphasized that both the inclusion of a hot internal cylinder and the strategic positioning of the outlet opening significantly improve the thermal performance of the system. [20] performed a numerical study on the transient behaviour of a conducting fluid moving through a vented square enclosure, where the flow interacts with two identical heated circular bodies. A uniform horizontal magnetic field was imposed perpendicular to the heated vertical wall, while the inlet and outlet openings were located at the midpoints of the left and right walls, respectively. Their results showed that depending on the governing control parameters, the system may exhibit steady, periodic oscillatory or chaotic flow behaviors. By applying a magnetic field (Hartmann number), we can induce a damping effect on the circulation. This magnetic resistance promotes overall stability in the flow field, ultimately helping the system reach a state of equilibrium. [21] extended this analysis to a three-dimensional vented cavity filled with CuO-water nanofluid under mixed convection, incorporating a rotating circular cylinder and surface corrugation effect while a uniform magnetic field was applied. The results demonstrated that the imposed magnetic field enhances the average heat transfer rate by suppressing recirculation zones, particularly near the lower region of the inlet port. Furthermore, introducing nanoparticles enhances the thermal and electrical conductivities of the base liquid, thereby intensifying the interaction with the magnetic field and boosting the overall convective performance. [22] investigated combined convective heat transfer within a square cavity containing a centrally positioned rotating cylindrical element that acts both as a thermally conductive body and an internal heat source. The authors investigated the role of the cylinder's diameter and rotational direction in governing the thermal characteristics and cooling effectiveness of the flow. Their findings suggest that changing the cylinder's rotational orientation has a negligible impact on the system's total thermal behaviour inside the enclosure. Observations suggest

that elevating the Hartmann number (Ha) yields a more consistent flow structure. Investigation of [23] utilizes the FEM to analyze the mixed convection of an electrically conducting fluid within a vertical lid-driven rectangular cavity. Their model featured a hollow circular obstruction at the center and a semi-circular heat source positioned at the midpoint of the lower boundary. The stabilizing influence of the magnetic field ensures this uniformity remains consistent whether the cylinder is spinning or remains at rest. Conversely, at lower Hartmann numbers, intense and localized circulation regions around the heated cylinder. The results further showed that mean thermal transport rises under conditions where buoyancy forces govern the flow, especially when the imposed magnetic field is aligned normal to the direction of gravity. Subsequently, [24] explored a similar numerical approach using the finite element method to examine MHD mixed convection in a non-Newtonian power-law fluid within a ventilated square chamber containing a heated, rotating internal cylinder.

While prior literature predominantly examines fluid flow or thermal transport within horizontal/vertical conduits or enclosed cavities, the dynamic behaviour of fluid alters significantly within a vertical rectangular configuration featuring asymmetric inlet and outlet ports. Furthermore, with the specific arrangement of buoyancy forces and primary forced convection interactions in a highly intricate manner can create an extra competitive atmosphere in the cavity. Additionally, introducing a rotating cylinder directly into the flow trajectory while simultaneously applying an external magnetic field establishes a complex, opposing physical environment. The magnetic field induces a braking effect via the Lorentz force, whereas the cylindrical rotation promotes mixing and fluid disturbance. Because this precise physical interaction and its practical engineering implications have not yet been systematically analyzed, this investigation was initiated to address this specific literary omission.

1.1 Scope of the Research

Although numerous investigations have evaluated the behavior of magneto-nanofluids within standard geometries, as far as can be determined from available literature, no existing study addresses the fluid behavior within a vertically rectangular vented passage hosting a centrally located rotating cylinder together with inlet and outlet ports. Therefore, the primary objective of this research is to bridge this knowledge gap. Specifically, this study explores how varying rotational speeds can mitigate or alter the influence of the Lorentz force when utilizing a CuO-water nanofluid suspension. By mapping these dynamics, this work evaluates the modifications in fluid velocity fields and localized thermal transport rates across diverse operational parameters, ultimately providing critical design benchmarks for optimizing high-efficiency thermal management systems.

2. Mathematical Formulation

2.1 Physical System and Geometry

The current investigation focuses on the numerical simulation of a steady, two-dimensional laminar, MHD mixed convective flow within a ventilated rectangular enclosure with a width of $W = 0.6H$ and a height of H . The system utilizes a CuO-water based nanofluid as the working medium. The geometry features a lower-left inlet port and an upper-right outlet port, both having a fixed width of $0.1H$. The incoming fluid enters the enclosure through the inlet with a uniform velocity (u_i) at a consistent cold reference temperature (T_c). To induce additional shear-driven flow, the top wall is moved

horizontally at a same constant uniform velocity ($U = u_i$) while being maintained at the same cold temperature (T_c).

Thermally, the system is actively driven by a uniformly heated bottom surface kept at a higher temperature (T_h), while all other exterior vertical boundaries are treated as adiabatic to prevent heat exchange with the surroundings. At the core of the domain, a circular cylinder with a radius of $R = 0.1H$ is positioned, which rotates at a prescribed angular velocity ω . The thermal and dynamic boundary conditions—incorporating the combined forced flow from the inlet, the shear force from the moving top wall, the thermal buoyancy from the heated base, and the rotation of the central cylinder—lead to a complex mixed convection phenomena inside the domain.

The fluid flow is modeled following the Boussinesq approximation to account for buoyancy-induced density variations, while viscous dissipation and Joule heating effects are neglected. For reference, the thermophysical properties of the nanoparticles are listed in Table 1, and the schematic of the physical model is illustrated in Fig. 1. Additionally, the mesh distribution used for the computational domain is shown in Fig. 2. To ensure computational precision, the domain is discretized into 22,012 triangular finite elements, with specialized local mesh refinement applied near the solid walls and around the cylinder to accurately resolve the boundary-layer characteristics.

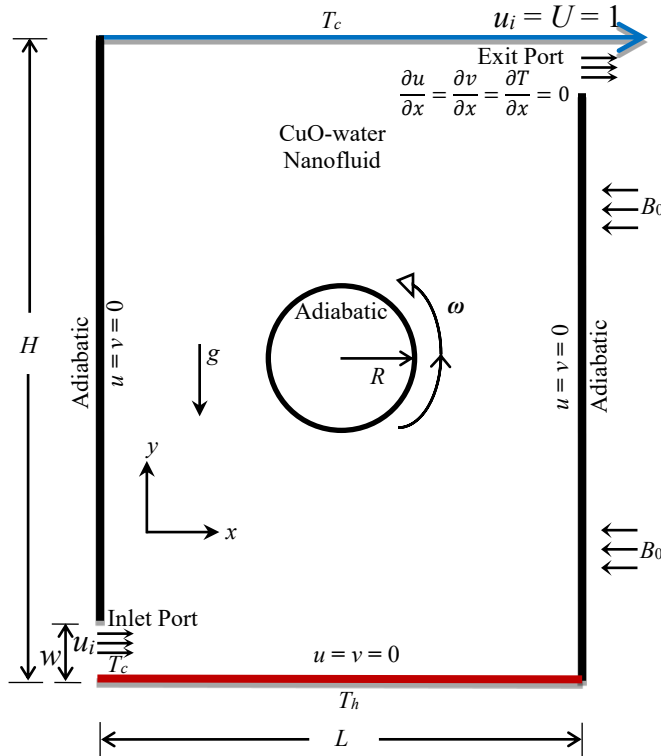


Fig. 1: Schematic diagram and the coordinate system.

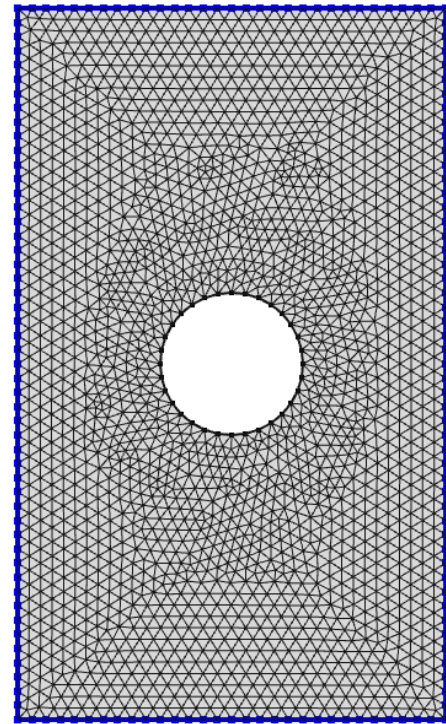


Fig. 2: Mesh distribution of the computational domain.

Table1: Thermophysical characteristics of the water-based fluid and CuO nanoparticles at reference conditions [21].

Material/Property	$C_p (J/kgK)$	$\rho(kg/m^3)$	$k(W/mK)$	$\beta(1/K)$	$\sigma(W/mK)$
Water (Base Fluid)	4179	997.1	0.613	2.1×10^{-4}	0.05

2.2. Dimensional Governing Equations

The present study investigates a two-dimensional mixed convection flow. To mathematically represent the transport phenomena within the system, we employ the fundamental conservation laws of mass, momentum, and energy. Based on the physical assumptions outlined in the previous section, the governing equations in their dimensional form are derived as follows [25], [26]:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \quad (1)$$

$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = -\frac{1}{\rho_{nf}} \frac{\partial p}{\partial x} + \frac{\mu_{nf}}{\rho_{nf}} \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right) \quad (2)$$

$$u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} = -\frac{1}{\rho_{nf}} \frac{\partial p}{\partial y} + \frac{\mu_{nf}}{\rho_{nf}} \left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right) + \frac{1}{\rho_{nf}} [g \beta_{nf} \rho_{nf} (T - T_c)] - \frac{\sigma_{nf}}{\rho_{nf}} B_0^2 v \quad (3)$$

$$u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} = \frac{k_{nf}}{(\rho C_p)_{nf}} \left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} \right) \quad (4)$$

2.3. Nanofluid Properties and Boundary Constraints

The transport phenomena of the nanofluid are governed by its specific thermo-physical properties. In this study, the nanofluid characteristics are determined based on the validated models and data sourced from [27]–[30]:

$$\rho_{nf} = (1 - \phi) \rho_f + \phi \rho_{np}$$

$$\beta_{nf} = (1 - \phi) \beta_f + \phi \beta_{np}$$

$$(\rho C_p)_{nf} = (1 - \phi) (\rho C_p)_f + \phi (\rho C_p)_{np}$$

$$\sigma_{nf} = \sigma_f (1 + ((3((\sigma_{np}/\sigma_f) - 1)\phi) / (((\sigma_{np}/\sigma_f) + 2) - ((\sigma_{np}/\sigma_f) - 1)\phi)) \quad (5)$$

$$\alpha_{nf} = k_{nf} / (\rho C_p)_{nf} \quad \mu_{nf} = \mu_f (1 - \phi)^{-2.5}$$

To specifically address the heat conduction within the fluid, the thermal conductivity is calculated using the modified Maxwell's model ([31]).

$$k_{nf} = k_f ((k_{np} + 2k_f - 2\phi(k_f - k_{np})) / (k_{np} + 2k_f + \phi(k_f - k_{np})))$$

To complete the mathematical model, the following dimensional boundary conditions are applied across the flow domain:

In inlet port: $u = u_i$, $v = 0$, and $T = T_c$

In outlet port: $\frac{\partial u}{\partial x} = \frac{\partial v}{\partial y} = \frac{\partial T}{\partial x} = 0$ and $p = 0$

At the bottom surface: $u = v = 0$, and $T = T_h$ (6)

At the upper boundary: $u = 1$, $v = 0$ and $T = T_c$

On the left and right adiabatic wall: $u = v = 0$, $\frac{\partial T}{\partial x} = 0$

On the rotating cylinder's adiabatic surface: $u = -\omega(y - y_c)$, $v = \omega(x - x_c)$, $\frac{\partial T}{\partial n} = 0$,

where the coordinates of the cylinder's center, located at the cavity center, are (x_c, y_c) .

2.4. Dimensionless Transformations, Reduced Governing Equations and B.Cs

To generalize the mathematical model and simplify the numerical computations, we introduce a set of non-dimensional variables. By scaling the physical coordinates, velocity components and thermal parameters, the governing equations (1)–(6) are transformed into a dimensionless form. The transformation is executed using the following set of variables:

$$X = x/H, Y = y/H, U = u/u_i, V = v/u_i, P = p/(\rho_{nf} u_i^2) \text{ and } \theta = (T - T_c)/(T_h - T_c) \quad (7)$$

Upon substituting these variables into the original dimensional equations, we obtain the dimensionless conservation equations for mass, momentum, and energy:

$$\frac{\partial U}{\partial X} + \frac{\partial V}{\partial Y} = 0 \quad (8)$$

$$U \frac{\partial U}{\partial X} + V \frac{\partial U}{\partial Y} = -\frac{\partial P}{\partial X} + \frac{\nu_{nf}}{\nu_f} \frac{1}{\text{Re}} \left(\frac{\partial^2 U}{\partial X^2} + \frac{\partial^2 U}{\partial Y^2} \right) \quad (9)$$

$$U \frac{\partial V}{\partial X} + V \frac{\partial V}{\partial Y} = -\frac{\partial P}{\partial Y} + \frac{\nu_{nf}}{\nu_f} \frac{1}{\text{Re}} \left(\frac{\partial^2 V}{\partial X^2} + \frac{\partial^2 V}{\partial Y^2} \right) + \frac{(\rho\beta)_{nf}}{(\rho_f \beta_f)} \frac{\rho_f}{\rho_{nf}} \frac{Gr}{\text{Re}^2} \theta - \frac{\rho_f}{\rho_{nf}} \left(\frac{\sigma_{nf}}{\sigma_f} \right) \frac{Ha^2}{\text{Re}} V \quad (10)$$

$$U \frac{\partial \theta}{\partial X} + V \frac{\partial \theta}{\partial Y} = \frac{\alpha_{nf}}{\alpha_f} \frac{1}{\text{RePr}} \left(\frac{\partial^2 \theta}{\partial X^2} + \frac{\partial^2 \theta}{\partial Y^2} \right) \quad (11)$$

$$\text{where } \text{Re} = \frac{u_i L}{\nu_f}, \quad Gr = \frac{g \beta_f (T_h - T_c) L^3}{\nu_f^2}, \quad Ha = B_0 L \sqrt{\frac{\sigma_f}{\mu_f}} \text{ and } \text{Pr} = \frac{\nu_f}{\alpha_f}$$

Applying the transformation variables to the initial boundary constraints (Eq. 6), the dimensionless boundary conditions are written as follows:

At inlet port: $U = 1$, $V = 0$, and $\theta = 0$

At outlet port: $\frac{\partial U}{\partial X} = \frac{\partial V}{\partial X} = \frac{\partial \theta}{\partial X} = 0$ and $P = 0$

Along the bottom heated surface: $U = V = 0$ and $\theta = 1$ (12)

Along the upper boundary: $U = 1$, $V = 0$ and $\theta = 0$

On the rotating cylinder's surface: $U = -\omega(Y - Y_c)$, $V = \omega(X - X_c)$, $\frac{\partial \theta}{\partial n} = 0$

Here (X_c, Y_c) represent the dimensionless coordinates of the cylinder center, while ω denotes the dimensionless angular rotational velocity.

2.4.1. Heat Transfer Analysis (Nusselt Number)

The local Nusselt number along the heated bottom wall of the cavity is determined from the temperature gradient at the surface as:

$$Nu_X = -\frac{k_{nf}}{k_f} \left(\frac{\partial \theta}{\partial Y} \right)_{Y=0}$$

The mean Nusselt number is subsequently determined by integrating the local Nusselt number distribution along the extent of the heated surface:

$$Nu_{av} = \frac{1}{0.6H} \int_0^{0.6H} Nu_X dx.$$

3. Numerical Methodology

To obtain the solutions for the non-linear dimensionless governing equations (8–11) under the specified boundary constraints (12), a robust numerical scheme based on the Galerkin Finite Element Method (GFEM) is implemented. The computational procedure involves the following steps:

- **Discretization:** The entire flow domain is partitioned into smaller, non-overlapping elements. Within each element, a weighted residual approach is utilized to transform the continuous partial differential equations into a discrete system ([9], [32]).
- **Linearization:** Through this framework, the complex governing equations are reformulated into a structured set of linear algebraic expressions
- **Iterative Solution:** The resulting global matrix is assembled and solved using an iterative solver. To ensure the reliability of the results, the process continues until the solution stabilizes.
- **Convergence Criteria:** Numerical convergence is strictly monitored for the dependent variables, including velocity components (U , V) and temperature (θ). The iteration is considered successful only when the maximum absolute residual for each variable falls below a stringent tolerance limit of 10^{-5} .

This computational strategy aligns with the standard Galerkin finite element protocols extensively validated in previous studies [9], [28], [33] – [35].

3.1. Grid Independence Test

The present analysis includes a grid independence test to ensure numerical accuracy and optimize computational costs. The computational domain is sub divided employing a non-uniform triangular grid system. For the reference case, the changes in the domain-averaged Nusselt number and outlet temperature are utilized as measures of sensitivity in relation to the mesh density. Table 2 presents five different mesh systems used in this test, for which the average Nusselt number (Nu_{av}) was evaluated. The results demonstrate that additional grid refinement exerts an insignificant influence on the evaluated parameters. Based on these observations, a mesh consisting of 11306 nodes and 22012 elements is selected as an optimal choice to ensure accurate numerical solutions with acceptable computational efficiency.

Table 2. Grid independency test at $Re = 100$, $Gr = 10^4$, $\omega = 10$, $Ha = 20$ and $\phi = 1\%$.

Nodes	2766	6103	9328	11306	15591
Element	5373	11838	18167	22012	30550
Nu_{av}	11.36498	12.57475	13.19698	13.5501	13.57367

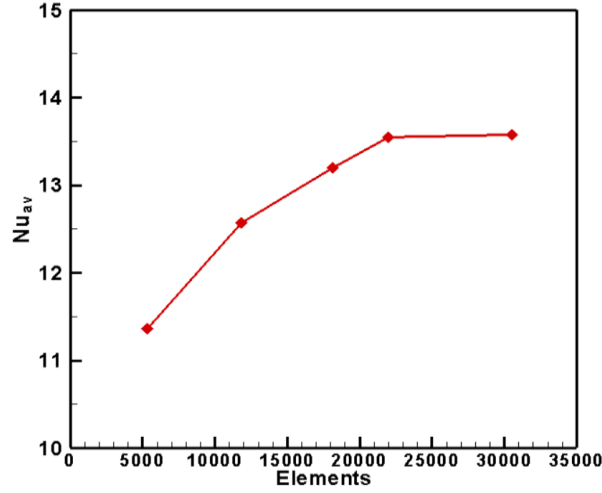


Fig. 3: Grid refinement test.

3.2. Validation of Computational Model

To ensure the reliability and precision of the developed numerical code, a comprehensive validation study was conducted. The current investigation, which focuses on mixed convection of a CuO-water nanofluid within a vented enclosure containing a rotating cylinder, was benchmarked against several established studies. The accuracy of our Finite Element Method (FEM) approach was verified by comparing the results with the work of [36], which employed a finite volume technique for a lid-driven cavity. Furthermore, we reproduced the results from earlier landmark studies, including those by [7], [10], [37]–[39].

3.2.1. Quantitative Agreement

A quantitative comparison of the average Nusselt number (Nu_{av}) at $Pr = 6.2$ and $Gr = 100$ for varying Reynolds numbers is presented in Table 3. The results demonstrate that the current numerical predictions are in excellent agreement with the values reported by [37] and other researchers [7], [10] and [38]. The calculated percentage error remains remarkably low (below 0.15%), confirming the robustness of the present algorithm.

Table 3: Comparative analysis of average Nusselt number (Nu_{av}) against existing literature at $Pr = 6.2$ and $Gr = 100$.

Re		100	400	1000
Nu_{av}	Present study	2.01	3.97	6.36
	Hussain et al. [37]	2.01	3.98	6.37
	Malleswaran and Sivasankaran [39]	-	4.08	6.48
	Sheremet and Pop [10]	2.05	4.09	6.70
	Zheng et al. [36]	-	4.14	6.61
	Abu-Nada and Chamkha [7]	2.09	4.16	6.55
	Sharif [1]	-	4.05	6.55
	Iwatsu et al. [38]	1.94	3.84	6.33
	Error (%)	0.149	0.100	0.07

3.2.2. Qualitative Agreement

In addition to numerical data, a qualitative assessment was performed by comparing the flow and thermal patterns with the results of [40], which introduced a class of high–

resolution schemes based on streamline and isotherm representations. Fig. 4 display the comparison of streamlines and isotherms between the previous study (Fig. 4(a)) and our current findings (Fig. 4(b)). The visual characteristics of the fluid flow and temperature distribution show a high degree of similarity, further validating the model's ability to capture the complex physics of a rotating cylinder within a thermal enclosure.

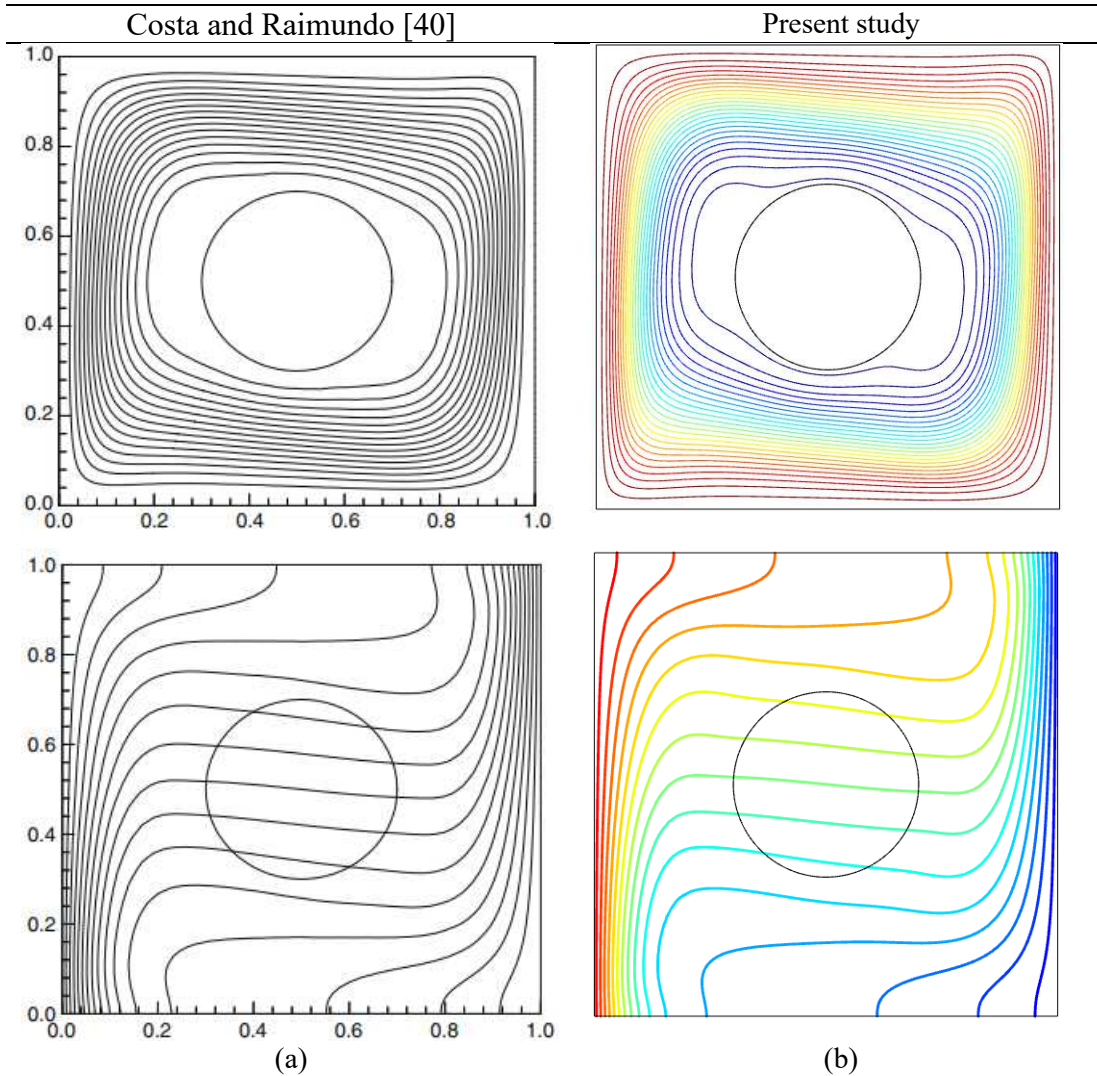


Fig. 4: The current study's (a) streamlines and (b) isotherms chartings are compared to the previous study by [40].

4. Results and Discussion

The investigation examines magnetohydrodynamic (MHD) combined convective heat transfer within a ventilated rectangular cavity containing a rotating adiabatic cylinder by employing the finite element method (FEM). A CuO-water nanofluid is considered to analyze the flow and thermal characteristics under varying physical conditions. The effects of key dimensionless parameters are examined, including the magnetic interaction parameter ($Ha = 0, 20, 50$, and 100), flow Reynolds number ($Re = 10, 100, 250$, and 500), Grashof number ($Gr = 10^4$), nanoparticle volume concentration ($\phi = 0\%, 1\%, 3\%$, and 6%), angular velocity of the inner cylinder ($\omega = 0, 10, 25$, and 50). The Prandtl number is maintained a constant value of $Pr = 6.2$ throughout the analysis.

4.1. Influence of Angular Velocity (ω)

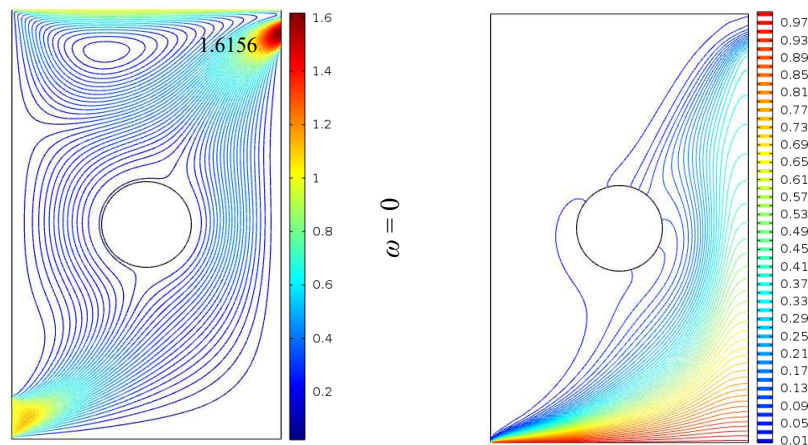
The impact of the cylinder's rotational speed ω , on the flow and thermal characteristics is depicted in Fig. 5. To isolate the effect of rotation, the values of ω are varied from 0 to 50, while other dimensionless parameters are maintained constant at $Pr = 6.2$, $Re = 100$, $Gr = 10^4$, $Ha = 20$, and $\phi = 1\%$

4.1.1. Flow Patterns (Streamlines)

Fig. 5(a) illustrates the evolution of streamlines across different angular velocities (ω). In the absence of rotation $\omega = 0$, the flow field is primarily governed by the competition between buoyancy and Lorentz forces. As the cylinder initiates an anticlockwise rotation, a substantial counterclockwise vortex develops around its surface. Across all simulated cases, a secondary vortex is consistently observed near the exit port. With an increase in ω , the intensity of fluid circulation surrounding the cylinder grows stronger, causing the upper circulating cell to gradually shrink. At a moderate velocity of $\omega = 10$, the streamline configuration begins to shift, and by $\omega = 25$, the reduction of the primary circulating cell becomes more prominent. At the highest angular velocity of $\omega = 50$, the intensified angular speed significantly amplifies the fluid motion around the spinning adiabatic cylinder, dominating the overall flow structure. This enhancement is quantitatively reflected in the central velocity magnitude, which escalates from 1.6156 at $\omega = 0$ to 9.9388 at $\omega = 50$.

4.1.2. Thermal Distribution (Isotherms)

The temperature contours for the corresponding angular velocities are presented in Fig. 5(b). As the rotational speed increases from $\omega = 10$ to 50, the isotherms surrounding the cylinder undergo more pronounced distortion, signaling a progressive enhancement in heat transfer efficiency. While the initial shift is clearly observed at $\omega = 10$, the impact becomes even more significant at higher angular velocities. This rotational effect induces a compression of the isotherms near the heated lower boundary, which in turn elevates the thermal gradients and the overall heat transfer rate. Consequently, as the anticlockwise rotation intensifies toward its maximum value, the temperature field becomes increasingly uniform throughout the enclosure due to more vigorous fluid mixing



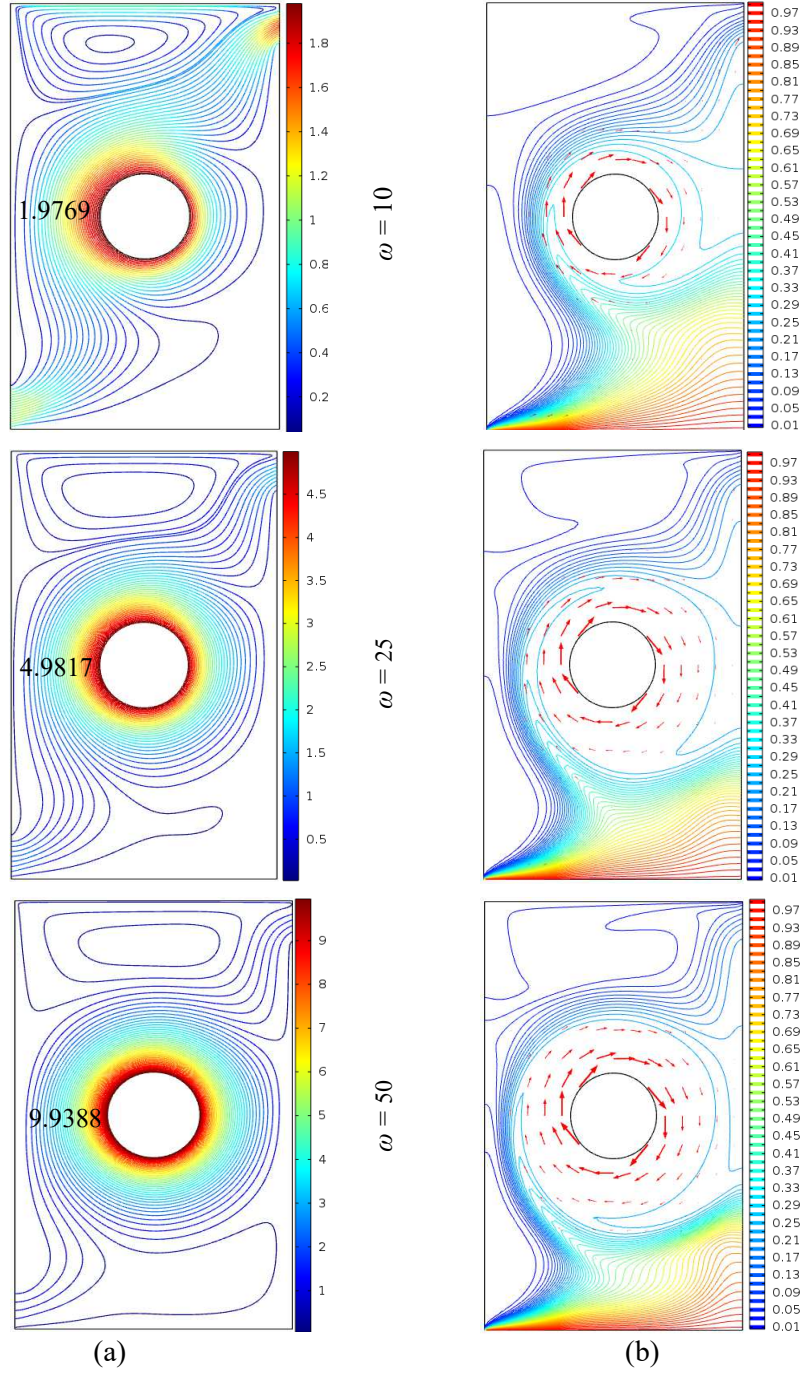


Fig. 5: (a) Streamlines and (b) isotherms for different ω at $Pr = 6.2$, $Re = 100$, $Gr = 10^4$, $Ha = 20$, $\phi = 1\%$.

4.2. Influence of Hartmann Number (Ha)

The influence of the magnetic field, characterized by the Hartmann number (Ha), on the transport phenomena is investigated by varying Ha from 0 to 100. During this analysis, other physical parameters are kept constant at $Pr = 6.2$, $Re = 100$, $Gr = 10^4$, $\omega = 10$, and $\phi = 1\%$. The resulting flow patterns and thermal distributions are illustrated in Fig. 6 through streamline and isotherm contours, respectively.

4.2.1. Flow Patterns (Streamlines)

The streamlines in Fig. 6(a) provide a clear visualization of the Lorentz force's suppressive effect. In the absence of a magnetic field ($Ha = 0$), the convective transport is dominant, leading to the formation of vigorous and expansive circulation cells on both

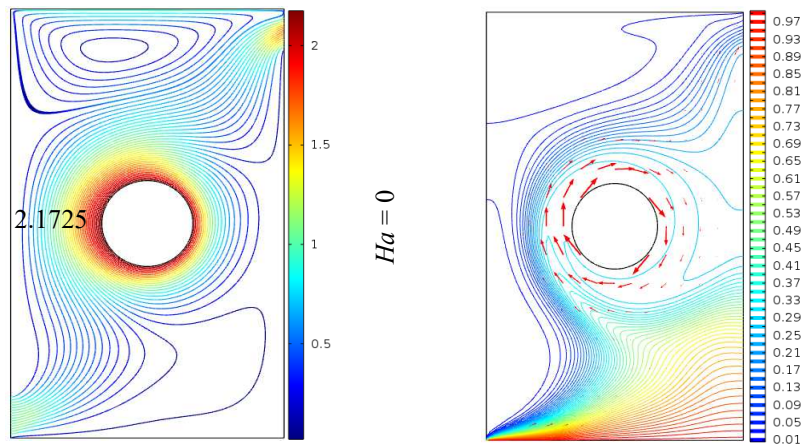
sides of the rotating cylinder. At this stage, the peak velocity magnitude is recorded at its maximum value of 2.1725. As the Hartmann number increases ($Ha = 20$ to 50), a resistive Lorentz force is generated, which acts in direct opposition to the fluid motion. This electromagnetic damping causes the circulation zones to contract and shift closer to the cylinder's surface. Consequently, the flow intensity gradually diminishes, with the velocity magnitude dropping to 1.9769 and 1.6832, respectively. At the strongest magnetic field intensity ($Ha = 100$), the secondary vortices are almost entirely suppressed, and the velocity magnitude further declines to 1.6174. This trend confirms that the magnetic field effectively "stiffens" the fluid, significantly inhibiting the convective flow.

4.2.2. Thermal Distributions (Isotherms)

The isotherms in Fig. 6(b) reflect the transition from convection-dominated to conduction-dominated heat transfer. At $Ha = 0$, the high fluid velocity causes significant distortion in the isothermal lines, particularly near the rotating cylinder and the heated lower wall, where a distinct thermal boundary layer is observed. This high degree of distortion indicates enhanced mixing and a higher heat transfer rate. As Ha increases to 20 and 50, the isotherms begin to lose their complexity and appear more symmetric and smoother. This transition occurs because the Lorentz force stifles the fluid motion, thereby reducing the convective energy transport. At the highest Hartmann number ($Ha = 100$), the isotherms become nearly parallel and vertically stratified around the cylinder. This behavior suggests that heat transfer is now primarily governed by conduction, resulting in a marked decline in the overall cooling efficiency of the system. In summary, increasing the Hartmann number stabilizes the thermal field while simultaneously reducing heat transfer performance.

4.2.3. Physical Highlight on System Novelty

The interaction between the cylinder's rotation and the magnetic force reveals some interesting physics. When the Hartmann number is low, the rotation of the cylinder dominates the flow, creating strong mixing near the heated walls and boosting the heat transfer. But when we increase the Hartmann number to 100, the Lorentz force acts like a strong brake, slowing down the fluid moves and reducing the vortex strength. This shows that the rotation can actually be used as a manual tool to control the negative impacts of a magnetic field, which is a key finding of our setup compared to fixed block designs.



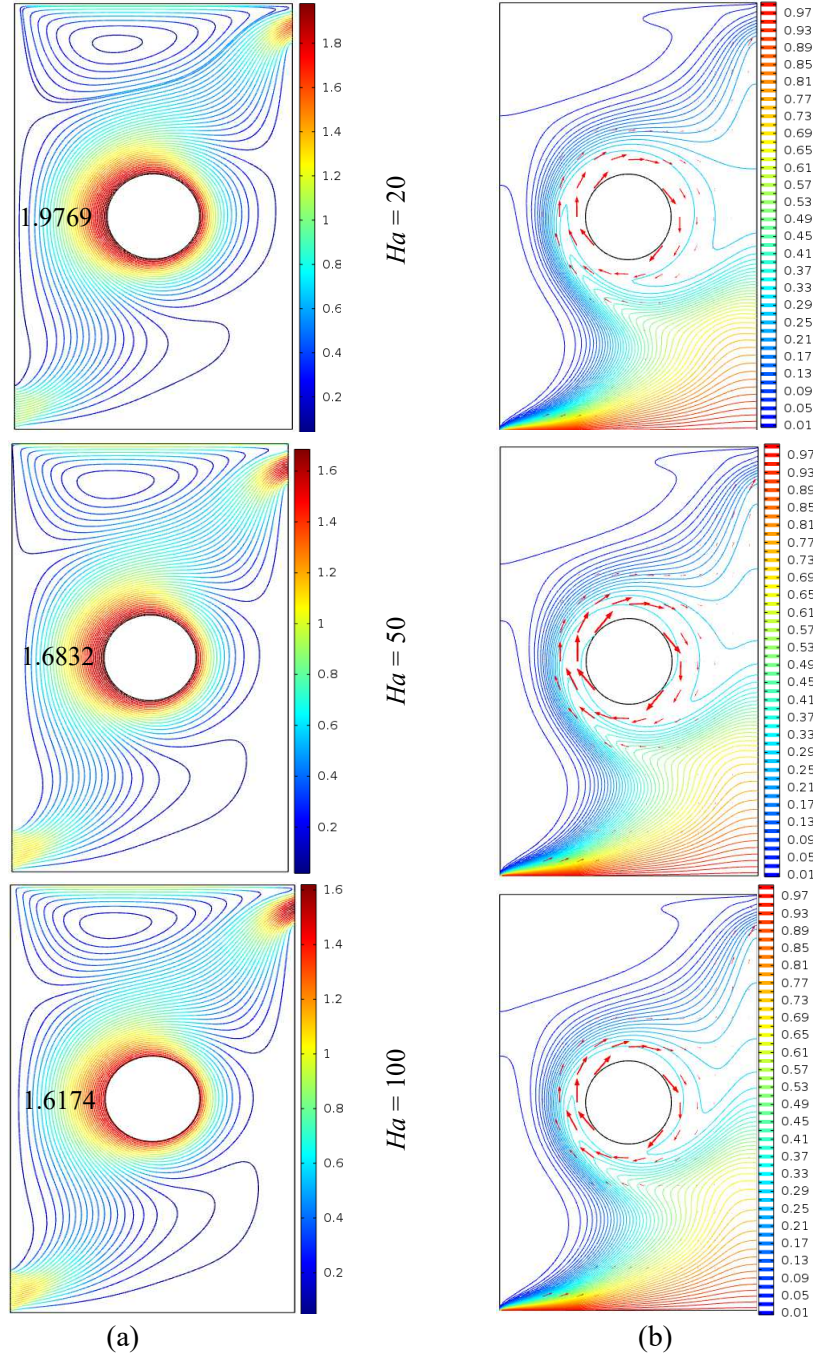


Fig. 6:(a) Streamlines and (b) isotherms for different Ha at $Pr = 6.2$, $Re = 100$, $Gr = 10^4$, $\omega = 10$, $\phi = 1\%$.

4.3. Influence of Reynolds Number (Re)

The transition in flow structures and thermal characteristics under the influence of varying Reynolds numbers ($10 \leq Re \leq 500$) is depicted in Figs. 7(a) and 7(b). The simulation is performed at constant parameters: $Pr = 6.2$, $Ha = 20$, $Gr = 10^4$, $\omega = 10$ and $\phi = 1\%$.

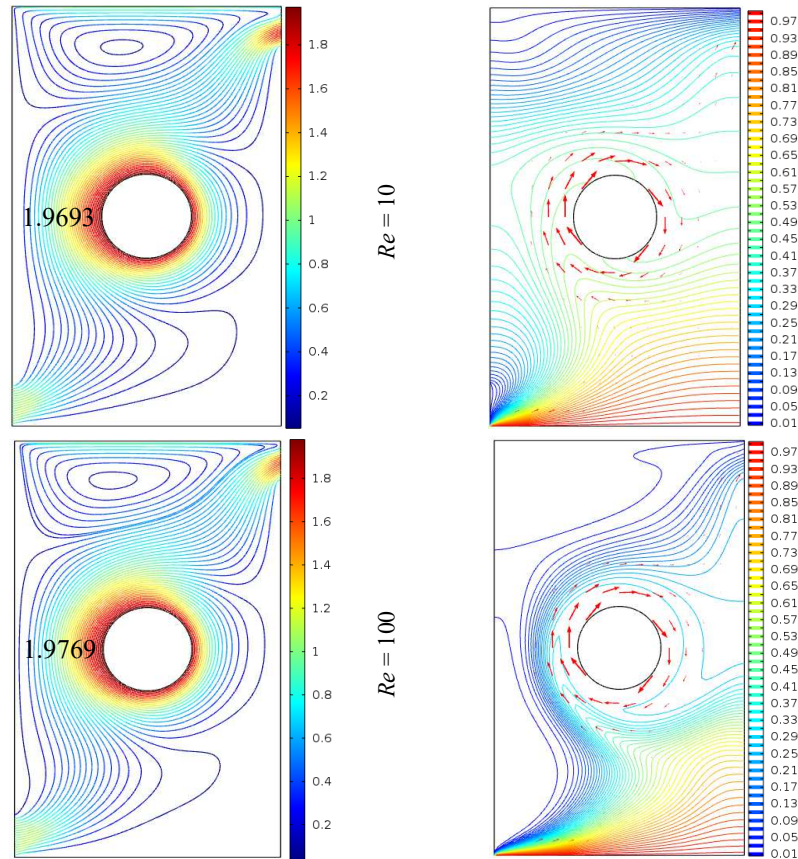
4.3.1. Flow Patterns (Streamlines)

As shown in the left column of Fig. 7(a), at a low Reynolds number ($Re = 10$), the inertia forces are weak, causing the nanofluid to follow a steady, laminar path directly from the inlet to the outlet. In this regime, the rotating cylinder has a negligible impact on the overall flow trajectory. However, as Re escalates to 100, the interaction between the

incoming flow and the rotating body intensifies. A significant change is observed at $Re = 250$, where a small recirculating eddy emerges above the inlet area. This phenomenon indicates the onset of flow separation due to higher momentum. At the maximum value of $Re = 500$, the streamlines become highly distorted with substantial circulation zones appearing downstream. The increased shear forces at the cylinder's surface at higher velocities lead to a more developed and thicker hydrodynamic boundary layer, fundamentally altering the flow distribution compared to lower Re cases. The flow intensity values exhibit a progressive rise from 1.9693 at $Re = 10$ to 2.0469 at $Re = 500$.

4.3.2. Thermal Distributions (Isotherms)

The right column of Fig. 7(b) demonstrates how the temperature field responds to increasing inertial effects. At $Re = 10$, the isotherms are nearly concentric and spread throughout the domain, signifying a conduction-dominated heat transfer mode. As Re increases to 100 and 250, the thermal contours begin to stretch vertically, reflecting the growing dominance of convective transport. The thinning of the thermal boundary layer is particularly evident at $Re = 500$. This reduction in thickness enhances the temperature gradient at the heated bottom wall, which directly corresponds to an augmented heat transfer rate. The loss of thermal symmetry at higher Re further confirms that convection has become the primary mode of energy transport. The numerical results support these visual observations. As evidenced by the bar diagrams, the peak fluid velocity and maximum heat transfer efficiency are achieved at the highest Reynolds number, which effectively leads to a reduction in the average fluid temperature within the channel.



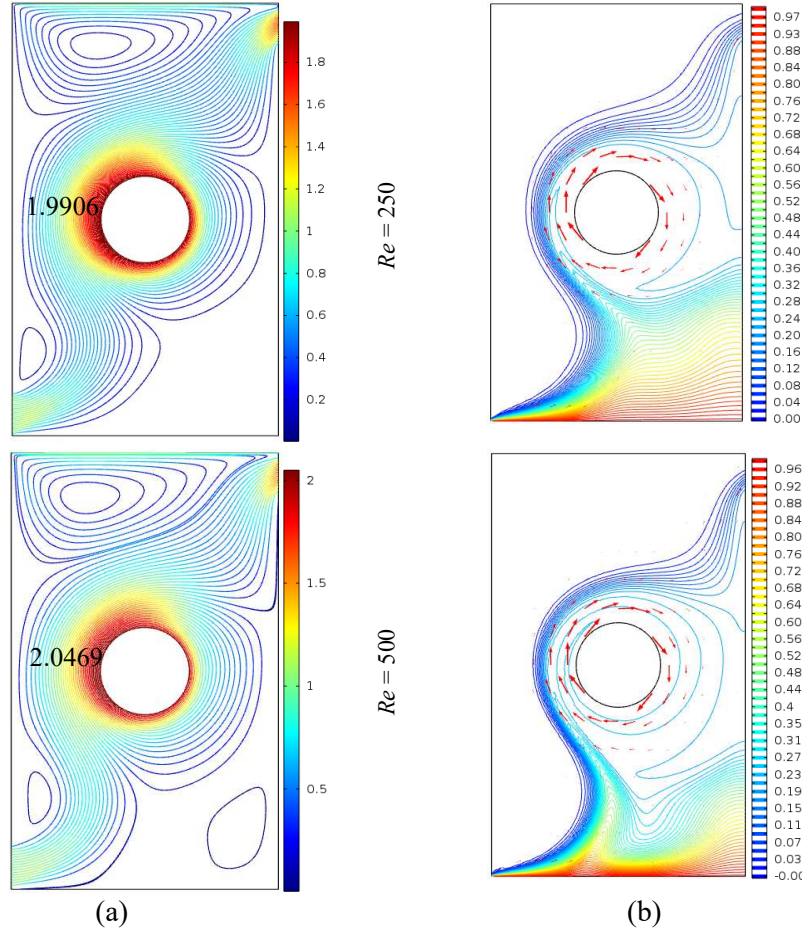


Fig. 7: (a) Streamlines (b) and isotherms for different Re at $Pr = 6.2$, $\omega = 10$, $Gr = 10^4$, $Ha = 20$, $\phi = 1\%$.

4.4. Influence of Volume Fraction (ϕ)

The impact of nanoparticle concentration on the mixed convection flow and thermal characteristics is presented in Figs. 8(a) and 8(b). This analysis is conducted by varying the volume fraction (ϕ) from 0% to 6%, while keeping other parameters fixed ($Pr = 6.2$, $Re = 100$, $Gr = 10^4$, $Ha = 20$, $\omega = 10$).

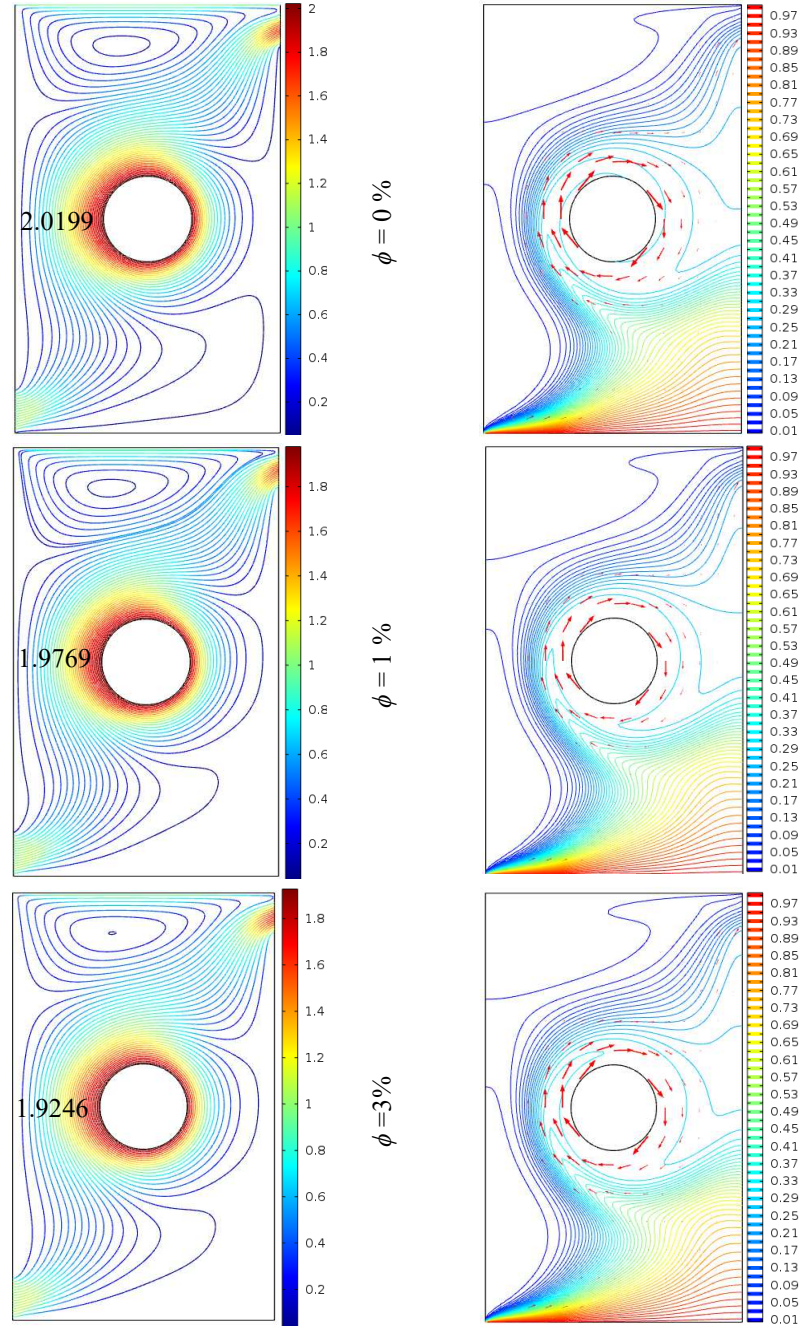
4.4.1 Flow Patterns (Streamlines)

Fig. 8(a) illustrates the evolution of streamlines as CuO nanoparticles are added to the base fluid. In the case of pure water ($\phi = 0\%$) a prominent vortex structure is observed downstream of the rotating cylinder, driven by the interaction between forced convection and rotational effects. As the solid volume fraction increases to 1%, 3%, and 6%, the fundamental flow pattern remains largely consistent, suggesting that the presence of nanoparticles does not drastically alter the streamline topology. However, a noticeable deceleration in momentum is observed. Quantitatively, the flow intensity shows a steady decline as the fluid becomes more concentrated. As indicated by the numerical results in Fig. 8(a), the velocity intensity drops from 2.0199 ($\phi = 0\%$) to 1.8737 ($\phi = 6\%$). This reduction is attributed to the enhanced viscosity of the nanofluid at higher concentrations, which resists fluid motion

4.4.2 Thermal Distributions (Isotherms)

The effect of nanoparticle loading on the thermal field is depicted in Fig. 8(b). For lower volume fractions ($\phi = 0\%$ and 1%) the isotherms are densely packed near the surface of the adiabatic cylinder. This density indicates steep thermal gradients, which are essential

for robust convective heat transfer. As the concentration rises to 3% and 6%, the distribution of isotherms suggests an enhancement in the effective thermal conductivity of the mixture. This leads to the development of a more refined thermal boundary layer. Furthermore, much of the cavity maintains a nearly uniform temperature distribution. This implies that the forced flow effectively transports heat from the isothermal wall directly toward the outlet, preventing the thermal energy from diffusing extensively throughout the entire enclosure.



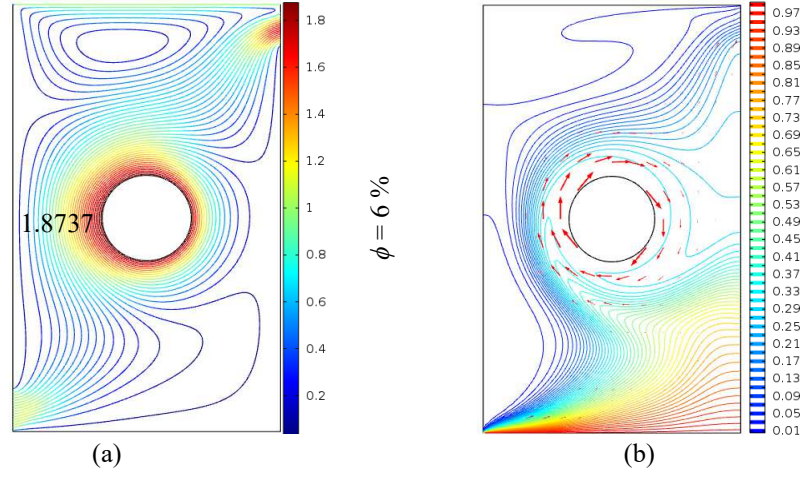


Fig.8: (a) Streamlines and (b) isotherms for different ϕ at $Pr = 6.2$, $Re = 100$, $Gr = 10^4$, $Ha = 20$, $\omega = 10$.

4.5. Influence of Ha , ϕ , and ω on Nu_{av} and θ_{av}

Figs. 9(a) and 9(b) illustrate the distribution of the average Nusselt number (Nu_{av}) alongside the mean temperature (θ_{av}) across a range of Reynolds numbers (Re) for different Hartmann numbers (Ha). Throughout this analysis, parameters such as $Gr = 10^4$, $\phi = 1\%$, and $Pr = 6.2$ are maintained at fixed values. The data reveals a consistent growth in Nu_{av} as the inertial force intensifies. Such a pattern implies that more vigorous fluid displacement facilitates better convective heat exchange, which subsequently leads to improved thermal transport at elevated levels.

On the other hand, elevating the magnetic strength results in a reduction of the Nu_{av} . This happens because the imposed magnetic field induces a Lorentz force, which acts as a resistive mechanism against the flow and stifles the intensity of convective cells. Consequently, thermal efficiency declines as Ha becomes more prominent. The peak heat transfer efficiency is identified in the absence of a magnetic field ($Ha = 0$). Furthermore, Fig. 9(b) demonstrates that the enclosure's mean temperature undergoes a marginal rise when Ha is increased owing to the suppression of fluid circulation while it shows a downward trend with higher Re .

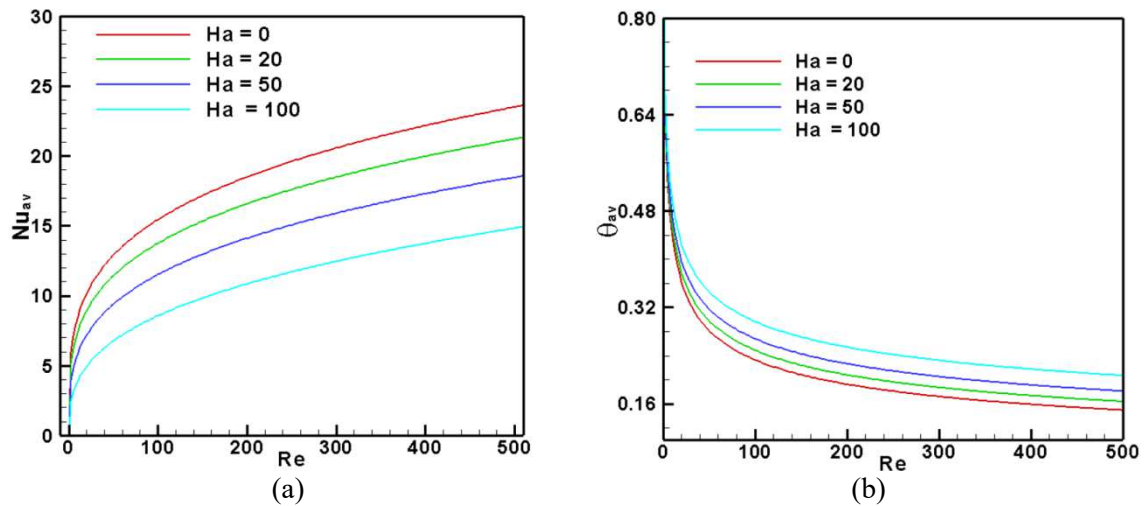


Fig. 9: Profiles of (a) Nu_{av} and (b) θ_{av} versus Re for different Ha at $Pr = 6.2$, $Gr = 10^4$, $\omega = 10$, $\phi = 1\%$.

The impact of nanoparticle concentration on the system's thermal behavior is depicted in Figs. 10(a) and 10(b). Observation of Fig. 10(a) shows that Nu_{av} improves consistently as the volume fraction (ϕ) grows, with the most notable gains occurring at elevated Reynolds

numbers ($Re = 500$). This thermal boost is primarily due to the nanofluid's enhanced effective thermal conductivity relative to the pure base fluid, which effectively intensifies the convective heat exchange process. On the other hand, Fig. 10(b) demonstrates a reduction in θ_{av} as the concentration of nanoparticles increases. This downward trend is associated with superior thermal diffusion within the medium, facilitating more rapid and efficient heat dissipation from the enclosure.

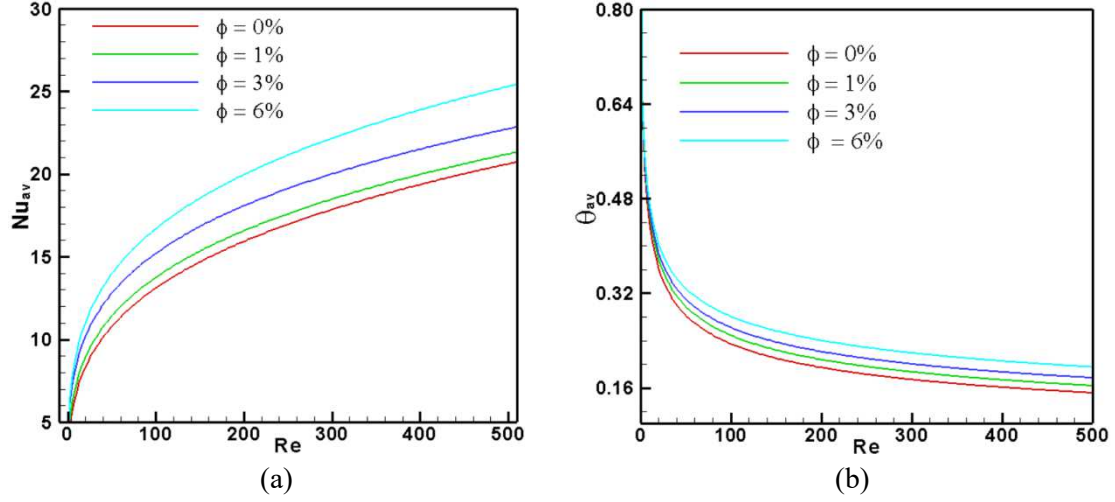


Fig.10: Profiles of (a) Nu_{av} and (b) θ_{av} versus Re at different concentration levels ϕ for $Pr = 6.2$, $Gr = 10^4$, $Ha = 20$, $\omega = 10$.

The impact of rotational velocity on the system's thermal characteristics, specifically the average Nusselt number (Nu_{av}) and mean fluid temperature (θ_{av}), are illustrated in Figs. 11(a) and 11(b) for a spectrum of Reynolds numbers (Re). The results clearly demonstrate that Nu_{av} follows an upward trajectory as Re grows, with the most substantial improvements occurring at higher rotation rates. Additionally, Fig. 11(b) reveals a downward trend in the enclosure's mean temperature as the inertial forces (Re) intensify. Specifically, for ω values of 0, 10, 25, and 50, a sharp decline in θ_{av} is observed when Re spans from 10 to 100. Conversely, this reduction stabilizes and becomes notably more balanced as Re progresses through the higher interval of 100 to 500.

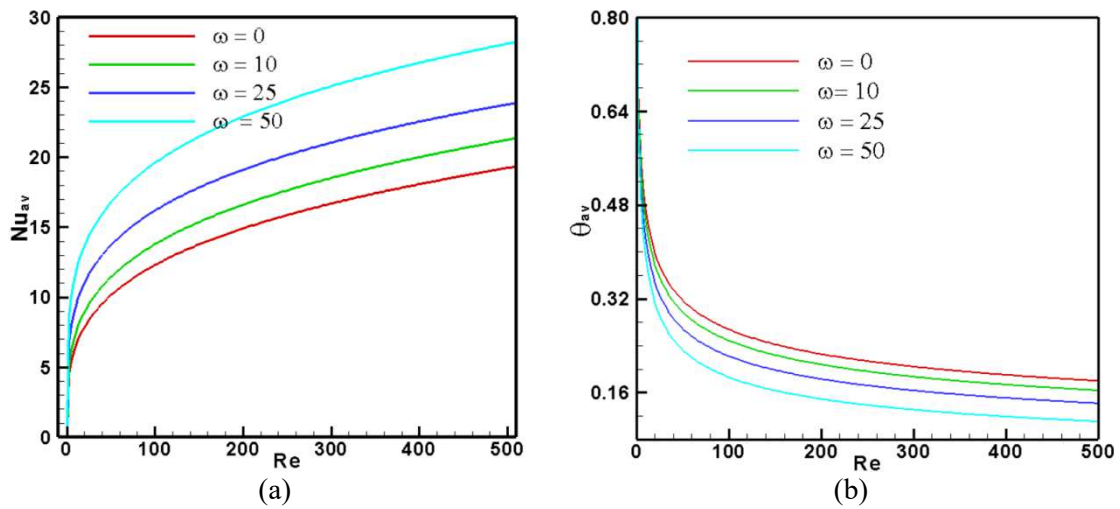


Fig. 11: : Profiles of (a) Nu_{av} and (b) θ_{av} versus Re at different ω for $Pr = 6.2$, $Gr = 10^4$, $Ha = 20$, $\phi = 1\%$.

5. Conclusions

This research numerically investigates the magnetohydrodynamic mixed convection of a nanofluid inside a vented cavity that includes a rotating adiabatic cylinder. In this analysis, the Prandtl number was kept constant at $Pr = 6.2$, and the Grashof number was fixed at $Gr = 10^4$. Meanwhile, the influences of the Hartmann number, Reynolds number, angular velocity, and nanoparticle volume fraction were carefully investigated. To solve the governing equations, the Galerkin finite element method was adopted.

To make sure that our model worked correctly, we checked our results against previously published data and found a very good match.

From the obtained computational results, the principal outcomes may be summarized as follows:

- The results demonstrate that an increase in the rotational speed of the cylinder significantly enhances fluid circulation within the domain.
- The streamlines patterns change clearly, with more curvature and stronger flow structures forming at higher speeds.
- The temperature distribution (isotherms) is also affected, showing better heat transfer as the rotation increases.
- For clockwise rotation of the cylinder ($\omega = 10$), the local Nusselt number rises noticeably with increasing Reynolds number. When Re increases from 10 to 500, the heat transfer rate improves by nearly 195.74%.
- Increasing the magnetic field strength tends to hinder fluid circulation, which in turn diminishes the convective heat transfer efficiency. Our results indicate that as the Hartmann number rises, there is a consistent decline in heat transfer performance. For instance, at a Reynolds number of 100, we observed that imposing a magnetic field of $Ha = 100$ leads to a nearly 47.90% drop in heat transfer compared to the scenario where no magnetic field is applied.
- By shifting thermal energy toward the right side of the base, the magnetic field demonstrates its effectiveness as a regulatory tool. This highlights how magnetic influence can be strategically used to manage localized heat distribution in such systems.
- Anticlockwise rotation of the cylinder enhances heat transfer near both ends of the bottom wall. In comparison with the stationary configuration, an improvement of about 13.09% is obtained at $Re = 100$.
- As the nanoparticle volume fraction rises, the nanofluid becomes more thermally conductive, leading to improved heat transfer rates. This shift demonstrates that adding more nanoparticles directly strengthens the overall thermal performance of the system.
- A solid volume fraction of $\phi = 6\%$ yields approximately a 5.60% enhancement in heat transfer relative to the base fluid with ($\phi = 0\%$).

Overall, the findings of this research demonstrate that using a rotating element is a very practical way to overcome the drop in heat transfer (which can go up to 47.90%) caused due to the Lorentz force effect. These results offer useful design ideas for engineers working on cooling systems for compact electronic devices where both magnetic fields and space limitations are major challenges.

Declarations

Competing Interest

The authors declare that they have no competing interests related to this research, authorship, or publication of this article.

Funding

No funding was received for this research, authorship, or publication of this study.

Availability of Data Material

The datasets generated and/or analyzed during the current study are available from the corresponding author upon reasonable request.

Code Availability

Not applicable

Authors' Contributions

M. M. Touhid Hossain, Md. Abdul Alim and Mohammad Mokaddes Ali conceived and planned the study. Pushpa Akhtar developed the Finite Element Model and computational program. Md. Abdul Alim assisted in running and simulating the model. Pushpa Akhtar performed the numerical simulations and generated the empirical results. M. M. Touhid Hossain contributed to the analysis and interpolation of the numerical findings. Mohammad Mokaddes Ali verified and validated the results. All authors contributed to the preparation and writing of the manuscript. The final version of the manuscript was reviewed, discussed, and approved by all authors prior to submission.

Acknowledgments

This research forms a part of the Ph.D. work of the second author. The authors gratefully acknowledge the Khulna University of Engineering & Technology (KUET), particularly the Department of Mathematics, for providing academic and financial support to conduct this study. The authors also sincerely thank the anonymous reviewers for their valuable comments and constructive suggestions, which have significantly improved the quality of this manuscript.

Nomenclature:

c_p	Specific heat at constant pressure	$(J\ kg^{-1}K^{-1})$	X, Y	Dimensionless coordinates	
g	Gravitational acceleration	$(m\ s^{-2})$	x_c, y_c	Dimensional coordinates of the centre of cylinder	(m)
L	Length of the cavity	(m)	X_c, Y_c	Dimensionless coordinates of the centre of cylinder	
H	Height of the cavity	(m)	u_i	Inlet velocity, top moving wall velocity	(ms^{-1})
R	Radius of circular cylinder	(m)	Greek symbols		
w	Height of the opening	(m)	α	Fluid thermal diffusivity	(m^2s^{-1})
B_0	Magnetic induction		β	Thermal expansion coefficient	(K^{-1})
h	Local heat transfer coefficient	$(Wm^{-2}K^{-1})$	ν	Kinematic viscosity	(m^2s^{-1})
k	Thermal conductivity	$(Wm^{-1}K^{-1})$	σ	Electrical conductivity	(Sm^{-1})
p	Dimensional pressure	(Nm^{-2})	ρ	Density	(kgm^{-3})
P	Dimensionless pressure		θ	Dimensionless temperature	
q_w	Heat flux	(Wm^{-2})	ϕ	Volume fraction of the nanoparticles	
Ri	Richardson number	$Ri = Gr / Re^2$	μ	Dynamic viscosity	(Nsm^{-2})
Gr	Grashof number	$gb(T_h - T_c)H^3 / \nu^2$	ω	Angular rotational speed of the cylinder	Sm^{-1}

Ha	Hartman number	$(\sigma B_0^2 H^2 / \rho \nu)^{1/2}$	Subscripts	
Pr	Prandtl number	ν / α	h	Hot
Re	Reynolds number	$U_0 H / \nu$	c	Cold
Nu	Nusselt number		L	Local
T	Dimensional temperature	(K)	av	Average
T_h	Hot wall temperature		f	Base fluid
T_c	Cold wall temperature	(K)	nf	Nanofluid
u, v	Dimensional velocity components	(ms^{-1})	np	Nanoparticles
U, V	Dimensionless velocity components		i	Inlet
x, y	Dimensional coordinates	(m)		

References:

- [1] Sharif, M. R. (2007). Laminar mixed convection in shallow inclined driven cavities with hot moving lid on top and cooled from bottom. *Appl. Therm. Eng.*, 27(5–6), 1036–1042. doi: 10.1016/j.applthermaleng.2006.07.035
- [2] Waheed, M. (2009). Mixed convective heat transfer in rectangular enclosures driven by a continuously moving horizontal plate, *Int J Heat Mass Transf.*, 52(21–22), 5055–5063. doi: 10.1016/j.ijheatmasstransfer.2009.05.
- [3] Ismael, M. A. and Jasim, H. F. (2018). Role of the fluid-structure interaction in mixed convection in a vented cavity, *International Journal of Mechanical Sciences*, 135, 190–202.
- [4] Koufi, L., Younsi, Z., Cherif, Y. and Naji, H., (2017). Numerical investigation of turbulent mixed convection in an open cavity: effect of inlet and outlet openings, *International Journal of Thermal Sciences*, 116, 103–117.
- [5] Koo, J. and Kleinstreuer, C. (2005). Laminar nanofluid flow in microheat-sinks, *International Journal of Heat and Mass Transfer*, 48(13), 2652–2661.
- [6] Oztop, H. F. and Abu-Nada, E. (2008). Numerical study of natural convection in partially heated rectangular enclosures filled with nanofluids, *International Journal of Heat and Fluid Flow*, 29(5), 1326–1336.
- [7] Abu-Nada, E. and Chamkha, A. J. (2010). Mixed convection flow in a lid-driven inclined square enclosure filled with a nanofluid, *European Journal of Mechanics B/Fluids*, 29(6), 472–482.
- [8] Sourtiji, E., Gorji-Bandpy, M., Ganji, D. D. and Hosseinizadeh, S. F. (2014). Numerical analysis of mixed convection heat transfer of Al_2O_3 -water nanofluid in a ventilated cavity considering different positions of the outlet port, *Powder Technology*, 262, 71–81.
- [9] Nasrin, R., Alim, M. A., and Chamkha, A. J. (2012). Buoyancy-driven heat transfer of water– Al_2O_3 nanofluid in a closed chamber: Effects of solid volume fraction, Prandtl number and aspect ratio, *International Journal of Heat and Mass Transfer*, 55(25–26), 7355–7365.
- [10] Sheremet, M. A., Pop, I. (2015). Mixed convection in a lid-driven square cavity filled by a nanofluid: Buongiorno's mathematical model, *Applied Mathematics and Computation*, 266, 792–808. <https://doi.org/10.1016/j.amc.2015.05.145>
- [11] Bahlaoui, A., Raji, A., Hasnaoui, M., and Naïmi, M. (2016). Mixed Convection Heat Transfer Enhancement in a Vented Cavity Filled with a Nanofluid, *Journal of Applied Fluid Mechanics*, 9(2), 593–604.
- [12] Oztop, H. F., Al-Salem, K. and Pop, I. (2011). MHD mixed convection in a lid-driven cavity with corner heater, *International Journal of Heat and Mass Transfer*, Vol. 54, p. 4943504.
- [13] Rahman, M., Oztop, H. F., Saidur, R., Mekhilef, S. and Al-Salem, K., (2013). Finite element solution of MHD mixed convection in a channel with a fully or partially heated cavity, *Computers and Fluids*, 79, 53–64.
- [14] Pekmen, B., and Sezgin, M. T. (2014). MHD flow and heat transfer in a lid-driven porous enclosure, *Computers and Fluids*, 89, 191–199.

- [15] Sheikholeslami, M., Bandpy, M. G., Ellahi, R., Zeeshan, A. (2014). Simulation of MHD CuO–water nanofluid flow and convective heat transfer considering Lorentz forces, *Journal of Magnetism and Magnetic Materials*, 369, 69–80. <https://doi.org/10.1016/j.jmmm.2014.06.017>
- [16] Hajialigol, N., Fattahi, A., Ahmadi, M.H., Qomi, M.E., and Kakoli, E., (2015). MHD mixed convection and entropy generation in a 3-d microchannel using Al₂O₃-water nanofluid, *Journal of The Taiwan Institute of Chemical Engineers*, 46, 30–42.
- [17] Milani, S. K., Öztop H. F., Al-Salem, K. (2017). Mixed magnetohydrodynamic convection in a Cu–water–nanofluid-ventilated square cavity using the Taguchi method: a numerical investigation and optimization, *Eur. Phys. J. Plus.*, 132, 1–11. <https://doi.org/10.1140/epjp/i2017-11471-3>
- [18] Rahman, Md. M., Elias, Md., and Alim, Md. A. (2011). Mixed convection flow in a rectangular ventilated cavity with a heat conducting solid circular cylinder at the centre, *IJE Transactions B: Applications*, 24(1), 93–105.
- [19] Mehrizi, A. A., Farhadi, M., Afroozi, H. H., Sedighi, K. and Darz, A. A. R. (2012). Mixed convection heat transfer in a ventilated cavity with hot obstacle: effect of nanofluid and outlet port location, *International Communications in Heat and Mass Transfer*, 39(7), 1000–1008.
- [20] Chatterjee, D., & Mishra, R. (2017). Numerical investigation of transient magnetohydrodynamic mixed convection in a ventilated cavity containing two heated circular Cylinders. *Heat Transfer Engineering*, 39(12), 1052–1066. <https://doi.org/10.1080/01457632.2017.1358487>
- [21] Selimefendigil, F., and Chamkha, A. J. (2020). MHD mixed convection of nanofluid in a three-dimensional vented cavity with surface corrugation and inner rotating cylinder, *International Journal of Numerical Methods for Heat & Fluid Flow*, 30(4), 1637–1660. doi: <https://doi.org/10.1108/HFF-10-2018-0566>
- [22] Azim, E., Sagor, Md. J. H., Hasan, A. B. M. R., and Saha, S. (2021). Investigation on conjugate mixed convection inside a square cavity with heat generating rotating circular cylinder, *AIP Conference Proceedings* 2324, 050025, 1–8.
- [23] Akhtar, P., Hossain, M. M. T., Alim, Md. A. and Ali M. M., (2023). Magnetohydrodynamic Mixed Convection in a Vertical Rectangular Cavity with Semi-circular Heat Source and Circular Obstacle, *Int. J. Appl. Comput. Math.* 9:70, 1–25. <https://doi.org/10.1007/s40819-023-01561-zv>
- [24] Chowdhury, Md. M. U., Islam, Md. M., Hossain, Md. Z., & Molla, Md. M. (2024). Magnetohydrodynamics mixed convection and sensitivity analysis of non-Newtonian power-law fluid in a ventilated cavity with a rotating heated cylinder, *Journal of Taibah University for Science*, 18(1). <https://doi.org/10.1080/16583655.2024.2427386>
- [25] Thohura, S., Molla, M. M, Sarker M. A., et al. (2021). Study of mixed convection flow of power-law fluids in a skewed lid driven cavity, *Heat Transf.*, 50(6), 6328–6357. doi: 10.1002/htj.v50.6
- [26] Mahmud, M. Z., Molla, M. M. (2024). Natural convection and heat transmission of two-phase Bingham Fe₃O₄–CoFe₂O₄– water hybrid nanofluid flow in an enclosure with a corrugated heated cylinder, *Phys Scr.*, 99(6)065267. doi:10.1088/1402-4896/ad4c1b
- [27] Selimefendigil, F. and Oztop, H. F. (2014). Numerical study of MHD mixed convection in a nanofluid filled lid driven square enclosure with a rotating cylinder, *International Journal of Heat and Mass Transfer*, 78, 741–754,
- [28] Ali, M. M., Alim, Md. A., and Ahmed, S. S. (2019). Oriented magnetic field effect on mixed convective flow of nanofluid in a grooved channel with internal rotating cylindrical heat source, *International Journal of Mechanical Sciences*, 151, 385–409.
- [29] Mehmood, K., Hussain, S., and Sagheer, M. (2017). Mixed convection in alumina-water nanofluid filled lid-driven square cavity with an isothermally heated square blockage inside with magnetic field effect: introduction,” *International Journal of Heat and Mass Transfer*, 109, 397–409.

- [30] Motlagh, S. Y., and Soltanipour, H. (2017). Natural convection of Al_2O_3 -water nanofluid in an inclined cavity using Buongiorno's two-phase model, *International Journal of Thermal Sciences*, 111, 310–320.
- [31] Cui, W., Shen, Z., Yang, J., and Wu, S. (2014). Modified prediction model for thermal conductivity of spherical nanoparticle suspensions (nanofluids) by introducing static and dynamic mechanisms, *Industrial & Engineering Chemistry Research*, 53(46), 18071–18080.
- [32] Zienkiewicz, O. C., and Taylor, R. L. The Finite Element Method, McGraw-Hill, USA, 1991.
- [33] Akhter, R., Mokaddes, M. A., and Alim, Md. A. (2023). Entropy generation due to hydromagnetic buoyancy-driven hybrid-nanofluid flow in partially heated porous cavity containing heat conductive obstacle, *Alexandria Engineering Journal*, vol. 62, pp. 17–45.
- [34] Taylor, C., and Hood, P. (1973). A numerical solution of the Navier-Stokes equations using the finite element technique, *Computers & Fluids*, 1, 73–100.
- [35] Dechaumphai, P. Finite Element Method in Engineering, Chulalongkorn Uni. Press, Bangkok, 1999.
- [36] Zheng, G. F., Ha, M. Y., Yoon, H. S., Park, Y. G. (2023). A numerical study on mixed convection in a lid-driven cavity with a circular cylinder, *J. Mech. Sci. Technol.*, 27 (1) 273-286.
- [37] Hussain, S., Ahmed, S. E., & Akbar, T. (2017). Entropy generation analysis in MHD mixed convection of hybrid nanofluid in an open cavity with a horizontal channel containing an adiabatic obstacle, *International Journal of Heat and Mass Transfer*, 114, 1054–1066.
- [38] Iwatsu, R., Hyun, J. M. & Kuwahara, K. (1993). Mixed convection in a driven cavity with a stable vertical temperature gradient, *International Journal of Heat and Mass Transfer*, 36(6) (1993) 1601-1608.
- [39] Malleswaran, A., Sivasankaran, S. (2016). A numerical simulation on MHD mixed convection in a lid-driven cavity with corner heaters, *J. Appl. Fluid Mech.* 9311-319.
- [40] Costa, V. A. F., & Raimundo, A. M. (2010). Steady mixed convection in a differentially heated square enclosure with an active rotating circular cylinder. *International Journal of Heat and Mass Transfer*, 53(5-6), 1208-1219.