International Journal of Research Publication and Reviews

Journal homepage: www.ijrpr.com ISSN 2582-7421

Simulation on Effect of Nanofluids Flow in a Square Enclosure with Internal Heated Objects

[1]Govindarao Nammi, [2]Sampath Kumar Peddi, [3]Lalith Kumar Gedi, [4]Mahesh Puli, [5]Komala Rao Pallina, [6]Kishore Padala, [7]Sai Kongarapu

[1,2,3,4,5,6,7] Mechanical Department, GMR Institute of Technology, Rajam, Andhra Pradesh, India

Email: govindarao.n@gmail.com

ABSTRACT

The present study examines simulation on effect of nanofluids the movement of fluid within a square enclosure containing internally heated objects, filled with SiC, Fe2O3, and copper-based water nanofluids, is analyzed. The findings are depicted through streamlines, isotherms, and the Nusselt number for Rayleigh number varies from 103 ≤ Ra ≤ 106, volume fraction varies from 1% ≤ ϕ ≤ 5%, and spacing between them is S=0.2. For circular, square and triangular internal heated objects, the findings indicate that the Nusselt number for SiC nanofluid is notably higher when compared to other nanofluids evaluated alongside various internally heated objects. As the volume fraction of the nanofluid rises, the Nusselt number begins to decline, suggesting that conduction is becoming the primary mode of heat transfer. At a Ra of 10^6, the Nusselt number for SiC is 7.54, Fe2O3 is 7.525, Cu is 7.521 in a circular heated object. Comparing to other internal heated objects the circular heated object performed better due to its uniform shape. These values demonstrate that SiC nanofluids provide the most significant heat transfer enhancement due to its Brownian motion and density.

1. Introduction

1.1 Convection

Convection is a form of heat transfer in fluids; fluid movement and temperature gradients influence both liquids and gases across the fluid. The primary reason for this process is the creation of density differences, which arise because of temperature differences within various parts of the fluid. When temperatures change within various parts of the fluid, warm areas become less dense, and therefore rise upwards, while cold and denser areas sink downwards. This cycle of rising and sinking continues to set up a circulation pattern in which it effectively transfers the heat within the fluid.

The process of convection takes plays a significant role in numerous natural and engineered systems. For instance, it influences climate trends, ocean flows, and the efficacy of heating and cooling systems in engineering. The speed of heat transfer through convection is determined by properties of fluids such as viscosity, thermal conductivity, and specific heat capacity; however, those at the exterior of a fluid can greatly influence The rate of convection can vary due to factors such as flow speed and surface characteristics. According to Newton's law of cooling, the heat transfer rate is represented by the equation $Q = h(T - T\ )$ (1.1). In this equation, Q signifies the heat transfer rate, h stands for the convective heat transfer coefficient, A denotes the surface area, T represents the surface temperature, and T is the ambient temperature.

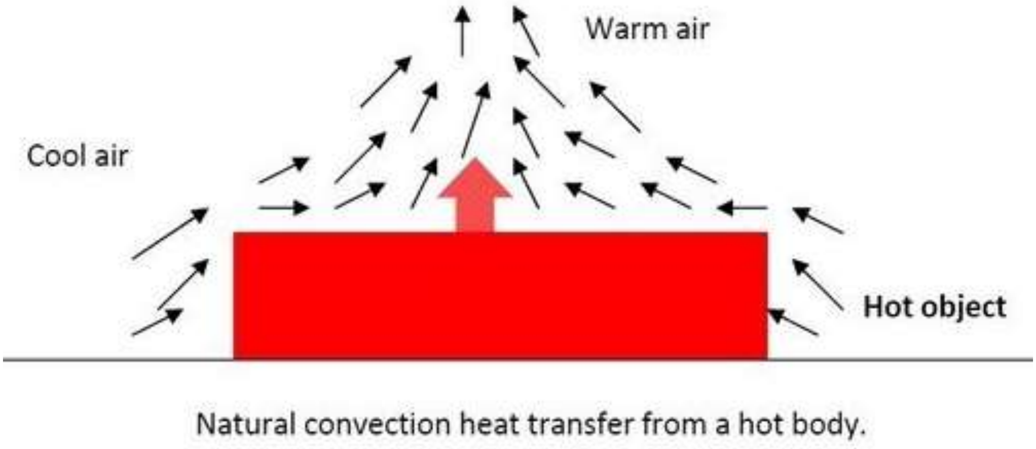


Fig. 1.1: Convection Process [1]

1.1.1 Types of Convection

There are two categories of convection: Natural convection and Forced convection. Natural convection refers to heat transfer that occurs due to fluid motion driven by buoyancy forces resulting from temperature variations within the fluid.

Forced convection relies on external mechanisms like fans or pumps to circulate fluid and improve heat transfer efficiency.

In natural convection, fluid movement happens through natural processes such as buoyancy. Because the fluid velocity in natural convection is generally low, the heat transfer coefficient associated with this method is also low.

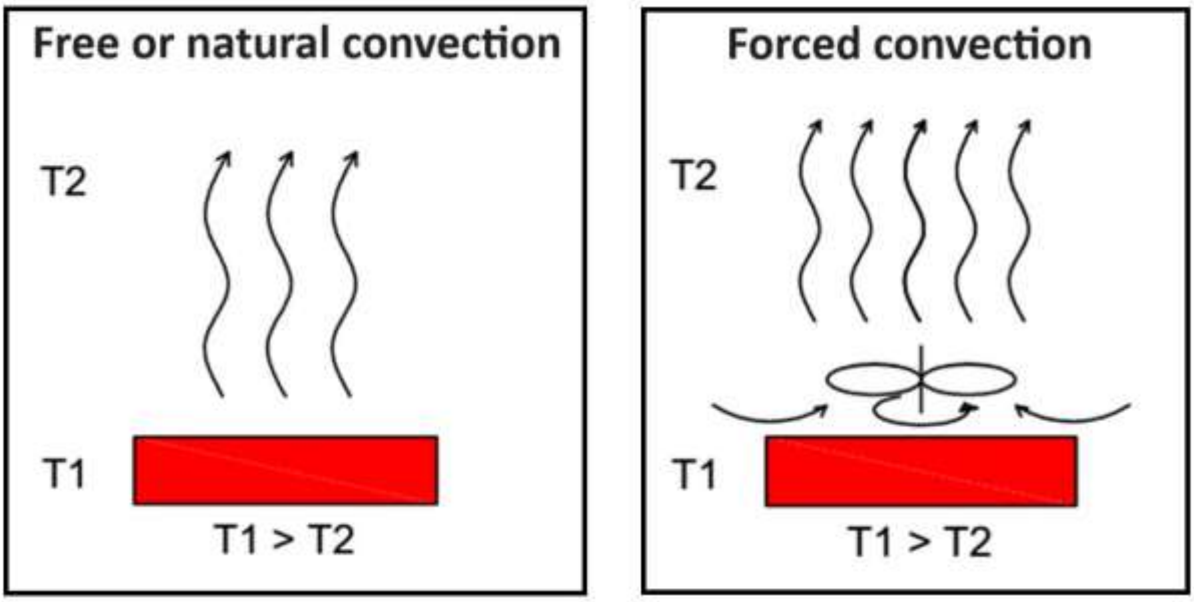


Fig. 1.2: Types of Convection [5]

1.1.2 Mechanism of Natural Convection

The buoyancy force behind the process can be described through Archimedes' principle. This principle simply explains that any fluid-containing body submerged in a fluid is subjected to an upward force equal to the weight of the fluid displaced. For the case of natural convection, as the portions of the fluid are heated up and expand, they in effect displace a quantity of surrounding cooler fluid. Since the warmer fluid is not as dense, it rises into the cooler and denser fluid surrounding it. These movements create a natural circulating pattern known as a convection current that facilitates conduction of heat in this fluid.

The buoyancy force F is responsible for this movement is expressed as:

$$F = \rho g V \tag{1.2}$$

Where ρ is the density of the fluid, g is the gravitational acceleration, and V is the volume of the displaced fluid. As temperature increases in certain areas of the fluid, ρ decreases, which reduces the weight of that region of fluid compared to the surrounding cooler areas. This density difference produces a net upward buoyancy force that drives the warmer fluid to rise.

Temperature gradient, in another word, is temperature difference between the heated surface and the fluid around the convection area. The basic example would be: assuming the bottom of the container is heated. Because it is warmer than its closest fluid layer, this warm layer of fluid expands. And because warm fluids rise up, this warmer, lower-density fluid goes upward while cooler fluid sinks into this space, continuing in an endless cycle or loop. This cycle of movement is called a convection current and helps in the efficient transfer of heat without mechanical agitation.

Some parameters that determine the convection current are the thermal expansion coefficient and viscosity of the fluid. The viscosity is the measure of resistance offered by the fluid when in motion. The thermal expansion coefficient is the extent to which the fluid expands or contracts with temperature changes. A fluid having a greater thermal expansion coefficient will undergo a greater density change for a given temperature difference, thus increasing the buoyancy forces and hence strengthening convection currents. However, viscosity is a force opposing fluid motion; highly viscous fluid will thus have slowly moving convection currents, which may result in reduced effectiveness of heat transfer. Natural convection plays an important role in both nature and engineering. In the atmosphere, for example, natural convection drives the formation of wind and weather patterns, as warm air rises and cooler air descends. In oceans, it creates currents essential for nutrient distribution and climate regulation. In engineering, natural convection is a way to cool systems, such as in heat sinks for electronics or building ventilation.

2. Literature

Boulahia et al. [1] examined how a cold block's height, Rayleigh number, and volume fraction affected the heat transfer of a nanofluid (Cu-water) for natural convection in a square enclosure. The current study examines the coordinate system and configuration of the enclosure under consideration. The container is filled with nanofluids and has a cold block within that is heated differentially. The temperature of the cold obstacle is kept at Tc. The horizontal walls are adiabatic, whereas the vertical walls are kept at hot (Th) and cold (Tc) temperatures, respectively. The nanofluid is supposed to be laminar, Newtonian, and incompressible. Both the nanoparticles and the base fluid are in a condition of thermal equilibrium.

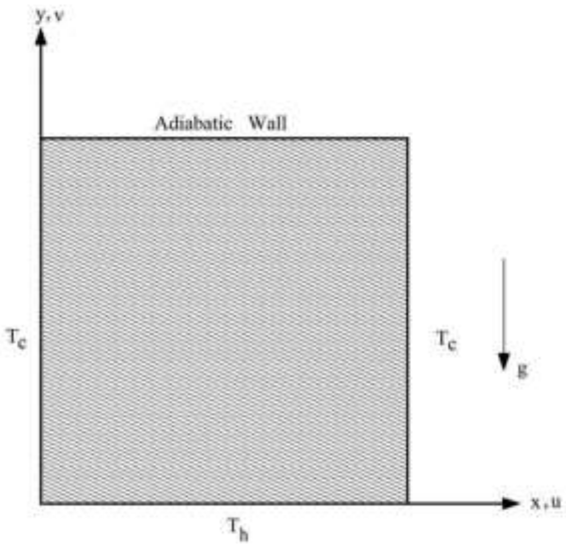


Fig. 2.1: Schematic diagram of the physical system [1]

Hussain et al. [2] studies that In a square enclosure filled with air (Pr = 0.7) and with all boundaries considered to be isothermal (at a constant low temperature), two-dimensional steady natural convection is carried out for a uniform heat source applied on the inner circular cylinder. The finite volume approach is used to solve the constructed mathematical model, which is governed by the coupled equations of continuity, momentum, and energy. The impacts of Rayleigh number and vertical cylinder positions on heat transfer and fluid flow performance are examined. Furthermore, a two-cellular flow field between the inner cylinder and the enclosure is produced by the numerical solutions. Additionally, the overall average Nusselt number exhibits nonlinear behavior in relation to places.

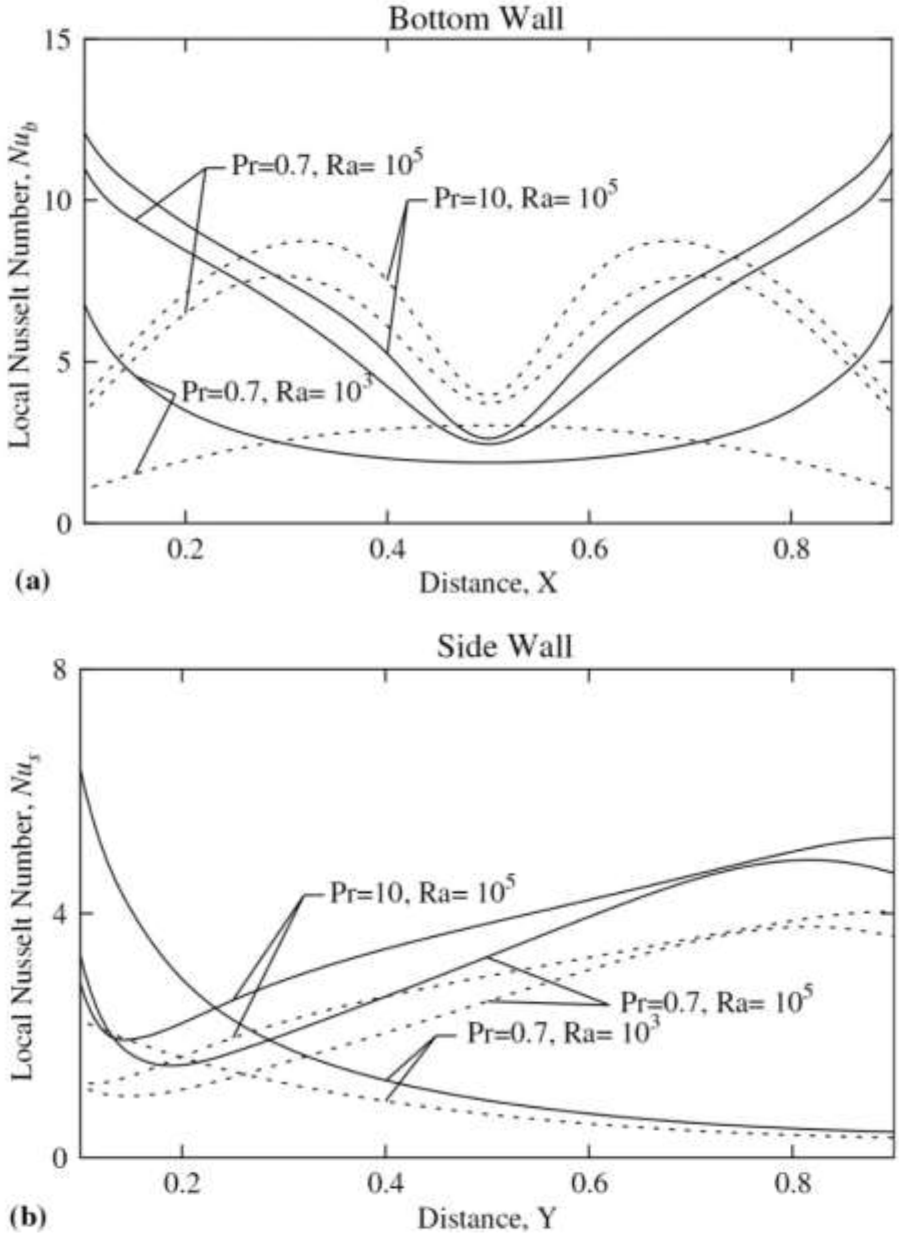


Fig. 2.2: Variation of local Nusselt number with distance (a) at the bottom wall (b) at the side wall for uniform heating and non-uniform heating [2]

Feng et al. [3] Using the Lattice Boltzmann Method (LBM), mathematical modeling is done to mimic the natural convection of Al2O3/water nanofluids in a vertical square box. The average Nusselt number rises as the Rayleigh number and particle volume concentration rise, according to the results. Under the same Rayleigh number, using a nanofluid has a greater average Nusselt number than using water. However, with a fixed temperature differential across the enclosure, the nanofluid's heat transfer rate is lower than that of water, mostly because of the notable increase in dynamic viscosity. Moreover, significant variations in calculated Nusselt numbers based on several models related to a nanofluid's physical characteristics are disclosed.

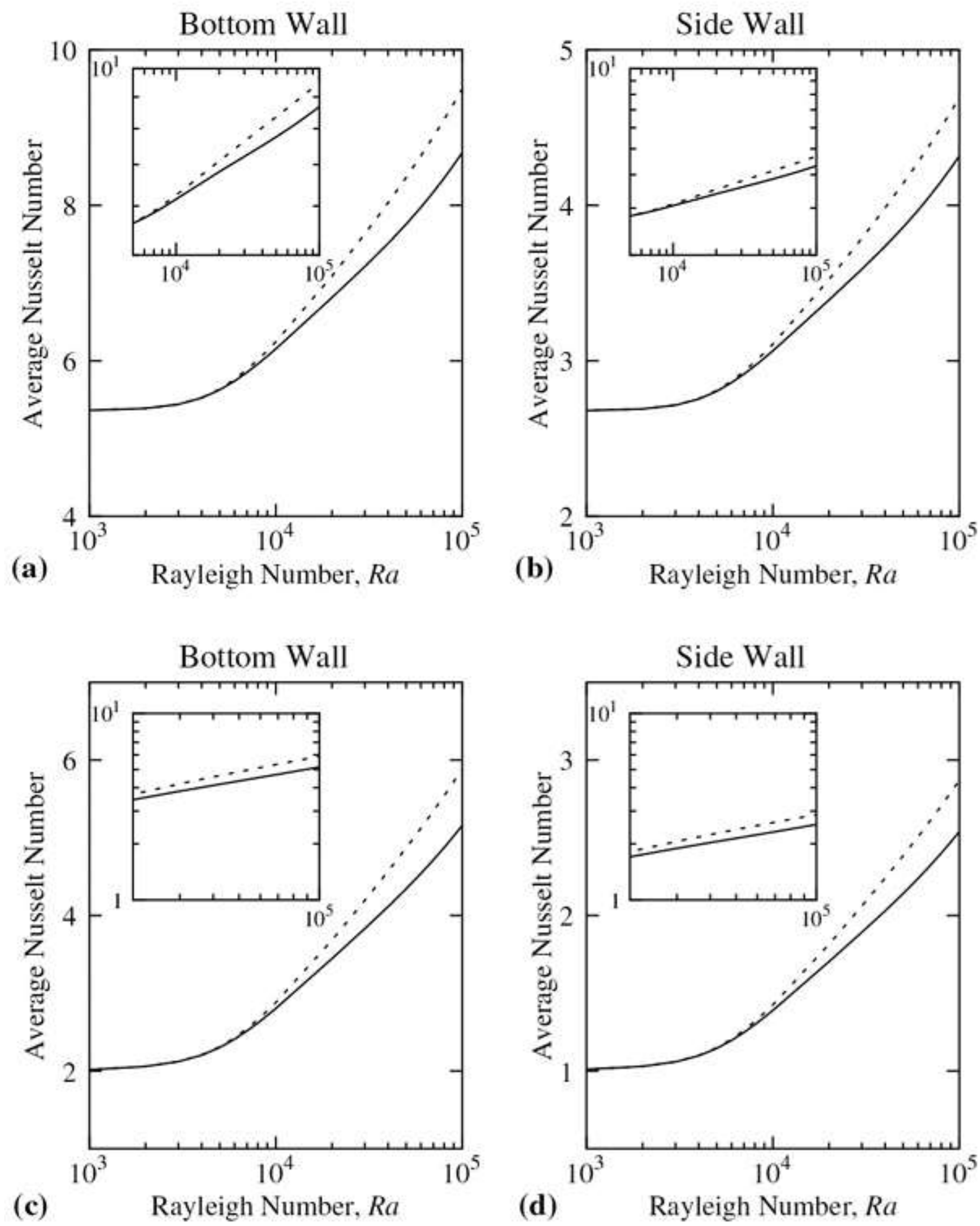


Fig. 2.3: Variation of average Nusselt number with Rayleigh number for uniform heating [(a) and (b)] and non-uniform heating [(c) and (d)] with Pr = 0.7 and Pr = 10 [3]

Mahmoodi et al. [4] Numerical research has been done on the heat transmission and free convection fluid flow of different water-based nanofluids in a square cavity with a thin heater within. The cavity's top and bottom sides are insulated, and its left and right walls are kept at a steady temperature Tc. Inside the hollow, which varies in length and location, is a thin heater with a temperature of Th (Th > Tc). The SIMPLER algorithm and the finite volume method were used to discretize the governing equations. A parametric analysis was conducted using the generated code to examine the impacts of relevant parameters on the fluid, including the Rayleigh number, the heater's position and location, the volume fraction of the nanoparticles, and different kinds of nanofluids.

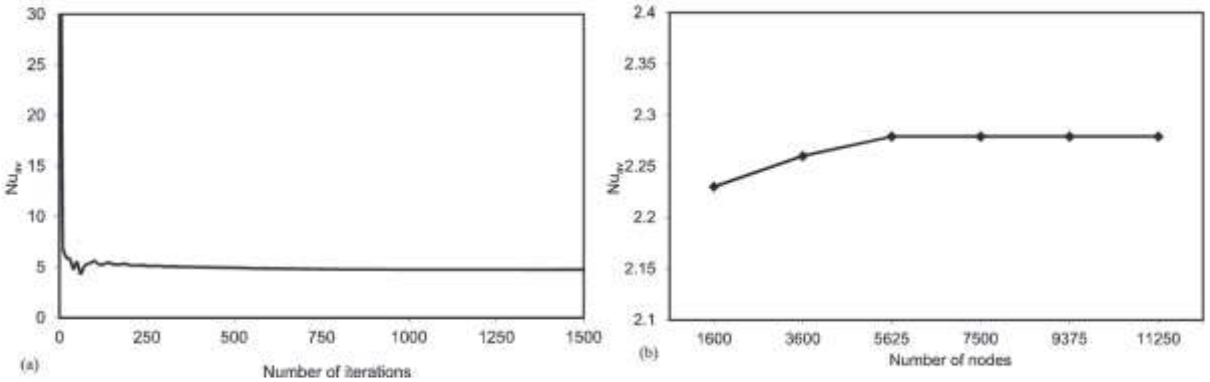


Fig. 2.4: Effect of (a) iterations and (b) nodes on average Nusselt number [4]

Roslan et al. [5] The use of nanofluids to improve heat transmission in a trapezoidal enclosure is examined for a number of relevant parameters. A stream-vorticity framework is used to describe transport equations, and a finite difference method is used to solve them numerically. Staircase-like zigzag lines are used to address the sloping, inclined boundaries. The impacts of the Grashof number, the sloping wall's inclination angle, the volume % of nanoparticles, and the enclosure's heat transfer rate on flow and temperature patterns are shown based on the numerical predictions. We tested water-Cu and water-Al2O3 nanofluids.

Tayebi at al. [6] It is analyzed numerically that natural convection occurs in an annulus between two confocal elliptic cylinders filled with a Cu-Al2O3/water hybrid nanofluid. The outer wall is isothermally chilled while the inside cylinder is heated to a constant surface temperature. Using the dimensionless form, the fundamental equations for 2D, laminar, and incompressible flow under steady-state conditions are constructed in terms of the vorticity-stream function formulation in elliptic coordinates. An internal FORTRAN algorithm is used to solve the governing equations after they have been discretized using the finite volume method. A range of nanoparticle volume fractions ($0 \leq \phi \leq 0.12$) and Rayleigh numbers ($103 \leq Ra \leq 3 \times 105$) are simulated numerically.

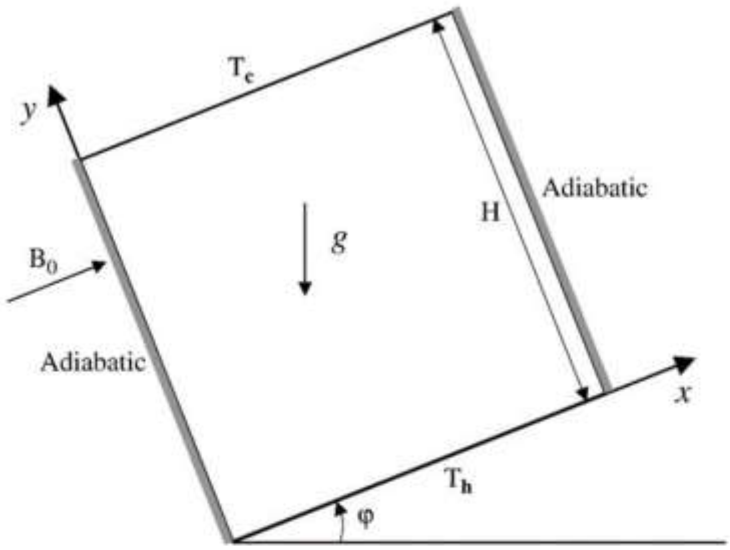


Fig. 2.5: Geometry and coordinates of cavity configuration with magnetic effect [6]

Shin et al. [7] investigation of the heat transfer and transitory phenomenon caused by a rotating rectangular item in a square cavity, followed by the periodic condition of laminar flow. Primitive variables are employed in a fixed-grid/sliding mesh computational approach based on finite volumes. In the center of a square cavity are revolving rectangular objects with varying aspect ratios (AR = 1, 2, 3, 4). At time t = 0, the stationary object is put into rotation with a fixed angular velocity. The hollow is kept as an isothermal enclosure for the insulated things and as a differentially-heated enclosure for the isothermal objects. Heat transport by natural convection is disregarded. For a fluid with Pr = 5, a range of rotating Reynolds numbers is covered for a given object form and constant angular velocity.

Usha et al. [8] investigated the convection of heat through a micropolar nanofluid in a rectangular, open container that was filled with a porous substance. The container has a heated non-planar bottom wall and a hot object inserted in it. Numerical comparisons are made between the impacts of three distinct shapes of the inserted heated object: diamond, L, and triangular. The nonlinear model momentum and energy equations are numerically addressed using the Gauss-Seidel iteration approach in conjunction with the successive over-relaxation method. The results are presented in terms of the average Nusselt number on the heated wall for various values of complex parameters, as well as streamlines, isotherms, and isolines of microrotation.

Bhowmick et al. [9] examined how a magnetohydrodynamic field affected natural convection in a square, porous enclosure with two heated circular cylinders inserted. The effects of the Hartmann number, Rayleigh number, Darcy number, and interspacing distance between the embedded cylinders on the thermal transport process and the production of total irreversibility are investigated numerically. It is found that while the distribution of streamlines is unaffected by the strength of the magnetic field, the isotherm distribution is significantly impacted by its presence. This emphasizes how the magnetic field has a significant impact on the properties of entropy formation and heat transmission. It demonstrates that a stronger magnetic field suppresses natural convection, as shown by the decrease in Nusselt number.

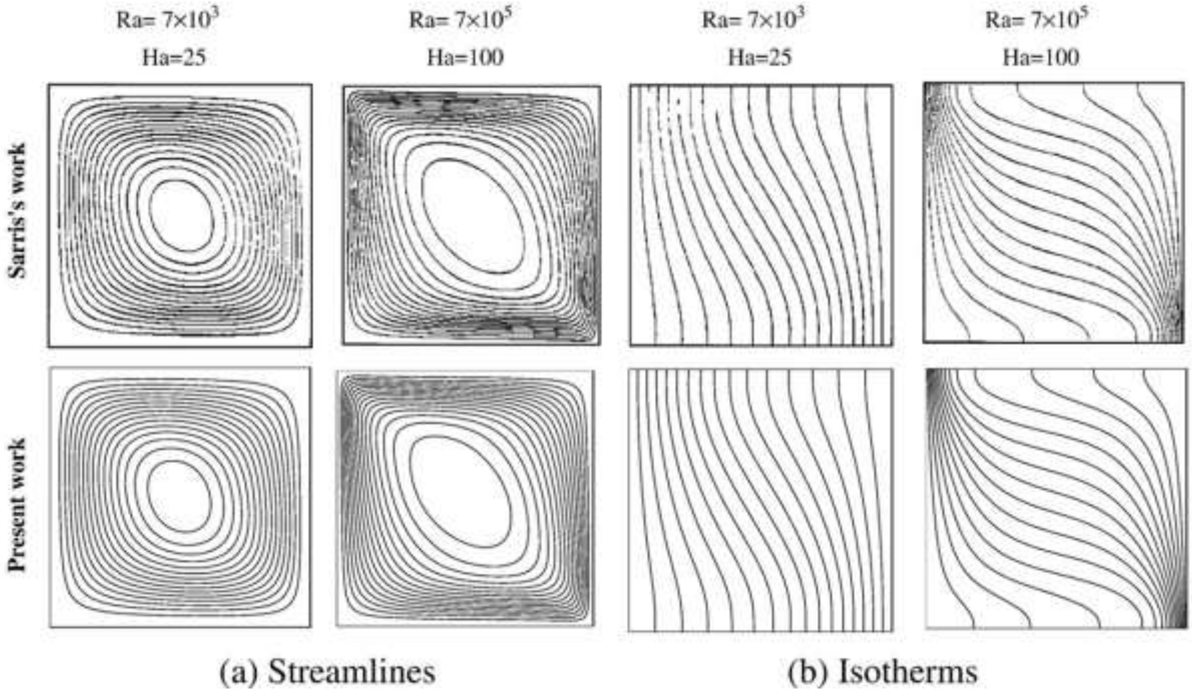


Fig. 2.6: (a) Streamlines and (b) Isotherms of natural convection in a square enclosure [9]

Basak et al. [10] The penalty finite element method has been used to numerically study natural convection flows in a square cavity filled with a porous matrix for both consistently and non-uniformly heated bottom walls and adiabatic top walls that maintain a constant temperature of cold vertical walls. The momentum transfer in the porous medium is simulated using the Darcy–Forchheimer model. With regard to continuous and discontinuous thermal boundary conditions, the numerical method used in this study produces consistent performance across a broad range of parameters (Rayleigh number Ra, 103 ⩽ Ra ⩽ 106, Darcy number Da, 10−5 ⩽ Da ⩽ 10−3, and Prandtl number Pr, 0.71 ⩽ Pr ⩽ 10).

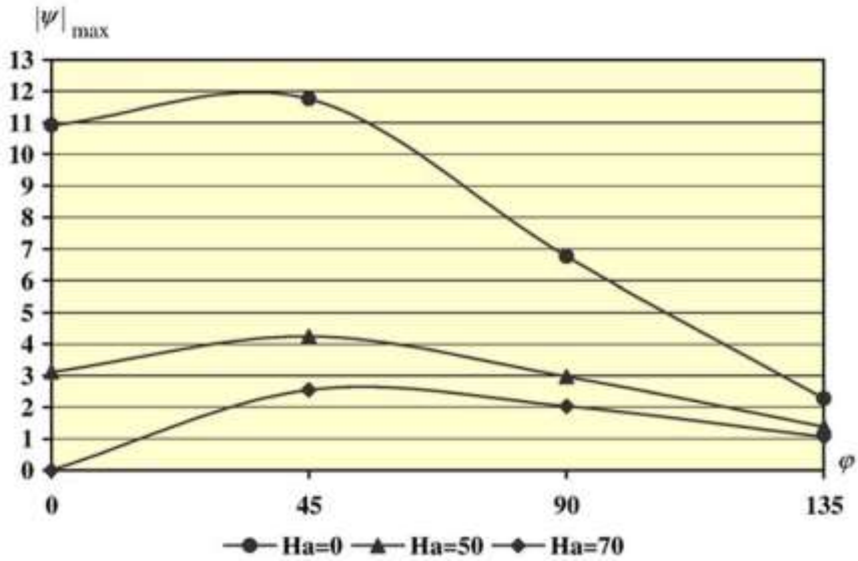


Fig. 2.7: Maximum stream function versus Ha for Ra = 10^5 [10]

Mishra et al. [11] In a square enclosure containing power-law fluids, two horizontal cylinders that are differently heated have been used to numerically study laminar natural convection. Two fundamental arrangements, where the cylinders are positioned at different angles and vertically, have been contemplated. The effect of the Grashof number (102–104), Prandtl number (0.7–100), and power-law index (0.2–2) for a variety of symmetric and asymmetric positions of the cylinders has been clarified by numerically solving the linked continuity, momentum, and energy equations. Plots showing the local and average Nusselt numbers for various cylinder locations, streamlines, and isothermal contours are used to visualize the temperature and velocity fields.

Mahmood et al. [12] Two distinct fluids are used as a heat transfer medium in a numerical investigation of natural convection from a horizontal cylinder within a square enclosure. There are two distinct heat transfer media: air and water. The cylinder has a diameter of 50 mm and a length of 500 mm. A tight square container measuring 300 mm by 300 mm is used to hold the cylinder. Surface temperatures ranging from 303 K to 414 K were covered by the range of working conditions. When air and water are the heat transfer media, natural convection is predicted using the two-dimensional Computational Fluid Dynamics (CFD) method.

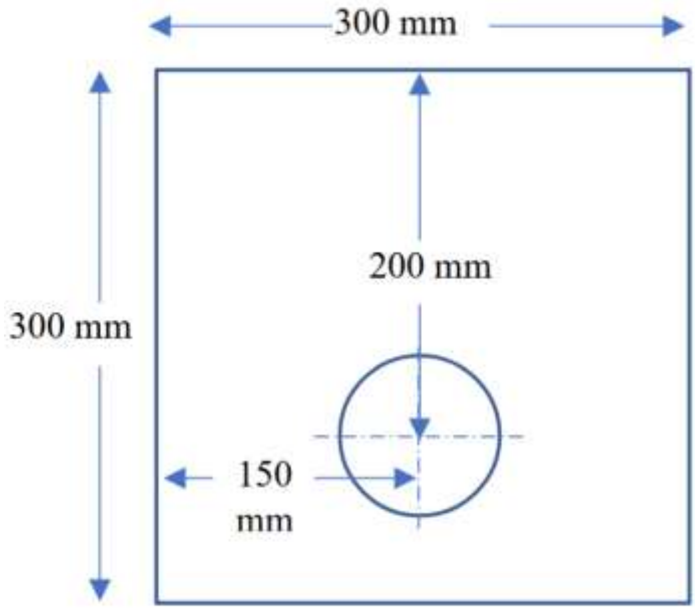


Fig. 2.8: Geometry of the horizontal cylinder and enclosure [12]

Minea et al. [13] examined the improvement of heat transport on a closed enclosure that was heated symmetrically from the lateral walls. The governing equations were solved using FLUENT's density-based solver. The simulation of radiation and convection in a closed domain was selected to solve the heat transfer problem. To examine the mesh spacing's independence in the numerical solution, one global variable, Nu, was tracked. The two enclosures under study, a rectangular one and an oval one, yielded results in terms of relative air rate and relative surface heat transfer coefficient. Two lateral panels were added to the enclosure, and the numerical research took into account their position as a variable. This increased air velocity by producing a chimney effect, which in turn enhanced natural convection.

Saied et al. [14] This article presents a numerical study of laminar natural convection with nonuniform side-wall temperature in a two-dimensional square cavity filled with pure air (Pr = 0.71). While the top and bottom walls are adiabatic, the heated vertical wall is thought to have spatial sinusoidal temperature changes around a constant mean value that is higher than the cold side-wall temperature. The nondimensional governing equations of the vorticity–stream function formulation are numerically solved using a finite-volume approach. The impact of the heated side-wall temperature variation's amplitude and wave number on the cavity's natural convection is examined. It is discovered that the hot-wall temperature affects the average Nusselt number.

5. Results

5.1 Model 1

The effects of nanoparticle concentration, Rayleigh number, geometry, and material on natural convection phenomena, represented through streamlines and isotherms. The shape CIRCLE is used with each embedded with three distinct nanoparticle materials: Copper (Cu), Iron Oxide (Fe_2O_3), and Silicon Carbide (SiC), dispersed in water. Each configuration is analyzed under varying nanoparticle volume concentrations (1%, 3%, and 5%) and Rayleigh numbers ranging from 10^3 to 10^6. The results, divided into streamlines and isotherms, portray the changes in fluid flow behaviour and heat transfer performance due to the aforementioned factors.

5.1.1 Effect of Streamlines for Model 1

The basic behavior seen in the streamlines, which show the fluid flow patterns surrounding the embedded nanoparticles, is firmly based in the physical implications of the Rayleigh number and the material's thermal properties. At lower Rayleigh numbers, such as 10^3, the streamlines tend to be sparse, subtle, and largely horizontal, signifying a regime dominated by pure conduction, where buoyancy forces are minimal and insufficient to trigger substantial fluid motion. As the Rayleigh number increases, particularly at 10^5 and 10^6, the streamlines evolve dramatically into well-defined circulation cells or vortices that depict the onset and dominance of natural convection. This change illustrates the increasing hotter fluid rises and cooler fluid falls due to buoyancy-driven flow, creating convective loops that are apparent through the compressed and intensified streamline structures.

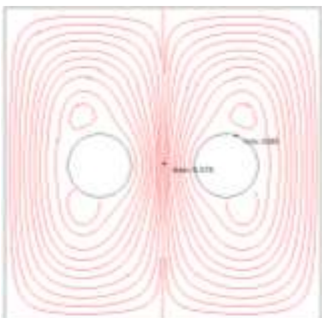
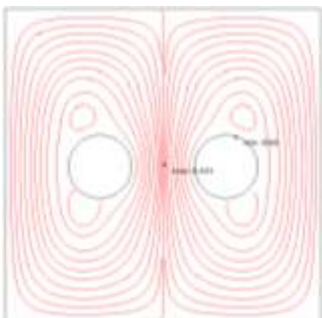
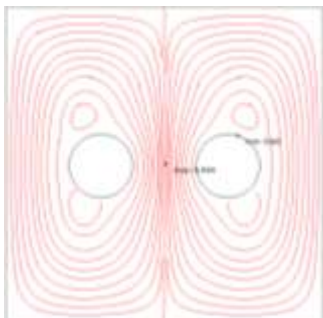
The impact of geometry on the streamlines is profound and consistent across all materials. Circular shapes typically exhibit more symmetric and centralized vortex formations due to their inherent geometric uniformity, which allows for even heat distribution and a balanced flow response. In contrast, circular shapes introduce that disturb the boundary layer development, leading to localized flow separations and secondary eddies in the corner regions. These features disrupt the symmetry and enhance flow complexity, especially at higher Rayleigh numbers. Which in turn generate multiple eddies and flow separation zones. These features enhance localized mixing and vortex strength, particularly under higher Ra and nanoparticle concentrations.

5.1.2 Effect of Isotherms for Model 1

The isotherms presented in the study provide a clear visualization of heat distribution within fluid-filled cavities that contain embedded nanoparticles of different materials and geometries. At low Rayleigh numbers, particularly at Ra = 10^3, the isotherms are mostly linear and horizontal, reflecting a conduction-dominated regime where thermal energy is transferred directly through molecular diffusion without significant fluid motion. In these scenarios, the effect of geometry and nanoparticle concentration is minimal, as natural convection is too weak to distort the thermal field.

As the Rayleigh number increases to 10^4 and beyond, natural convection begins to influence the temperature distribution. This is evident in the curvature of the isotherms, which start bending and spreading asymmetrically, especially near the hot surfaces of the embedded shapes. The increase in nanoparticle concentration enhances this behavior further. With higher concentrations, thermal conductivity improves, leading to a stronger coupling between thermal gradients and buoyant forces, which in turn amplifies the convective motion. This results in isotherms that are more distorted and denser near the heated surfaces, especially around sharp corners in triangular and square shapes.

Material plays a significant role, with copper consistently exhibiting the most pronounced isotherm distortion due to its superior thermal conductivity. In contrast, Fe_2O_3 shows the least curvature, maintaining a more conduction-like appearance even at high Rayleigh numbers and particle concentrations. SiC lies between the two, with moderate curvature and distribution changes. Among the shapes, the triangle causes the sharpest local gradients due to its acute vertices, concentrating heat and distorting isotherms most intensely. The square, with its flat faces and corners, also induces strong local gradients, while the circle distributes heat more uniformly due to its symmetry. Overall, the progression of isotherm behavior from smooth to highly distorted with increasing Ra and concentration captures the transition from conduction to convection and highlights the interplay between geometry, material, and nanoparticle enhancement in thermal transport.

Cu water - Circle	Stream lines		
	1%	3%	5%
Ra=10^3			

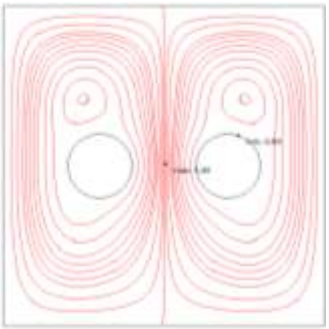
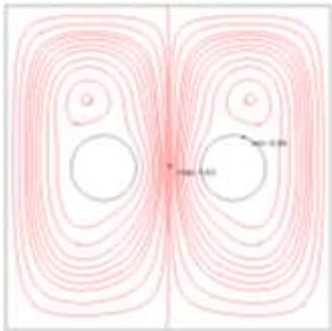
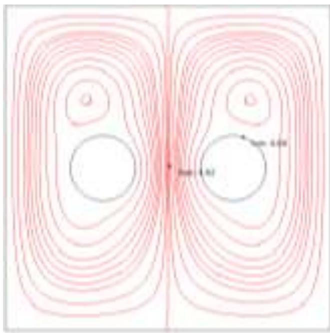
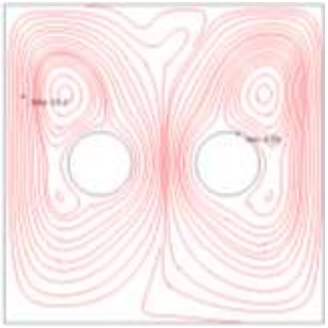
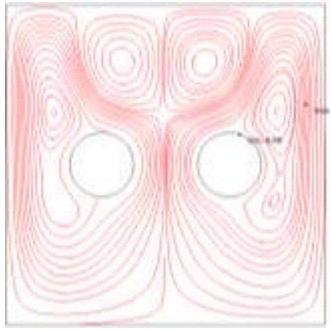
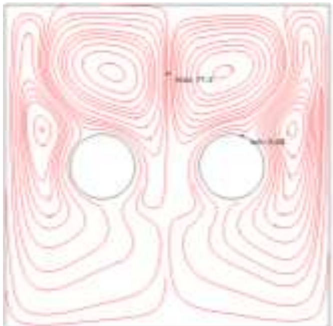
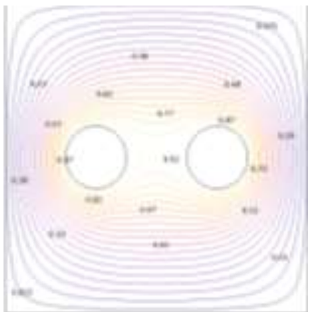
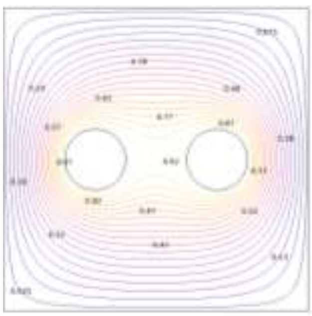
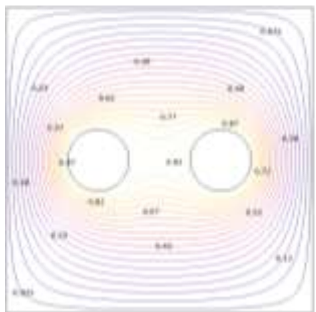
Ra=10^4			
Ra=10^5			
Ra=10^6			

Fig. 5.1: Stream lines at Rayleigh's number 10^3, 10^4, 10^5 and 10^6 at volume fractions 1%, 3% and 5% for Copper (Cu) circle

Cu water - Circle	Isotherms		
	1%	3%	5%
Ra=10^3			

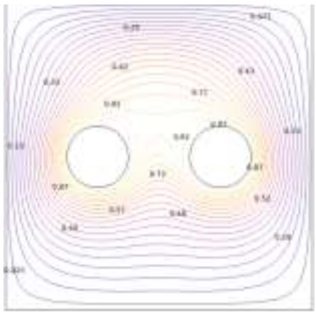
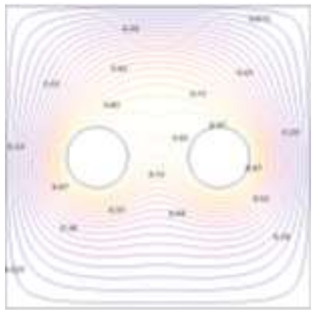
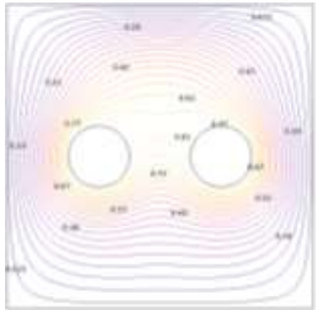
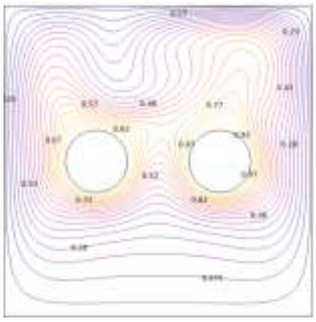
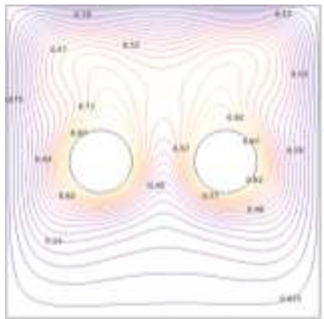
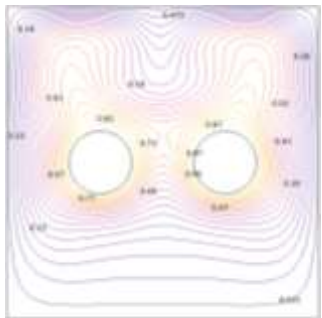
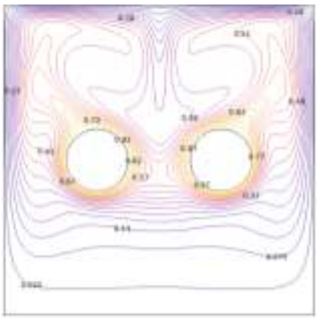
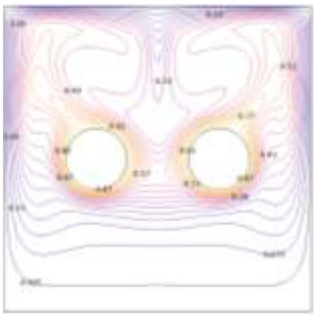
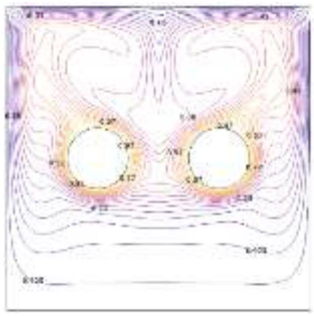
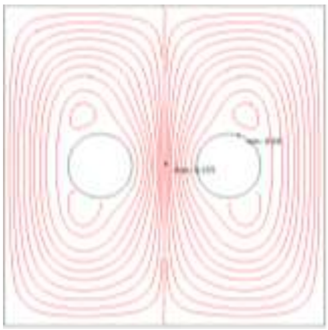
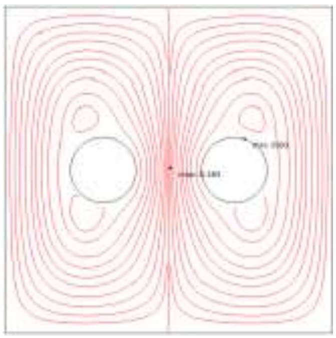
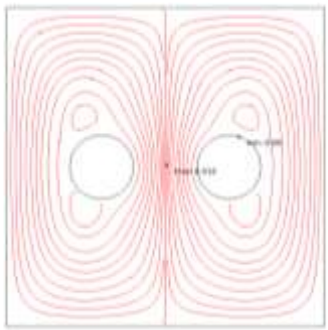
Ra=10^4			
Ra=10^5			
Ra=10^6			

Fig. 5.2: Isotherms at Rayleigh's number 10^3, 10^4, 10^5 and 10^6 at volume fractions 1%, 3% and 5% for Copper (Cu) circle

Fe_2O_3 - Circle	Stream lines		
	1%	3%	5%
Ra=10^3			

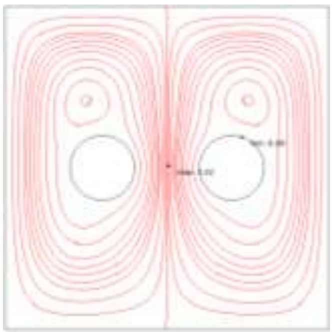
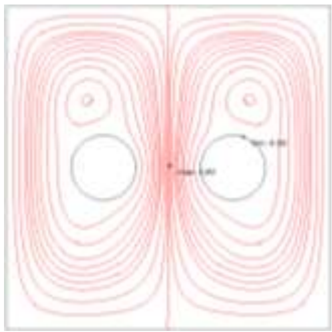
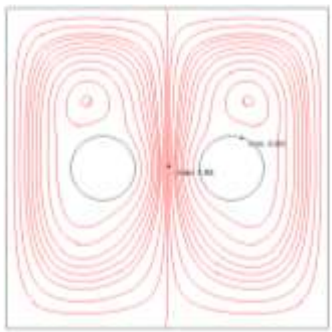
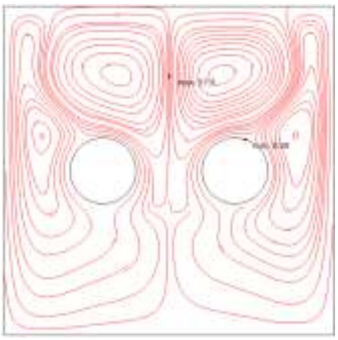
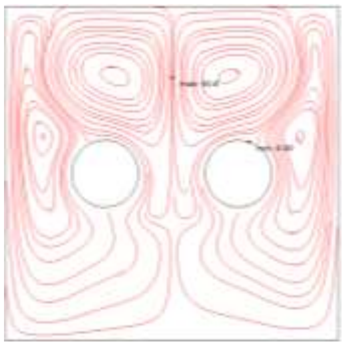
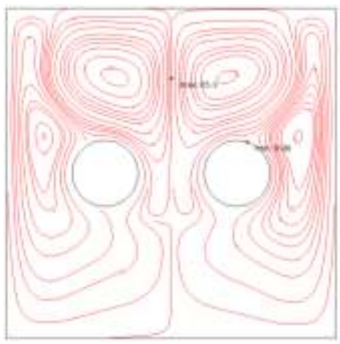
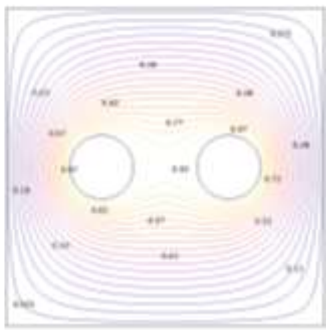
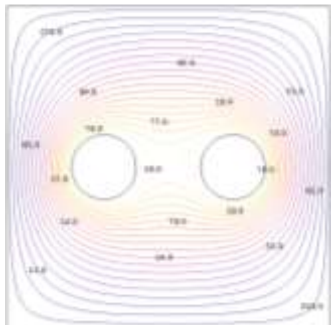
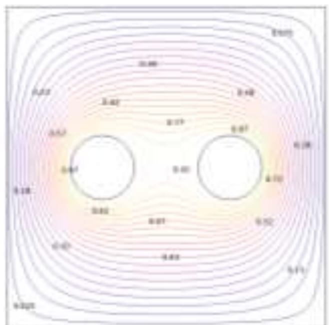
Ra=10^4			
Ra=10^5			
Ra=10^6			

Fig. 5.6: Stream lines at Rayleigh's number 10^3, 10^4, 10^5 and 10^6 at volume fractions 1%, 3% and 5% for Ferrous Oxide (Fe_2O_3) circle

Fe_2O_3 - Circle	Isotherms		
	1%	3%	5%
Ra=10^3			

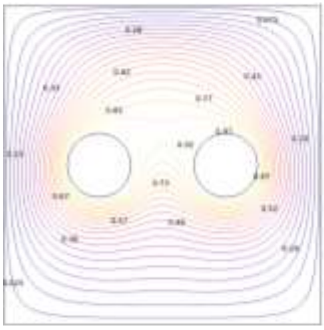
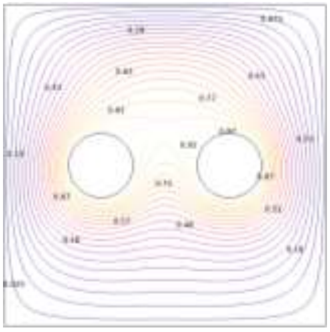
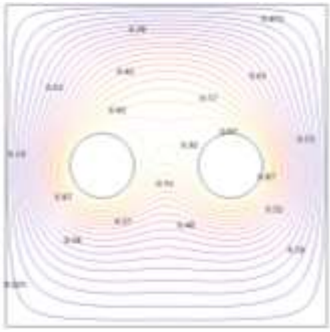
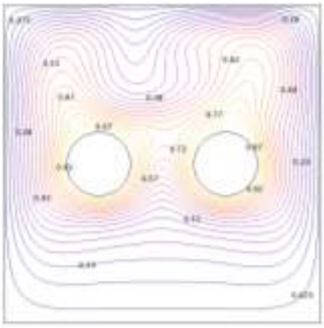
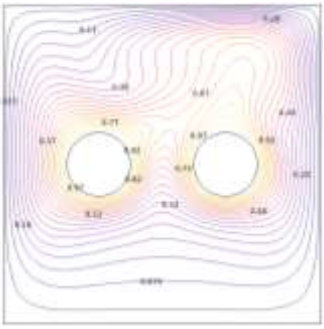
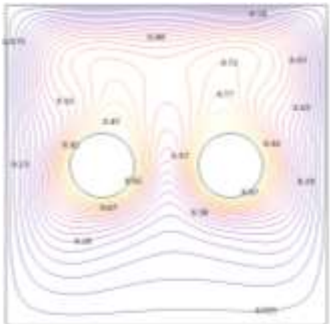
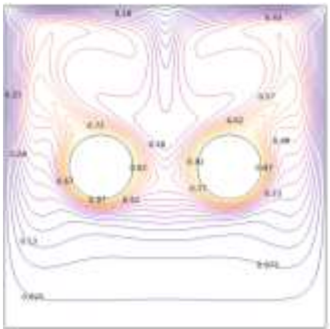
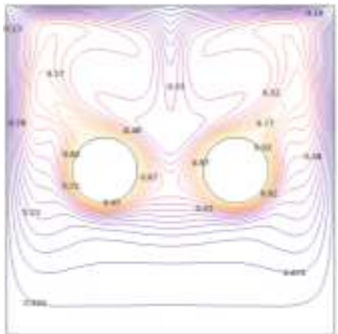
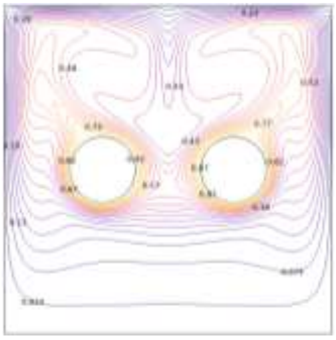
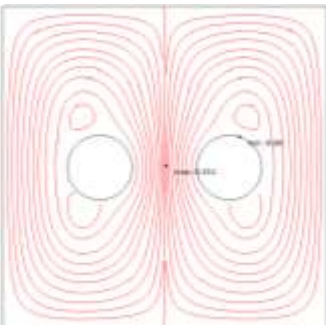
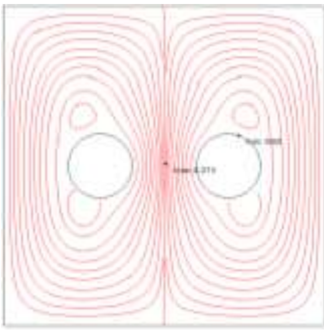
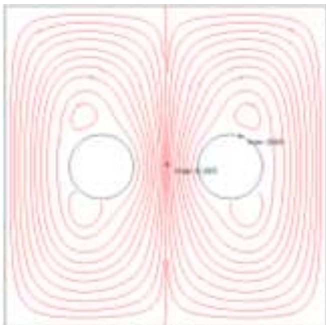
$Ra=10^4$			
$Ra=10^5$			
$Ra=10^6$			

Fig. 5.7: Isotherms at Rayleigh's number 10^3, 10^4, 10^5 and 10^6 at volume fractions 1%, 3% and 5% for Ferrous Oxide (Fe_2O_3) circle

SiC - Circle	Stream lines		
	1%	3%	5%
$Ra=10^3$			

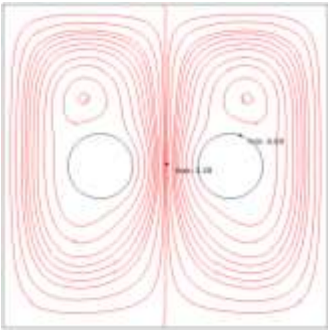
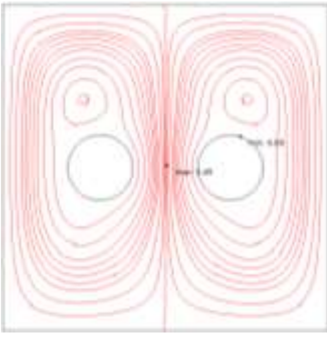
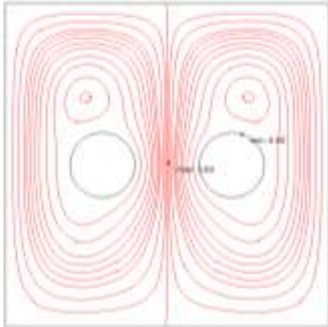
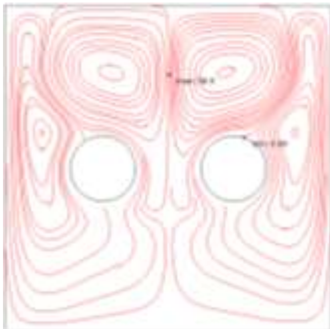
Ra=10^4			
Ra=10^5			
Ra=10^6			

Fig. 5.11: Stream lines at Rayleigh's number 10^3, 10^4, 10^5 and 10^6 at volume fractions 1%, 3% and 5% for Silicon Carbide (SiC) circle

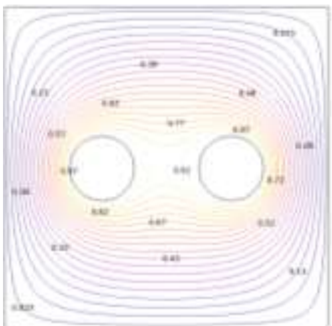
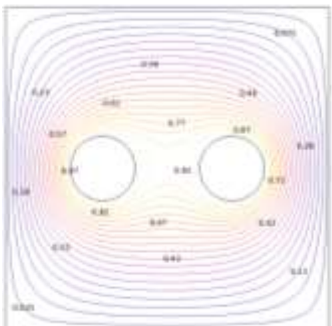
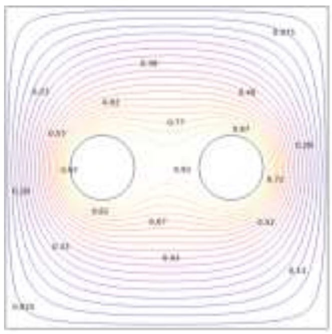
SiC - Circle	Isotherms		
	1%	3%	5%
Ra=10^3			

Fig. 5.12: Isotherms at Rayleigh's number 10^3, 10^4, 10^5 and 10^6 at volume fractions 1%, 3% and 5% for Silicon Carbide (SiC) circle

Conclusion

The present study investigates simulation of flow of nanofluid in a square container with hot items within. Straightlines, isotherms, Nusselt numbers for $10^3 \leq Ra \leq 10^6$, $1\% \leq \leq 5\%$, and the spacing between them S=0.2 are used to display the results. Below is a description of the key findings:

- In a square enclosure for circular geometry (Model 1), the maximum surface-averaged Nusselt number was observed in SiC base water nanofluid. At a low Ra of 10^3, the Nusselt number was 2.84 for ϕ = 1%, 2.829 for ϕ = 3%, and 2.82 for ϕ = 5%. As the volume fraction was increased the Nusselt number decreased, SiC nanofluid yielded at ϕ = 1%, showing the strengthening of natural convection currents. For Fe_2O_3 and Cu base water nanofluid the Nusselt numbers are slightly decreased due to the density variations and Brownian motion of the nanofluid.
- In the square object configuration (Model 2), the influence of Ra and ϕ followed a similar trend, although the overall Nu_t values were slightly lower due to the geometric resistance created by sharp corners, which hinder the formation of strong convective cells. At Ra = 10^3 and ϕ = 1%, SiC nanofluid showed a Nusselt number of approximately 2.38 which decreased 16% compared to the square heated object, The maximum value for square geometry was recorded at Ra = 10^6, where SiC reached its value 5.90 at ϕ = 1% achieved. This performance gap between circular and square objects becomes more pronounced at higher Rayleigh numbers.

References

[1] Boulahia, Z., Wakif, A., & Sehaqui, R. (2016). Natural convection heat transfer of the nanofluids in a square enclosure with an inside cold obstacle. International journal of innovation and scientific research, 21(2), 367-375.

[2] Hussain, S. H., & Hussein, A. K. (2010). Numerical investigation of natural convection phenomena in a uniformly heated circular cylinder immersed in square enclosure filled with air at different vertical locations. International Communications in Heat and Mass Transfer, 37(8), 1115-1126.

[3] Lai, F. H., & Yang, Y. T. (2011). Lattice Boltzmann simulation of natural convection heat transfer of Al2O3/water nanofluids in a square enclosure. International journal of thermal sciences, 50(10), 1930-1941

[4] Mahmoodi, M. (2011). Numerical simulation of free convection of nanofluid in a square cavity with an inside heater. International Journal of Thermal Sciences, 50(11), 2161-2175

[5] Saleh, H., Roslan, R., & Hashim, I. (2011). Natural convection heat transfer in a nanofluid filled trapezoidal enclosure. International journal of heat and mass transfer, 54(1-3), 194-201.

[6] Tayebi, T., & Chamkha, A. J. (2016). Free convection enhancement in an annulus between horizontal confocal elliptical cylinders using hybrid nanofluids. Numerical Heat Transfer, Part A: Applications, 70(10), 1141-1156.

[7] Shih, Y. C., Khodadadi, J. M., Dai, H. W., & Fan, L. (2009, January). Periodic Fluid Flow and Heat Transfer in a Square Cavity Due to an Insulated or Isothermal Rotating Rectangular Object. In Heat Transfer Summer Conference (Vol. 43574, pp. 853-862).

[8] Ushachew, E. G., Sharma, M. K., & Makinde, O. D. (2021). Heat convection in micropolar nanofluid through porous medium-filled rectangular open enclosure: effect of an embedded heated object with different geometries. Journal of Thermal Analysis and Calorimetry, 146(4), 1865-1881.

[9] Bhowmick, D., Chakravarthy, S., Randive, P. R., & Pati, S. (2020). Numerical investigation on the effect of magnetic field on natural convection heat transfer from a pair of embedded cylinders within a porous enclosure. Journal of Thermal Analysis and Calorimetry, 141(6), 2405-2427.

[10] Basak, T., Roy, S., Paul, T., & Pop, I. (2006). Natural convection in a square cavity filled with a porous medium: effects of various thermal boundary conditions. International Journal of Heat and Mass Transfer, 49(7-8), 1430-1441.

[11] Mishra, L., & Chhabra, R. P. (2018). Natural convection in power-law fluids in a square enclosure from two differentially heated horizontal cylinders. Heat Transfer Engineering, 39(10), 819-842.

[12] Mahmood, R. A., Ibrahim, A. K., Kamilxy, A. G. M., & Saeed, R. I. (2023, July). Natural convection from a horizontal cylinder placed in a square enclosure: CFD simulations. In AIP Conference Proceedings (Vol. 2830, No. 1). AIP Publishing.

[13] Minea, A. A. (2015). Numerical studies on heat transfer enhancement in different closed enclosures heated symmetrically. Journal of Thermal Analysis and Calorimetry, 121(2), 711 720.

[14] Saeid, N. H., & Yaacob, Y. (2006). Natural convection in a square cavity with spatial side-wall temperature variation. Numerical Heat Transfer, Part A: Applications, 49(7), 683-697.