International Journal of Heat and Technology
Vol. 43, No. 5, October, 2025, pp. 1817-1825
Journal homepage: http://iieta.org/journals/ijht

Numerical Investigation of Natural Convection Heat Transfer Around Cylinder with Different Configurations Inside Inclined Square Enclosure

Nagham Q. Shari[1], Sarmad A. Ali[2*]

[1] Department of Mechanical Engineering, College of Engineering, Wasit University, Al-Kut 52001, Iraq
[2] Department of Automobile Engineering, College of Engineering-Al Musayab, University of Babylon, Hillah 51001, Iraq

Corresponding Author Email: sarmad.ahmed96@uobabylon.edu.iq

https://doi.org/10.18280/ijht.430520

Received: 28 May 2025
Revised: 5 September 2025
Accepted: 17 September 2025
Available online: 31 October 2025

Keywords:
natural convection, square enclosure, Rayleigh number, laminar flow, streamline, Nusselt number, configurations

ABSTRACT

Recent years have witnessed the study of natural convective heat transfer inside enclosures on a large scale because this automatic method of heat transfer has a wide range of industrial engineering applications. The current work involved the use of one of the numerical simulation tools program (Ansys Fluent) to study the effect of the inclusion of three different configurations (circular, rectangular and square) with the same hydraulic diameter of (D_H = 0.15 m) inside the middle of a square two-dimensional enclosure inclined at an angle (45°) on heat transfer by natural convection and laminar air flow as a working fluid at the value of the Prandtl number (0.708) within the ranges of Rayleigh numbers ($10^6 \leq Ra \leq 10^9$) boundary conditions were applied by insulating the upper and lower walls and projecting cold temperatures for the right and left wall while the cylinder perimeter is exposed to hot temperature. For continuity, momentum, and energy equations, the laminar Boussinesq approximation is employed with convergence requirements of fewer than (10^{-6}). The numerical results indicated an increase in the Rayleigh numbers, and the Nusselt number also gradually increases as a result of the density difference, the buoyancy force is generated, therefore, a gradient of temperature and speed occurs with the appearance of vortices. The results of this study were displayed using a variety of visual aids. Visual depictions of the flow and temperature distribution inside the inclined square enclosure will include streamlines, temperature contours, and heat transfer enhancement. For various engineering applications, this study may provide valuable insights into the effective design of thermal systems in addition to improving heat transfer.

1. INTRODUCTION

As a result of large-scale technological development, natural convective heat transfer has been used as an effective method of heat dissipation in various modern industries, as well as its use in other applications, such as cooling systems for nuclear reactors and electronic devices such as high-speed computer processors, as well as auxiliary systems in solar collectors, as well as used in some applications of agricultural industries. Industrial engineering applications are important for natural convection, for example, the Square and varied geometry of the container filled with working fluid and vertical surfaces. Researchers have been motivated to enhance the characteristics of those traditional fluids due to the numerous thermal applications associated with convective heat transfer utilizing a fluid medium. Many attempts represented by investigations and behaviors are used as experiments to improve the efficiency and performance of convective heat transfer, where, for comparison, the coefficient of heat transfer by forced convection is greater than the coefficient of heat transfer by natural convection (free) [1-5]. Lee and Goldstein [6] presented an experimental study of heat transfer by natural convection inside a square container tilted at different angles (0, 15, 30, and 45 degrees) with laminar flow of the Rayleigh number range (10-1.5). Two extreme temperature values are shown, the temperature gradient is at the oblique angle (0 and 15 degrees), and one value is shown at the other angles (30 and 45 degrees). Alshuraiaan [7] presented a research work using one of the tools of numerical simulation (COMSOL Multiphysics) on the interaction of the structure of liquids with the properties of heat transfer by natural convection inside a cavity in the form of the letter (L). Several parameters have been studied, including Rayleigh numbers and Wall elasticity. Barik and Al-Farhany [8] conducted a numerical analysis using one of the simulation tools (COMSOL Multiphysics) was reviewed to study free convection heat transfer and nanofluid flow inside a square container. The left wall of constant thickness is exposed to a hot temperature, the right wall is exposed to a cold temperature, and the upper and lower walls are thermally insulated. A baffle at different angles is inserted into the center of the lower wall of the cavity. Several numerically verified parameters, such as the effect of volumetric fractions of nanomaterials, Rayleigh number, and angles of inclination of the baffle and the thickness of the solid left wall on temperature lines, Nusselt numbers, and flow patterns. Ternik and Rudolf [9] presented a numerical study using the finite volume method to optimize heat transfer by natural convection inside a square-shaped cavity. Various nanomaterials have been added, including (Au, Al_2O_3, Cu, and TiO_2), with the primary working fluid (water) as an improvement technique. The simulation process is within the range of Rayleigh numbers (10^3-10^5) and with volumetric fractions (0-0.1). The results of the study showed that the rate of the Nusselt number gradually increases with increasing volumetric fractions and also with increasing Rayleigh numbers. In addition, enhancing heat transfer using nanofluids is effective compared to ordinary conventional liquids. Lugarini et al. [10] presented a numerical analysis of the study of heat transfer by free natural convection combined with the process of surface irradiation employing an open fracture of a solid wall facing a reservoir containing a calm isothermal fluid. The fracture is designed as a regular C-shaped path through the wall, with the vertical surface heated and the horizontal surface isothermal. Numerical simulation and mathematical modeling of the phenomenon of thermal radiation in a systematic parametric study of the thermal process as influenced by changes in the size of the fracture channel, changes in the volume of the solid center section (0-1), surface emissivity (0-1), Rayleigh numbers (10^5-10^8) and the Prandtl number (0.71). It should be noted that the fully enclosed (no opening) arrangement with a single block of fixed [11-14] and variable aspect ratios [15-18] is advanced by the research of natural convection in side-open cavities with a single solid within the cavity. Miroshnichenko and Sheremet [19] contributed to the use of numerical and experimental techniques to study heat transfer by turbulent flow inside rectangular cavities by reviewing the previous literature on the subject by investigating various parameters such as the geometric shape of cavities, angles of inclination, Rayleigh and Prandtl numbers, surface emissivity, as well as thermal properties. Corcione et al. [20] presented a numerical study to improve the thermophysical properties by incorporating a nanomaterial (Al_2O_3) added to the basic single-phase fluid with laminar flow inside a square cavity. The equations governing the flow (mass, momentum, and thermal energy) were solved using one of the numerical solution tools (Ansys software). There are simulations for a range of values of the nanoparticle volume fraction (0-0.06), the diameter of the suspended nanoparticles (25-100 nm), the temperature of the heated sidewall (range 298-343 K), the temperature of the cooled sidewall range of (293-313 K), and the Rayleigh number of the base fluid range of (10^3-10^7). The results of the study indicated that with increasing volume fraction of the nanomaterial, the average Nusselt number gradually increases.

The purpose of the current research work is to use one of the numerical simulation tools (Ansys Fluent 2023 R2) to study the issue of heat transfer by free convection and laminar flow of the working fluid (air) with the Prandtl number (0.708) within the ranges of Rayleigh numbers ($10^6 \leq Ra \leq 10^9$) inside a square enclosure inclined at an angle (45°) by inserting different shapes of the cylinder (circular, rectangle and square) in the center line of the enclosure by applying boundary conditions where the upper and lower walls are thermally insulated, the right and left walls are exposed to a cold temperature, in addition, the cylinder walls (perimeter) subjected to a hot temperature.

2. COMPUTATIONAL MODEL WITH COORDINATE SYSTEM

Figure 1 shows the current working scenario where the numerical investigation involves the analysis of two-dimensional steady-state, incompressible, and laminar flow as well as heat transfer by natural convection inside a square enclosure inclined at an angle (Ø = 45°). The dimensions of the enclosure are equal in terms of length (L = 0.4 m) and height (H = 0.4 m). The boundary conditions were applied to the physical model by projecting a cold temperature on the right and left walls (Tc = 300 K) while the upper and lower walls were thermally insulated. The cylinder was inserted with three different configurations (circular, rectangular, and square) with the same hydraulic diameter of (D_H = 0.15 m) in the center of the enclosure, as shown in Figure 2, and the cylinder wall was subjected to a hot temperature. Air was used as a working fluid inside the enclosure, which is characterized by a Prandtl number with a value of (Pr = 0.708).

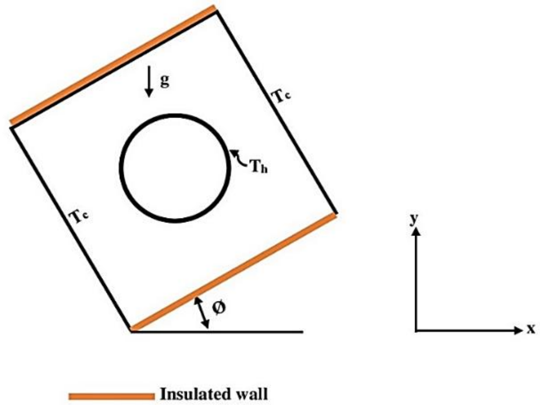

Figure 1. Schematic diagram of the inclined square enclosure at an angle (45°)

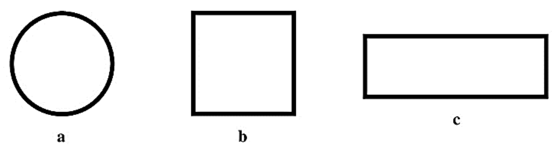

Figure 2. Various configurations of cylinder: (a) circular, (b) square, and (c) rectangular

3. FLOW GOVERNING EQUATIONS

In the current numerical work, the governing equations of the two-dimensional laminar flow of the incompressible fluid inside the inclined square enclosure are presented by the Cartesian formula as follows [21-24]:

The equation of continuity:

$$\frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial x} = 0 \qquad (1)$$

The equation of momentum in the x and y direction:

$$\frac{\partial(\rho u^2)}{\partial x} + \frac{\partial(\rho uv)}{\partial y} = -\frac{\partial p}{\partial x} + \frac{\partial}{\partial x}(\mu\frac{\partial u}{\partial x}) + \frac{\partial}{\partial y}(\mu\frac{\partial u}{\partial y}) \qquad (2)$$

$$\frac{\partial(\rho uv)}{\partial x} + \frac{\partial(\rho v^2)}{\partial y} = \rho g_y - \frac{\partial p}{\partial x} + \frac{\partial}{\partial x}(\mu\frac{\partial v}{\partial x}) + \frac{\partial}{\partial y}(\mu\frac{\partial v}{\partial y}) \qquad (3)$$

The heat energy equation:

$$\frac{\partial}{\partial x}(\rho u C_p T) + \frac{\partial}{\partial y}(\rho v C_p T) = \frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right) \tag{4}$$

In the study of convective heat transfer within Enclosures, dimensionless and thermodynamic equations are essential for understanding and analyzing the physical phenomena affecting fluid motion and thermal energy transfer. The Grashof number is used to estimate the effect of buoyancy forces resulting from temperature differences compared to viscous forces, while the Prandtl number expresses the relationship between kinetic diffusion and thermal diffusion of a fluid. The Rayleigh number combines the Grashof and Prandtl effects to reflect the ability of natural convection to transport heat within a system. The heat flux rate is used to determine the amount of thermal energy transferred across surfaces, while the Nusselt number indicates the ratio of convective heat transfer to conductive heat transfer, an important indicator of the heat transfer efficiency in the studied system [25-28].

$$Gr = \frac{g\beta(T_h - T_c)L_{ca}^3}{\upsilon^2} \tag{5}$$

$$Pr = \frac{\upsilon}{\alpha} = \frac{\mu C_p}{k} \tag{6}$$

$$Ra = Gr \times Pr = \frac{g\beta(T_h - T_c)L_{ca}^3}{\alpha\upsilon} \tag{7}$$

For a specific liquid, tables of thermophysical properties can be used to determine the coefficient of volume expansion (β), but for ideal gases, the following equation can be used according to the following:

$$\beta = 1/K \tag{8}$$

$$T_h = \frac{Ra \times \alpha \times \upsilon}{g \times \beta \times L_{ca}^3} + T_c \tag{9}$$

The Thermophysical properties of the working fluid at 300 K used in numerical calculations for natural convection heat transfer inside the square cavity are given in Table 1. These are dynamic viscosity (μ), specific heat at constant pressure (Cp) and thermal conductivity (k) together with the Prandtl number (Pr), thermal diffusivity (α), density (ρ) and kinematic viscosity (ν). These parameters are necessary to obtain the thermal and hydrodynamic characteristics of fluid during heat transfer in studied region.

Table 1. Thermophysical properties of the working fluid at a temperature of (300 K)

Variable	Value	Unit
μ	1.8462×10^{-5}	kg. m/s
Cp	1.0057	kJ/kg. K
k	0.02624	W/m. K
Pr	0.708	--------
α	0.22160×10^{-4}	m^2/s
ρ	1.1774	kg/m^3
υ	15.69×10^{-6}	m^2/s

Heat transfer by natural convection is represented by the Nusselt number, which is calculated using the following equation:

$$q^{"} = h(T_h - T_c) \tag{10}$$

$$Nu = \frac{h \times L_{ca}}{k} \tag{11}$$

$$q^{"} = \frac{Nu \times k}{L_{ca}}(T_h - T_c) \tag{12}$$

4. GENERATION MESH OF NUMERICAL MODEL

The numerical study was carried out using a commercial code of an organized process for generating an angled two-dimensional square enclosure model (45°) with the placement of three cylinders of equal hydraulic diameter, as shown in Figure 3. The first step is to build a mathematical model with the specified dimensions in terms of length, height, and diameter of the cylinder, after which the boundary conditions of the matter are determined, including the insulated and cold walls and the hot cylinder wall. The next step in preprocessing is to generate a mesh, which serves as a framework for computations, after the model has been set up. The accuracy of the CFD simulation results is strongly impacted by the mesh's quality, as shown in Figure 4. Nevertheless, there is a trade-off between hardware capabilities, computing time, and precision. A computer with significant processing capability is needed to produce extremely precise results. The continuity and momentum equations controlled how the model behaved in the fluid domain.

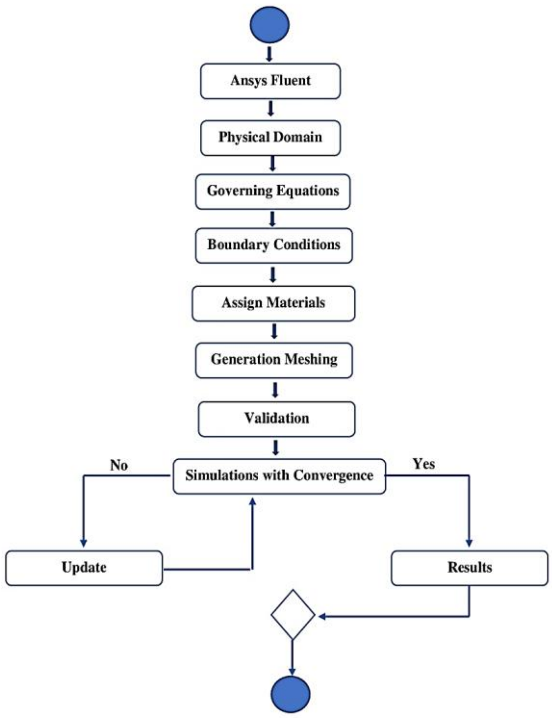

Figure 3. Numerical simulation flowchart of the present work

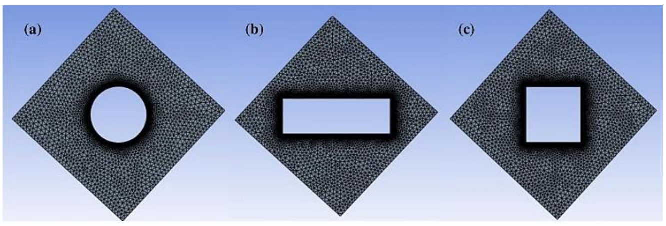

Figure 4. Creating the mesh for cylinder configuration inserted in the center of the enclosure: (a) circular, (b) rectangular, and (c) square

5. VALIDATION OF NUMERICAL MODEL

The present numerical results were verified by comparison with those of a published study [29], under almost the same geometric conditions.

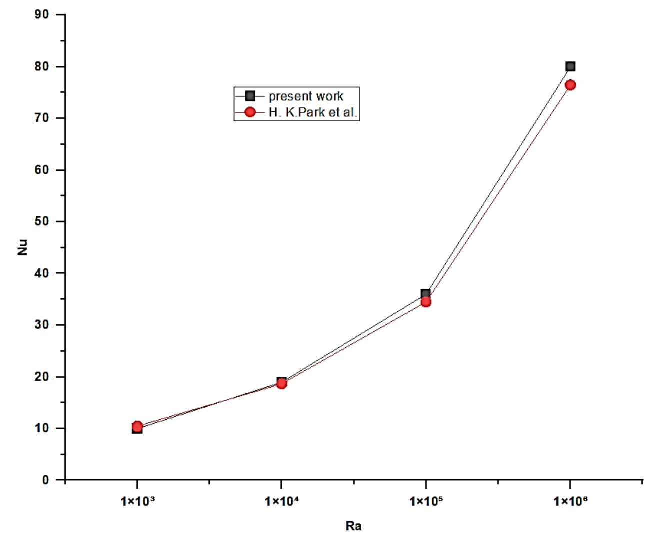

Figure 5. Validation of the numerical solution by comparison with the results of a published study

The study investigated natural convection within a 45° inclined square cavity containing a hot inner cylinder with a diameter of 0.15 m, cold side walls, and the remaining walls insulated. The fluid used was air at Pr ≈ 0.7 over the Rayleigh number range from 10^3 to 10^6. The comparison showed good agreement between the values, as the trend of change in the Nusselt number coincided with increasing Rayleigh number, and the relative differences were within acceptable limits (less than 7%). Figure 5 compares the values obtained from the present simulation with the published values, confirming the accuracy and validity of the numerical solution used in this study.

6. RESULTS AND DISCUSSION

Figure 6 indicates the velocity distribution of the fluid inside the inclined square enclosure passing around the different cylinders included inside, and for different ranges of Rayleigh numbers. The gradient of the maximum value of the speed can be observed located at the edge of the cold right wall, where the velocity gradient is the lowest value in the left area and increases to the right. It can also be observed that as the Rayleigh numbers increase, the agglomeration of areas of maximum speed fades, and the velocity begins to increase in other areas inside the enclosure, so it can be concluded that the rate of velocity of the fluid increases with the Rayleigh number.

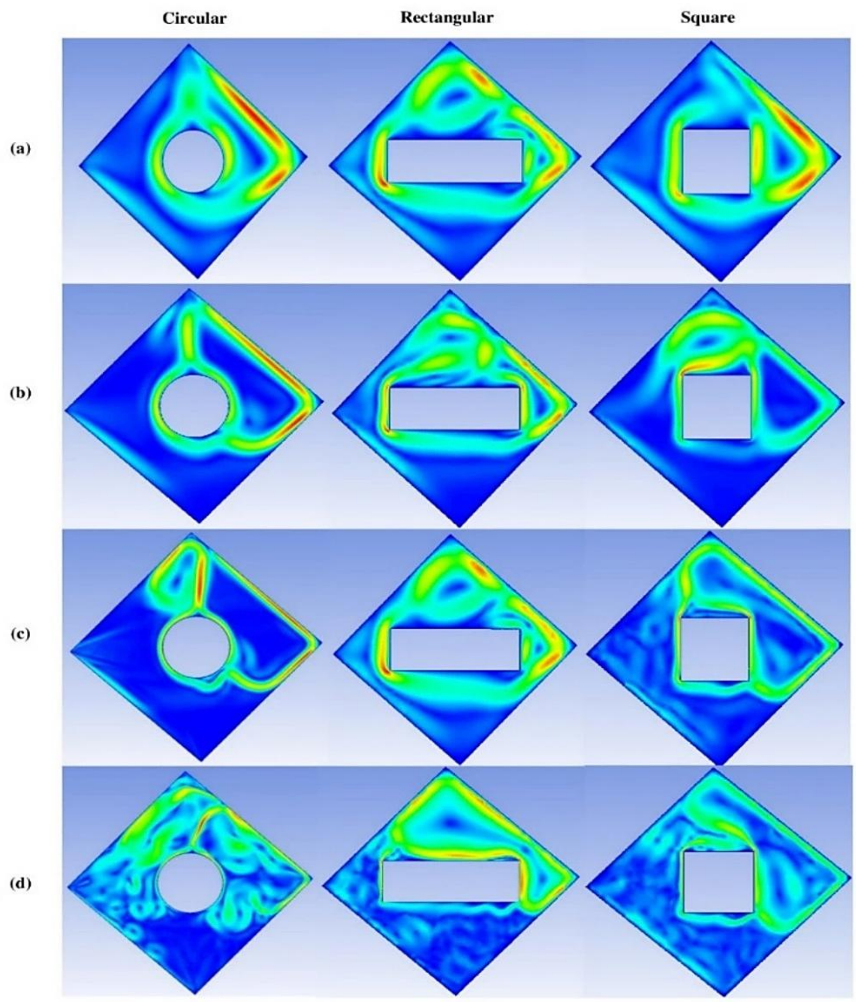

Figure 6. Distribution of velocity for different configuration cylinders at (a) Ra of 10^6, (b) Ra of 10^7, (c) Ra of 10^8, and (d) Ra of 10^9

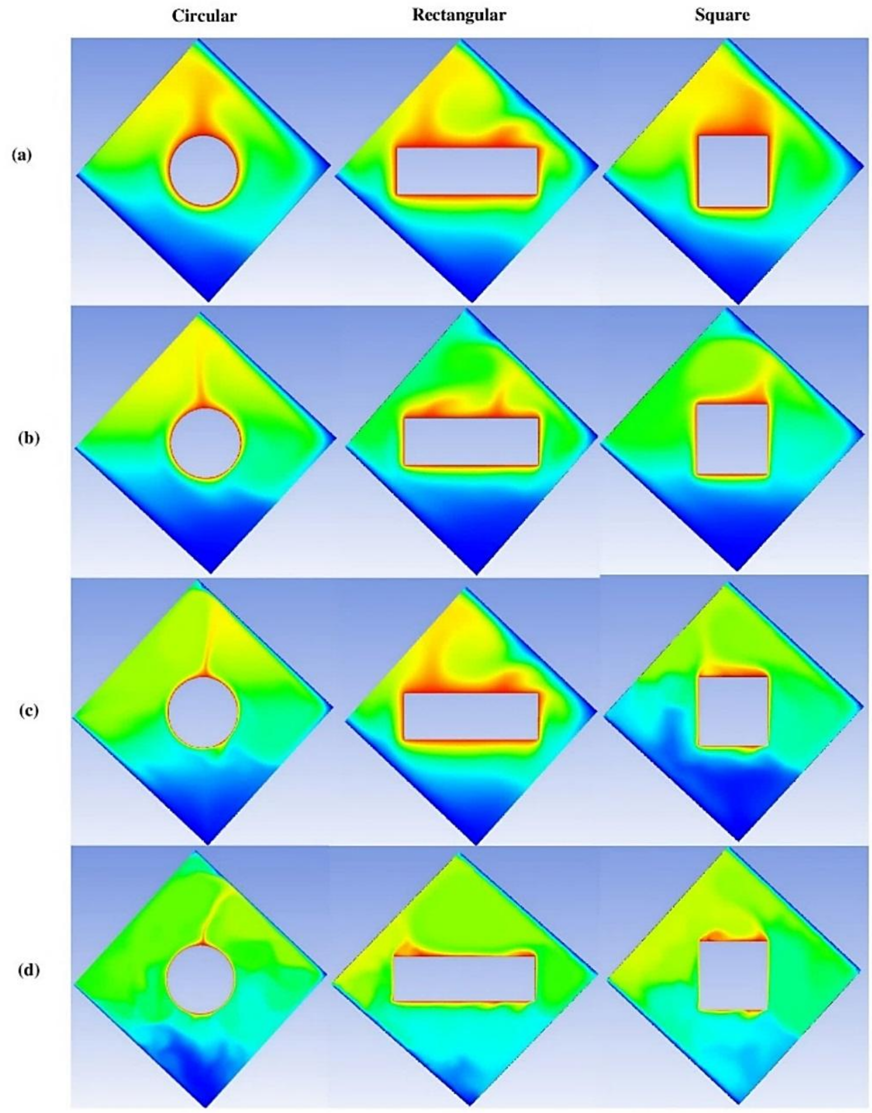

Figure 7. Distribution of temperature for different configuration cylinders at (a) Ra of 10^6, (b) Ra of 10^7, (c) Ra of 10^8, and (d) Ra of 10^9

Figure 7 indicates the temperature distribution of the normal load around cylinders of various configurations (circular, rectangular, and square) inserted inside the square enclosure inclined at an angle (45°) and for the ranges of different Rayleigh numbers. The temperature can be observed at the walls of the cylinder at the maximum value and begins to gradually decrease at the walls of the enclosure, where the temperature in the right and left Walls is low because they are exposed to a cold temperature, while the gradient occurred in the upper region of the enclosure with the observation of the temperature gradient with increasing Rayleigh numbers expands inside the enclosure and decreases around the perimeter of the cylinders. At low Rayleigh numbers, it can be observed that the square and rectangular configuration gives deformation to high temperatures and density due to sharp angles compared to the circular configuration, and the deformation decreases with increasing Rayleigh numbers. It is evident that when the Rayleigh number rises, heat transfer diffusion increases as well. As a result, the distorted and crowded temperature fields become closer to the half-down wall, demonstrating that conduction heat transfer disappears and behavior completely shifts to convection. Since buoyancy is directly proportional to the fluid's density gradients, or the temperature differential within the enclosure, this difference causes the phenomenon of buoyancy, which in turn causes heat transfer through free convection. Additionally, gravity causes the denser layer to descend to replace the hot layer.

Figure 8 describes the streamlines in the order of the shapes of the hot cylinder (circular, rectangular, and square) inside the inclined square enclosure (Ø = 45°) at different Rayleigh numbers (10^6-10^9). As a result of the temperature difference between the hot cylinder wall and the cold walls to the right and left, the buoyancy effect will appear, so vortices are generated inside the inclined enclosure. The working fluid moves from the beginning of the wall of the hot cylinder until it collides with the walls of the enclosure, which leads to a decrease in air temperature to moderate with a difference in density. By increasing the Rayleigh numbers, it was observed that the circular configuration generates more vortices compared to the others because the passage of air was not subjected to an obstacle that reduced the flow. Moreover, at low Rayleigh values, the convergence of the streamlines is very obvious, where the area with convergence has the air velocity, while the divergence of the lines indicates a decrease in velocity.

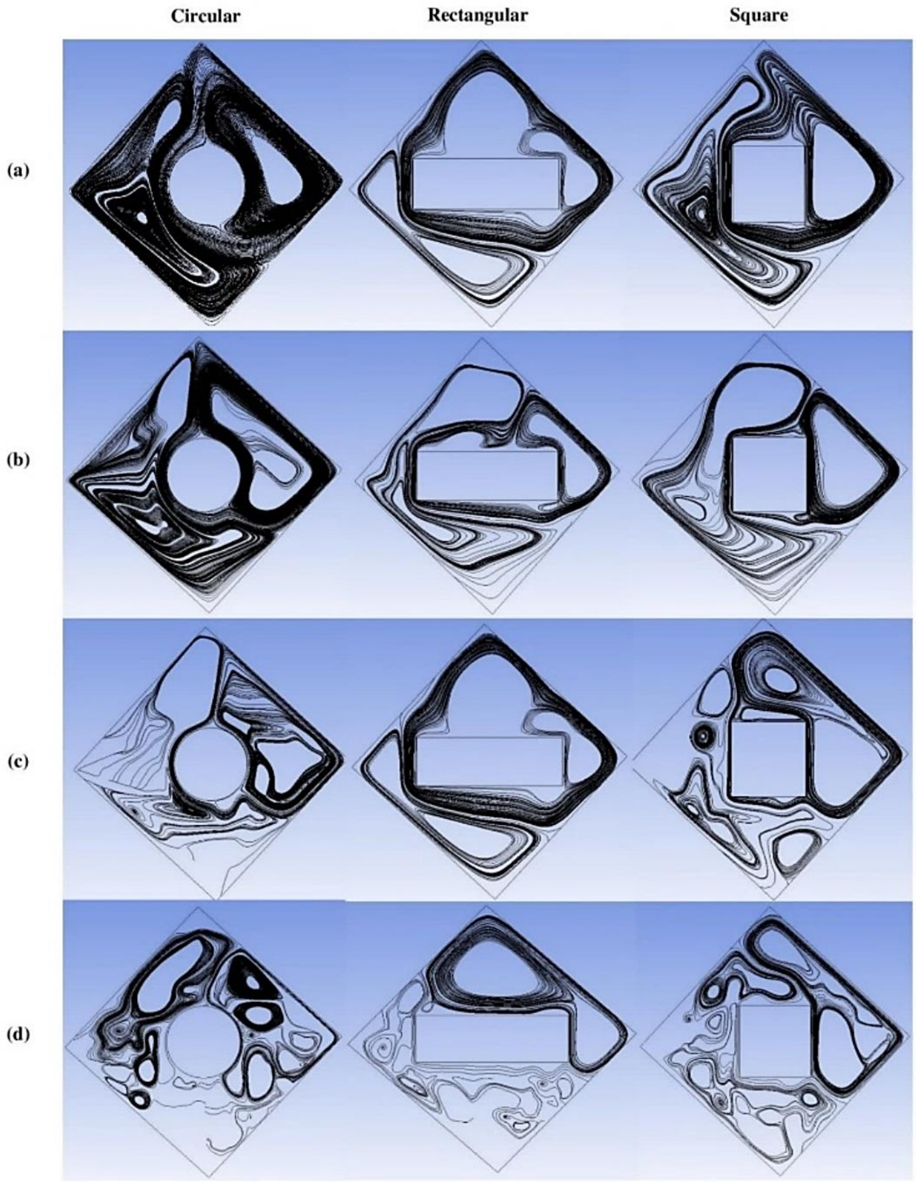

Figure 8. Distribution of Streamline for different configuration cylinders at (a) Ra of 10^6, (b) Ra of 10^7, (c) Ra of 10^8, and (d) Ra of 10^9

Vortexes are generated as a result of flow separation around an obstacle. The shape difference alters the pressure distribution and separation zones, which directly affects the velocity distribution pattern. In the case of a circular obstacle, the streamlines are more streamlined, producing relatively symmetrical vortices, whereas rectangular and square obstacles exhibit larger separation zones and sharp corners that concentrate shear stresses, increasing flow irregularity. These vortices interact with the thermal boundary layers on the interior walls of the cavity, enhancing heat mixing in areas where the vortices directly collide with the hot or cold walls, while weakening heat transfer in areas where low-speed recirculation zones form. This interaction between vortex dynamics and boundary layer development plays a crucial role in determining the efficiency of heat transfer within a system.

Figure 9 shows the change of Nusselt numbers with Rayleigh numbers within the laminar flow of heat transfer around cylinders exposed to a hot temperature of various configurations. It can be observed that the Nusselt numbers gradually increase with increasing the ranges of the Rayleigh numbers, whereas the difference of the Rayleigh numbers increases, and a jump occurs by increasing the Nusselt number. It can also be noted that the circular configuration of the cylinder has achieved maximum heat transfer compared to the square and rectangular configurations as a result of the convergence of the flow lines to each other and the increase in fluid velocity. The Rayleigh number (Ra) represents the driving force of natural convection, is a function of the temperature difference, the physical properties of the liquid, and geometric dimensions. The higher the Rayleigh number, the greater the intensity of convection, and therefore the Nusselt number (Nu) representing the rate of convective heat transfer increases. The circular cylinder shows the highest values of the Nusselt number at all Rayleigh number values, as it causes the least resistance to the air flow around it, which leads to enhanced natural convection as well as causes a more uniform and less separated flow of runoff, which enhances heat transfer. As for the square cylinder, its performance is lower than the circular one, but higher than the rectangular one. The presence of sharp corners creates stagnation zones and thermal pockets, which reduces the efficiency of heat exchange compared to the round shape. They still retain an acceptable amount of heat exchange because the exchange surface is approximated in different directions. The rectangular cylinder shows the lowest values of the Nusselt number. The rectangular shape increases the resistance to runoff and creates larger stagnation zones and areas with eddy currents (vortices), which reduces the efficiency of convection. The effect of tilting the inclined enclosure changes the distribution of buoyancy forces generated by the temperature difference. In inclined cavities, the direction of gravity controls the thermal flow pattern, which makes the effect of the internal body shape more prominent on the thermal transition. This change in the distribution of runoff affects differently each of the internal forms, depending on how the runoff interacts with it.

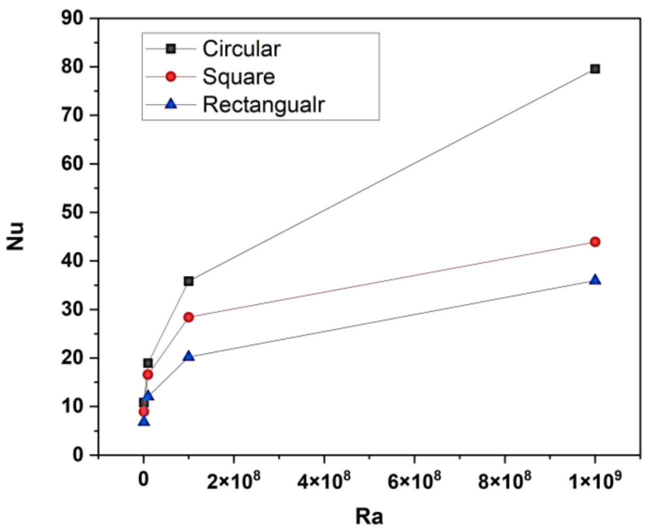

Figure 9. Variations of Nu against Ra for different configurations of the cylinder

7. CONCLUSIONS

Numerical study of heat transfer by the method of natural convection of a square enclosure inclined at an angle of (45°), two-dimensional laminar flow within the ranges of Rayleigh numbers (10^6-10^9) filled with air as a working fluid, various cylinder configurations (circular, rectangular, and square) with equal hydraulic diameter were listed. The upper and lower walls were assumed to be thermally insulated, while the right and left walls were exposed to a cold temperature, and the cylinder was exposed to a heating temperature, keeping the angle of inclination of the container constant for all cases at (45°). Therefore, the most prominent conclusions of the current numerical study can be summarized as follows:

1. The Nusselt numbers gradually increase with increasing the Rayleigh numbers, noting that the Nusselt number is affected by the configuration of the cylinder inserted inside the enclosure, where when changing the configuration, the values of the Nusselt number change.

2. The heat transfer of the circular configuration gave the best improvement compared to other configurations (rectangle and square).

3. Increasing the Rayleigh number affects the flow of air inside the inclined square enclosure, which leads to an increase in the rate of heat transfer due to an increase in the convection phenomenon.

4. The distribution of the working fluid velocity at the upper regions of the enclosure is higher compared to other regions. As an increase in Rayleigh numbers increase, the velocity gradient around the cylinder configurations is averaged, and areas with high speed begin to decrease.

5. The flow of the working fluid around the cylinder generates vortices, and by increasing the Rayleigh numbers, the number of vortices increases as a result of increasing the velocity of the air inside the enclosure.

6. As a result of the sharp ribs, the appearance of vortices in a cylinder with a square configuration is more compared to rectangular and circular.

7. The streamlines of the circular configuration are closer compared to other configurations (rectangle and Square). This convergence means that there are areas with high air velocity.

8. Buoyancy is directly proportional to the density gradients of the fluid, i.e., directly to the temperature difference of the enclosure, so because of this difference, the phenomenon of buoyancy occurs leading to heat transfer by free convection, moreover, as a result of the action of gravity, the denser layer descends to replace the hot layer.

9. The slope changes the distribution of buoyancy forces generated by the temperature difference. In inclined cavities, the direction of gravity controls the thermal flow pattern, which makes the effect of the internal body shape more prominent on the thermal transition.

Future developments of the present study can be proposed by studying the effect of different obstacle dimensions and angles, in addition to using complex geometries or variable fluid properties (such as nanofluids) to improve thermal mixing, as well as expanding the range of Reynolds and Rayleigh numbers to analyze more diverse flow and heat transfer patterns.

REFERENCES

[1] Liu, C., Bao, Y., Huang, K., Lyu, P., Liu, X., Rao, Z. (2025). Experimental study on natural convection heat transfer performance of microencapsulated phase change material slurry in a square cavity. Applied Thermal Engineering, 259: 124883. https://doi.org/10.1016/j.applthermaleng.2024.124883

[2] Suvanjumrat, C., Priyadumkol, J., Khaothong, K., Chaiworapuek, W. (2025). An empirical investigation into enhancing natural convection heat transfer through corona wind in a needle-to-cylinder configuration. Case Studies in Thermal Engineering, 68: 105864. https://doi.org/10.1016/j.csite.2025.105864

[3] Zemani, F., Sabeur, A., Ladjedel, O., Chelih, A. (2025). Natural convective heat transfer in a square cavity with a curved hot wall partially heated from below. Physics of Fluids, 37(1): 019101. https://doi.org/10.1063/5.0244480

[4] Njoroge, J., Gao, P. (2025). Natural convection heat transfer and intensification for a discrete heat source in a vertical annulus. European Journal of Mechanics-B/Fluids, 109: 170-179. https://doi.org/10.1016/j.euromechflu.2024.10.011

[5] Ranjan, K., Biswal, G., Rout, S.K., Hussein, A.K., Eladeb, A., Alshammari, B.M., Kolsi, L. (2025). Optimization of heat transfer in a solid torus: A parametric and numerical approach under natural convection. Case Studies in Thermal Engineering, 65: 105680. https://doi.org/10.1016/j.csite.2024.105680

[6] Lee, J.H., Goldstein, R.J. (1988). An experimental study on natural convection heat transfer in an inclined square enclosure containing internal energy sources. ASME Journal of Heat and Mass Transfer, 110(2): 345-349. https://doi.org/10.1115/1.3250490

[7] Alshuraiaan, B. (2023). Fluid structure interaction analysis of natural convective flow and heat transfer in an L-shaped cavity. Journal of Advanced Research in Fluid Mechanics and Thermal Sciences, 109(1): 1-12. https://doi.org/10.37934/arfmts.109.1.112

[8] Barik, A.M., Al-Farhany, K. (2019). Numerical investigation of the effect of baffle inclination angle on nanofluid natural convection heat transfer in a square enclosure. Al-Qadisiyah Journal for Engineering Sciences, 12(2): 61-71. http://doi.org/10.30772/qjes.v12i2.589

[9] Ternik, P., Rudolf, R. (2012). Heat transfer enhancement for natural convection flow of water-based nanofluids in a square enclosure. International Journal of Simulation Modelling, 11(1): 29-39. https://doi.org/10.2507/IJSIMM11(1)3.198

[10] Lugarini, A., Franco, A.T., Junqueira, S.L., Lage, J.L. (2018). Natural convection and surface radiation in a heated wall, C-shaped fracture. Journal of Heat Transfer, 140(8): 082501. https://doi.org/10.1115/1.4039643

[11] House, J.M., Beckermann, C., Smith, T.F. (1990). Effect of a centered conducting body on natural convection heat transfer in an enclosure. Numerical Heat Transfer, 18(2): 213-225. https://doi.org/10.1080/10407789008944791

[12] Zhao, F.Y., Liu, D., Tang, G.F. (2007). Application issues of the streamline, heatline and massline for conjugate heat and mass transfer. International Journal of Heat and Mass Transfer, 50(1-2): 320-334. https://doi.org/10.1016/j.ijheatmasstransfer.2006.06.026

[13] Bhave, P., Narasimhan, A., Rees, D.A.S. (2006). Natural convection heat transfer enhancement using adiabatic block: Optimal block size and Prandtl number effect. International Journal of Heat and Mass Transfer, 49(21-22): 3807-3818. https://doi.org/10.1016/j.ijheatmasstransfer.2006.04.017

[14] De, A.K., Dalal, A. (2006). A numerical study of natural convection around a square, horizontal, heated cylinder placed in an enclosure. International Journal of Heat and Mass Transfer, 49(23-24): 4608-4623. https://doi.org/10.1016/j.ijheatmasstransfer.2006.04.020

[15] Oh, J.Y., Ha, M.Y., Kim, K.C. (1997). Numerical study of heat transfer and flow of natural convection in an enclosure with a heat-generating conducting body. Numerical Heat Transfer, Part A Applications, 31(3): 289-303. https://doi.org/10.1080/10407789708914038

[16] Lee, J.R., Ha, M.Y. (2005). A numerical study of natural convection in a horizontal enclosure with a conducting body. International Journal of Heat and Mass Transfer, 48(16): 3308-3318. https://doi.org/10.1016/j.ijheatmasstransfer.2005.02.026

[17] Lee, J.R., Ha, M.Y. (2006). Numerical simulation of natural convection in a horizontal enclosure with a heat-generating conducting body. International Journal of Heat and Mass Transfer, 49(15-16): 2684-2702. https://doi.org/10.1016/j.ijheatmasstransfer.2006.01.010

[18] Zhao, F.Y., Tang, G.F., Liu, D. (2006). Conjugate natural convection in enclosures with external and internal heat sources. International Journal of Engineering Science, 44(3-4): 148-165. https://doi.org/10.1016/j.ijengsci.2005.10.006

[19] Miroshnichenko, I.V., Sheremet, M.A. (2018). Turbulent natural convection heat transfer in rectangular enclosures using experimental and numerical approaches: A review. Renewable and Sustainable Energy Reviews, 82: 40-59. https://doi.org/10.1016/j.rser.2017.09.005

[20] Corcione, M., Cianfrini, M., Quintino, A. (2015). Natural convection in square enclosures differentially heated at sides using alumina-water nanofluids with temperature-dependent physical properties. Thermal Science, 19: 591-608. http://doi.org/10.2298/TSCI120328111C

[21] An, B., Meng, X., Bergadà, J.M., Xi, C., Sang, W. (2025). Natural convection inside square and cubic cavity with left wall partially heated. Journal of Thermophysics and Heat Transfer, 1-18. https://doi.org/10.2514/1.T7100

[22] Gasiunas, S., Seporaitis, M. (2025). CFD and experimental insights into natural convection in symmetrically heated vertical cylinders for renewable energy accumulation. Renewable Energy, 241: 122313. https://doi.org/10.1016/j.renene.2024.122313

[23] Sethy, P., Kumar, A., Ray, A.K., Chaurasiya, V.K., Kumari, A., Sheremet, M.A. (2025). Heat transfer performance in a hybrid nanoliquid enclosure with a hot inner circular obstacle under Lorentz force effect. Journal of Thermal Analysis and Calorimetry, 150(1): 395-412. https://doi.org/10.1007/s10973-024-13750-z

[24] Shihab, S.H., Lafta, A., Alsaedi, H.M. (2025). Investigation of natural convection in vertical convergent plates: A review. AIP Conference Proceedings, 3169(1): 040030. https://doi.org/10.1063/5.0254270

[25] Seo, Y.M., Park, Y.G. (2025). Three-dimensional natural convection in an enclosure with four cylinders: Effects of vertical spacing on flow and heat transfer. Case Studies in Thermal Engineering, 74: 106763. https://doi.org/10.1016/j.csite.2025.106763

[26] Ali, S.A. (2024). Numerical investigation of effect partially corrugated heated wall on behaviour natural convection heat transfer in a square enclosure. Al-Rafidain Journal of Engineering Sciences, 2(2): 374-382. https://doi.org/10.61268/zj5tvt41

[27] Bawazeer, S.A. (2025). Laminar natural convection in a square cavity with a horizontal fin on the heated wall: A numerical study of fin position and thermal conductivity effects. Energies, 18(13): 3335. https://doi.org/10.3390/en18133335

[28] Abdulsahib, A.D., Alkhafaji, D., Albayati, I.M. (2025). Improving the natural convection of the heat sink inside an enclosure by fin perforation for electronic applications. Heat Transfer, 54(1): 389-419. https://doi.org/10.1002/htj.23181

[29] Park, H.K., Ha, M.Y., Yoon, H.S., Park, Y.G., Son, C. (2013). A numerical study on natural convection in an inclined square enclosure with a circular cylinder. International Journal of Heat and Mass Transfer, 66: 295-314. https://doi.org/10.1016/j.ijheatmasstransfer.2013.07.029

NOMENCLATURE

Ansys	Analysis System
AR	Aspect Ratio (H/L)
Cp	Heat capacity of the working fluid at constant pressure, J/kg. K

D	Diameter of cylinder, m
g	Acceleration due to gravity, m/s^2
Gr	Grashof Number
h	Coefficient of heat transfer, W/m^2. K
H	Total height of enclosure, m
k	Working fluid thermal conductivity, W/m. K
L	The total length of the enclosure, m
Nu	Nusselt number
p	Working fluid pressure, N/m^2
Pr	Prandtl number
q''	Heat flux, W/m^2
Ra	Rayleigh number
T	Working fluid temperature, K
u and v	Working fluid velocity in x and y direction, respectively, m/s
x and y	Cartesian coordained

Greek symbols

μ	Working fluid dynamics viscosity, N.s $/m^2$
Ø	The inclined angle of the square enclosure, degree
α	Working fluid thermal diffusivity, m^2/s
β	Volume expansion coefficient, 1/K
υ	Working fluid kinematic viscosity, m^2/s

Subscripts

c	Cold
ca	Characteristics
h	Hot
H	Hydraulic