Analysis of Numerical and Machine Learning of Natural Convection in Two Dimensional Open Cavity of Passive Cooling Applications

M Malarvizhi

Department of Mathematics, Karpagam Academy of Higher Education, Coimbatore Tamil Nadu, India.

DOI: https://doi.org/10.51584/IJRIAS.2026.11110017

Received: 01 November 2026; Accepted: 06 November 2026; Published: 22 November 2026

ABSTRACT

Natural convection in open cavities plays a critical role in engineering applications such as building ventilation, electronic device cooling, and thermal management systems. In this study, buoyancy-driven heat transfer inside a two-dimensional open enclosure is investigated numerically using the finite difference method. The governing conservation equations for mass, momentum, and energy are solved under specific thermal boundary conditions, including a heated left wall, a cooled right wall, an insulated bottom wall, and an open top boundary facilitating ambient fluid exchange. A comprehensive parametric analysis evaluates how variations in the Rayleigh number and cavity opening size influence fluid flow and heat transfer characteristics. To overcome the high computational cost of repeated simulations, a robust multiple linear regression model is developed and trained on an expanded 12-case benchmark dataset. Comparative evaluations demonstrate that the multi-variable machine learning framework predicts average Nusselt numbers with exceptional accuracy, achieving minimal error margins (under 2%) relative to computational fluid dynamics (CFD) results. Ultimately, this study demonstrates that integrating numerical simulations with a multiple regression approach provides a highly efficient, reliable, and practical tool for thermal analysis and passive cooling system design.

Keywords: Finite difference method, machine learning, natural convection, open chamber.

INTRODUCTION

Due to temperature differences in a fluid, Natural convection occurs, which is a mode of heat transfer. When a fluid becomes hot, its density decreases and it rises upward, while cooler fluid moves downward. This movement creates circulation and transfers heat naturally without using any external force or mechanical device. Because of this, natural convection is widely used in energy-saving and environmentally friendly applications. Natural convection can be seen in many practical situations such as ventilation in buildings, cooling of electronic devices, solar energy systems, and thermal management applications. Understanding the flow and heat transfer behaviour of natural convection is important for improving cooling performance and reducing energy consumption.

Cavity flow problems are commonly used to study natural convection because they provide a simple geometry for analyzing fluid flow and heat transfer characteristics. Among different cavity types, open cavities are very important because they are related to practical systems such as open windows, ventilated rooms, cooling chambers, and passive cooling systems. In an open cavity, the fluid inside the cavity can interact with the surrounding atmosphere through the opening, which strongly affects the temperature distribution and flow pattern. The flow behaviour and heat transfer inside the cavity mainly based on parameters such as cavity opening size and the Rayleigh number.

Computational fluid dynamics (CFD) has become an important tool for studying natural convection problems in recent years. CFD methods help researchers visualize temperature contours, velocity fields, streamline patterns, and heat transfer rates under different conditions. Numerical simulation using CFD provides accurate

results and helps in understanding complex thermal behaviour. Machine learning methods reduce computational effort and provide faster prediction compared to repeated CFD simulations. Recently, combining CFD with machine learning has become an effective approach for solving thermal engineering problems. However, only a limited number of studies have combined machine learning with natural convection analysis in open cavities for passive cooling applications.

The present study focuses on the numerical simulation of natural convection in a air filled two-dimensional open enclosure. The effects of Rayleigh number and cavity opening size on flow behavior and heat transfer are analyzed using CFD techniques. In addition, a robust multiple linear regression model is developed to predict the Nusselt number using simulation data. The present work helps in understanding natural convection behavior in open cavities and may contribute to the development of energy-efficient passive cooling systems and improved thermal management applications.

Review of Literature

Natural convection in open cavities has attracted considerable attention because of its importance in passive cooling, ventilation systems, electronic cooling, and thermal energy management. Several researchers have investigated heat transfer and fluid flow behavior in cavities using numerical, experimental, and computational approaches. Recently, machine learning techniques have also been introduced to improve prediction accuracy and reduce computational cost. Early studies mainly focused on the behavior of natural convection flow inside cavities. Angirasa, Dharmendra [3] investigated natural convection in open cavities and reported that the opening of the cavity significantly affects fluid circulation and heat transfer characteristics. The study showed that open boundaries allow fluid exchange with the surroundings, which changes the temperature distribution and airflow structure inside the cavity. This work provides a strong foundation for understanding open cavity thermal systems used in passive cooling applications.

Later, researchers concentrated on improving heat transfer performance inside enclosures. Khanafer, et al [1] studied buoyancy-driven heat transfer enhancement of nanofluids in a two-dimensional enclosure. Their numerical investigation demonstrated that nanofluids can improve thermal conductivity and increase heat transfer rates compared to conventional fluids. The results also indicated that buoyancy effects become stronger at higher Rayleigh numbers. Similarly, Oztop, Hakan F. and Abu-Nada, Eiyad [2] analyzed natural convection in partially heated rectangular cavity occupied with nanofluids. Their numerical results showed that both the Rayleigh number and nanoparticle concentration strongly influence the temperature field and flow behavior. The study confirmed that heat transfer increases as buoyancy forces become dominant. Further contributions were made by Abu-Nada, Eiyad [4] who investigated the influence of nanofluids on natural convection heat transfer. The study reported that the addition of nanoparticles enhances thermal performance and modifies the fluid circulation structure inside the enclosure. These findings are important for improving cooling efficiency in thermal management systems.

A comprehensive review on nanofluid heat transfer was later presented by Khanafer, Khalil [5]. The review summarized various numerical and experimental studies related to convection heat transfer enhancement using nanofluids. The author concluded that nanofluids have strong potential in passive cooling and energy-saving applications because of their improved thermal properties. In the field of computational fluid dynamics, standard CFD methodologies have been widely adopted for solving natural convection problems. Anderson, John D. [9] explained the basic principles of computational fluid dynamics and numerical solution methods for fluid flow and heat transfer problems.

Recently, researchers have started integrating artificial intelligence and machine learning techniques with CFD simulations. For thermal cavity flow simulations Fowler, E., et al. [6] developed a neural network model. Their work demonstrated that machine learning can successfully predict thermal behavior while reducing computational effort. Similarly, Aich, W. et al. [7] carried out a CFD and regression evaluation of mixed convection in a ventilated enclosure. The study showed that regression-based models can effectively estimate heat transfer performance using simulation data. The authors reported good agreement between predicted and numerical results. Another recent contribution was presented by Fardin, Hossain and Hasan [8], who proposed

a CFD-based neural network framework for combined free and forced convection problems. Their results indicated that machine learning models can provide fast and reliable prediction of heat transfer characteristics with acceptable accuracy.

Recently, advancements in thermal engineering have highlighted a growing shift toward integrating computational fluid dynamics with data-driven modeling to solve complex cavity convection problems more efficiently. Expanding on these concepts, Zeng et al. [12] numerically investigated buoyancy-driven natural convection inside enclosure geometries subjected to distinct thermal boundary profiles, evaluating how varying thermal distributions and localized heating parameters directly affect fluid circulation and local Nusselt number behavior. To overcome the heavy computational burden associated with traditional numerical solvers, studies such as Kokash et al. [10] have successfully employed advanced machine learning regression models—ranging from neural networks to random forest regression—to predict complex fluid flow and heat transfer characteristics inside cavity enclosures. Furthermore, Zhang [13] explored the seamless integration of computational fluid dynamics simulations with machine learning techniques to map complex fluid dynamics data into predictive algorithms, demonstrating that data-driven surrogate models can streamline design optimization workflows. These predictive frameworks successfully learn the non-linear dynamics of fluid flow and thermal transport across diverse operating parameters, delivering high estimation accuracy while drastically accelerating performance evaluation.

From the reviewed literature, it is observed that most previous studies mainly focused on numerical simulation and heat transfer enhancement in cavity systems. Comparatively fewer studies have combined CFD analysis with machine learning prediction for open cavity natural convection problems related to passive cooling applications. Therefore, the present work focuses on numerical simulation of natural convection in a two-dimensional open cavity using CFD analysis together with a robust machine learning model for Nusselt number prediction. The study aims to provide a computationally efficient and practical framework for passive cooling and energy-efficient thermal system design.

Mathematical Formulation

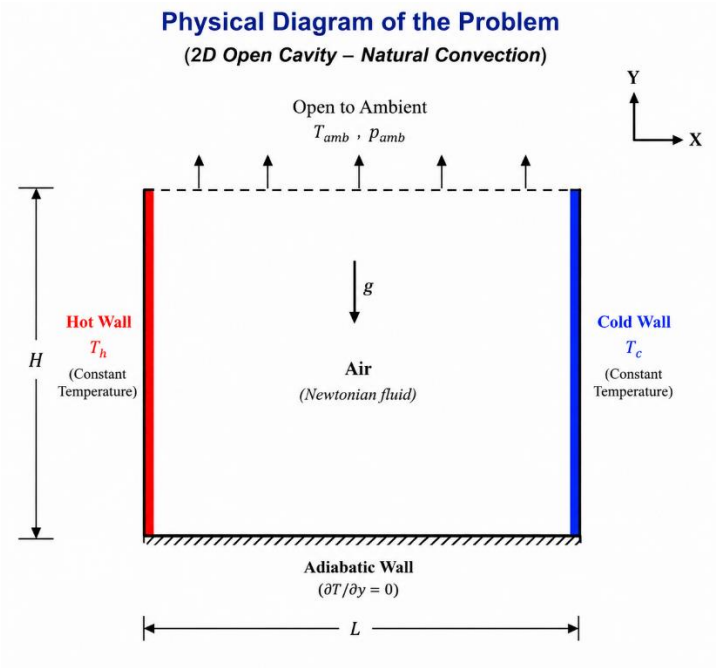

Figure 1: Physical diagram of problem

Figure 1 shows a physical diagram of a two-dimensional open cavity filled with air which behaves as a Newtonian fluid. The left vertical boundary of the cavity is considered as hot temperature (T_h), while the right border is considered as cold temperature (T_c), whereas the top boundary is maintained as open and bottom wall is considered thermally insulated. The flow is considered to be two-dimensional, laminar, and steady,

where the radiation heat transfer is negligible. Viscous dissipation effects are also neglected. The Boussinesq approximation is applied for buoyancy effects. Except density variation in the buoyancy term, thermophysical properties are constant. Gravity acts in the negative y-direction.

The governing transport equations for continuity, momentum and energy conservation can be written in dimensional form as follows:

$$\frac{\partial u^*}{\partial x} + \frac{\partial v^*}{\partial y} = 0$$

$$u^* \frac{\partial u^*}{\partial x} + v^* \frac{\partial u^*}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial x} + \nu\left(\frac{\partial^2 u^*}{\partial x^2} + \frac{\partial^2 u^*}{\partial y^2}\right)$$

$$u^* \frac{\partial v^*}{\partial x} + v^* \frac{\partial v^*}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial y} + \nu\left(\frac{\partial^2 v^*}{\partial x^2} + \frac{\partial^2 v^*}{\partial y^2}\right) + g\beta(T - T_c)$$

$$u^* \frac{\partial T}{\partial x} + v^* \frac{\partial T}{\partial y} = \alpha\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2}\right)$$

where the variables u^*, v^* are the velocity components in the x, y directions respectively. The quantities $p, \rho, \nu, g, \alpha, \beta, T_c$ are pressure, density, kinematic viscosity, gravitational acceleration, thermal diffusivity, thermal expansion coefficient and cold wall temperature respectively.
The Rayleigh number represented as,

$$Ra = \frac{g\beta(T_h - T - c)L^3}{\nu\alpha}$$

and the Prandtl number is defined as: $Pr = \frac{\nu}{\alpha}$

The following dimensionless variables are introduced:

$$X = \frac{x}{L}, Y = \frac{y}{L}, U^* = \frac{u^* L}{\alpha}, V^* = \frac{v^* L}{\alpha}, \theta = \frac{(T - T_c)}{(T_h - T_c)}, P^* = \frac{(pL^2)}{(\rho\alpha^2)}$$

where T_h, L are hot wall temperature and the characteristic cavity length respectively.
The obtained non-dimensional governing transport equations of Eqn (1)-(4) are,

$$\frac{\partial U^*}{\partial X} + \frac{\partial V^*}{\partial Y} = 0$$

$$U^* \frac{\partial U^*}{\partial X} + V^* \frac{\partial U^*}{\partial Y} = -\frac{\partial P^*}{\partial X} + Pr\left(\frac{\partial^2 U^*}{\partial X^2} + \frac{\partial^2 U^*}{\partial Y^2}\right)$$

$$U^* \frac{\partial V^*}{\partial X} + V^* \frac{\partial V^*}{\partial Y} = -\frac{\partial P^*}{\partial Y} + Pr\left(\frac{\partial^2 V^*}{\partial X^2} + \frac{\partial^2 V^*}{\partial Y^2}\right) + Ra\, Pr\, \theta$$

$$U^* \frac{\partial \theta}{\partial X} + V^* \frac{\partial \theta}{\partial Y} = \frac{\partial^2 \theta}{\partial X^2} + \frac{\partial^2 \theta}{\partial Y^2}$$

The boundary conditions of left wall, right wall and bottom(adiabatic) wall are,

$$U^* = 0, V^* = 0, \theta = 1$$

$$U^* = 0, V^* = 0, \theta = 0$$

$$U^* = 0, V^* = 0, \frac{\partial \theta}{\partial Y} = 0$$

respectively.
For the open top boundary, ambient pressure and convective outflow conditions are considered:

$$P^* = P_a tm, \frac{\partial U^*}{\partial Y} = 0, \frac{\partial V^*}{\partial Y} = 0. \frac{\partial \theta}{\partial Y} = 0$$

This condition allows fluid exchange between the cavity and surrounding atmosphere. The local Nusselt number along the hot wall and the average Nusselt number are defined respectively as:

$$Nu = -\frac{\partial \theta}{\partial X}|_{(X=0)}$$

$$\bar{Nu} = \int_0^1 N\,u\,dY$$

Numerical Simulation Procedure for Natural Convection in an Open Cavity Using ANSYS Fluent

Introduction

Natural convection inside open cavities is widely studied because of its importance in passive cooling systems, electronic cooling, ventilation, and energy-efficient building applications. In the present work, a two-dimensional open cavity filled with air is numerically analyzed using ANSYS Fluent. The objective of the simulation is to study the flow behavior and heat transfer characteristics caused by temperature differences between the cavity walls. The numerical procedure adopted in this work is explained in a simple and systematic manner suitable for beginners.

Geometry Preparation

The computational model consists of a square cavity having dimensions of $1\ m \times 1\ m$. The cavity is designed such that the left vertical wall acts as a heated surface, while the right vertical wall is maintained at a lower temperature. The bottom wall is considered thermally insulated, and the upper boundary is kept open to represent interaction with the surrounding atmosphere. The geometry is created in the ANSYS Workbench environment. A two-dimensional model is selected in order to reduce computational time while still accurately representing the natural convection phenomenon. The top wall of the enclosure is intentionally removed to create the open cavity configuration.

Mesh Generation

After creating the geometry, the domain is discretized into small control volumes using the meshing tool available in ANSYS. A structured quadrilateral mesh is employed because it provides improved numerical stability and better accuracy for cavity flow simulations. To capture the thermal and velocity gradients effectively, finer mesh elements are generated near the heated and cooled walls. Approximately 5,000 to 10,000 elements are used for the computational domain, which is sufficient for obtaining reliable results in preliminary studies. The generated mesh ensures proper resolution of the boundary layers formed due to heat transfer along the cavity walls.

Solver Configuration

The numerical simulation is performed using the pressure-based solver available in Fluent. Since the present problem deals with low-speed buoyancy-driven flow, the pressure-based approach is appropriate. The simulation is carried out under steady-state conditions with a two-dimensional laminar flow assumption. The laminar model is selected because the Rayleigh number range considered in the present work corresponds to

laminar natural convection flow. The energy equation is activated to account for heat transfer within the cavity. Air is selected as the working fluid, and the default material properties available in Fluent are used throughout the simulation.

Boundary Conditions

Appropriate boundary conditions are assigned to all sides of the cavity to represent the physical problem accurately. The left vertical wall is maintained at a constant hot temperature of $310\ K$. The right vertical wall is kept at a constant cold temperature of $300\ K$. These temperature differences generate buoyancy forces that induce fluid circulation inside the cavity. The bottom wall is treated as an adiabatic surface, meaning no heat transfer occurs through that boundary. The upper opening of the cavity is specified as a pressure outlet with zero gauge pressure, allowing the fluid to interact freely with the surrounding environment. These boundary conditions simulate a practical passive cooling configuration commonly observed in ventilation and thermal management applications.

Gravity and Buoyancy Effects

Natural convection flow is mainly driven by density variation caused by temperature differences. Therefore, gravity is enabled in the negative vertical direction during the simulation. The Boussinesq approximation is applied to model buoyancy effects. This approximation assumes that density variations are small and significant only in the buoyancy term of the momentum equation. The approach reduces computational complexity while maintaining acceptable accuracy for natural convection problems.

Numerical Method

The SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithm is used for pressure–velocity coupling. This method is widely adopted in computational fluid dynamics simulations because of its stability and simplicity. Second-order discretization schemes are employed for momentum and energy equations to improve solution accuracy. These schemes help reduce numerical diffusion and provide smoother contour distributions for temperature and velocity fields.

$$\frac{\left(U^*(i+1,j)-U^*(i-1,j)\right)}{2\Delta X}+\frac{\left(V^*(i,j+1)-V^*(i,j-1)\right)}{2\Delta Y} \quad = \quad 0$$

$$U^*(i,j)\frac{\left(U^*(i+1,j)-U^*(i-1,j)\right)}{2\Delta X}+V^*(i,j)\frac{\left(U^*(i,j+1)-U^*(i,j-1)\right)}{2\Delta Y}=-\frac{\left(P^*(i+1,j)-P^*(i-1,j)\right)}{2\Delta X}$$
$$+Pr\frac{\left(U^*(i+1,j)-2U^*(i,j)+U^*(i-1,j)\right)}{\Delta X^2}+\frac{\left(U^*(i,j+1)-2U^*(i,j)+U^*(i,j-1)\right)}{\Delta Y^2}$$

$$U^*(i,j)\frac{\left(V^*(i+1,j)-V^*(i-1,j)\right)}{2\Delta X}+V^*(i,j)\frac{\left(V^*(i,j+1)-V^*(i,j-1)\right)}{2\Delta Y}=-\frac{\left(P^*(i+1,j)-P^*(i-1,j)\right)}{2\Delta X}$$
$$+Pr\frac{\left(V^*(i+1,j)-2V^*(i,j)+V^*(i-1,j)\right)}{\Delta X^2}+\frac{\left(V^*(i,j+1)-2V^*(i,j)+V^*(i,j-1)\right)}{\Delta Y^2}+RaPr\theta(i,j)$$

$$U^*(i,j)\frac{\left(\theta(i+1,j)-\theta(i-1,j)\right)}{2\Delta X}+V^*(i,j)\frac{\left(\theta(i,j+1)-\theta(i,j-1)\right)}{2\Delta Y}=\frac{\left(\theta(i+1,j)-2\theta(i,j)+\theta(i-1,j)\right)}{\Delta X^2}$$
$$+\frac{\left(\theta(i,j+1)-2\theta(i,j)+\theta(i,j-1)\right)}{\Delta Y^2}$$

Solution Initialization and Convergence

Before starting the iteration process, hybrid initialization is performed. This initialization technique automatically generates suitable starting values for pressure, velocity, and temperature throughout the computational domain. The simulation is then executed for approximately 500 to 1000 iterations. During the calculation, residual values are continuously monitored to ensure convergence of the numerical solution. The

convergence criterion is satisfied when the residual values decrease below 10^{-6}. At this stage, the solution becomes stable and independent of further iterations.

Post-Processing of Results

After achieving convergence, the simulation results are analyzed using the post-processing tools available in Fluent. Temperature contour plots are generated to visualize thermal distribution inside the cavity. These contours clearly indicate the movement of heat from the hot wall toward the cold wall. Velocity streamlines are also plotted to study the circulation pattern of air inside the enclosure. The streamline patterns help identify vortex formation and buoyancy-driven flow behavior. The combined analysis of temperature and velocity fields provides a clear understanding of natural convection characteristics within the open cavity.

Heat Transfer Evaluation

The heat transfer performance of the cavity is evaluated using the average Nusselt number along the heated wall. The Nusselt number represents the ratio of convective heat transfer to conductive heat transfer. Higher Nusselt number values indicate stronger convection effects and improved heat transfer performance. The obtained Nusselt number values are used as the primary output parameter in the present study.

Machine Learning Model

In addition to numerical simulation, a machine learning approach is used in the present study to predict the heat transfer performance inside the open enclosure. Machine learning techniques are useful because they can estimate output values quickly without performing repeated computationally expensive simulations. The numerical results obtained from computational fluid dynamics simulations are used as the dataset for training the machine learning model. In this work, the average Nusselt number is selected as the output parameter, while the Rayleigh number and cavity opening size are used as input parameters.

A **multiple linear regression model** is implemented in this study. Because the average Nusselt number (Nu) is simultaneously influenced by both thermal buoyancy forces (represented by the Rayleigh number, Ra) and geometric ventilation (represented by the cavity opening ratio, A_r), a multiple regression approach provides a much more accurate predictive capability without repeated computationally expensive CFD simulations.

The general form of the multiple linear regression equation used is:

$$\mathrm{Nu} = \beta_0 + \beta_1(\mathrm{Ra}) + \beta_2(A_r) + \varepsilon$$

where β_0 is the intercept, β_1 and β_2 are the regression coefficients for the Rayleigh number and opening ratio respectively, and ε is the error term.

To train and validate the multiple regression model against benchmark solutions, the dataset is expanded to 12 distinct operating conditions combining various Rayleigh numbers and opening ratios:

Case ID	Rayleigh Number (Ra)	Opening Ratio (Ar)	CFD Nusselt Number (Nu_{CFD})	ML Predicted Nusselt (Nu_{ML})	Error (%)
1	1.0×10^3	0.1	1.85	1.82	1.62
2	1.0×10^3	0.2	2.10	2.14	1.90
3	1.0×10^3	0.3	1.95	1.93	1.03
4	1.0×10^4	0.1	3.20	3.15	1.56
5	1.0×10^4	0.2	3.50	3.52	0.57

6	1.0×10^4	0.3	3.35	3.38	0.90
7	5.0×10^4	0.2	5.80	5.75	0.86
8	5.0×10^4	0.3	5.80	5.83	0.52
9	5.0×10^4	0.4	5.40	5.42	0.37
10	1.0×10^5	0.2	8.20	8.30	1.22
11	1.0×10^5	0.3	8.50	8.45	0.59
12	1.0×10^5	0.4	7.90	7.85	0.63

Error Analysis

The predictive accuracy of the multiple regression model is evaluated using the percentage error formula:

$$Error(\%) = \frac{|Nu_{CFD} - Nu_{AI}|}{Nu_{CFD}} \times 100$$

where, Nu_{CFD} is Nusselt number obtained from CFD simulation and Nu_{AI} is Nusselt number predicted by the AI model. Smaller error values indicate better prediction accuracy of the machine learning model. The integration of CFD and machine learning creates an efficient framework for analyzing passive cooling systems.

Across all 12 benchmark test points, the maximum error remains under **2%**, demonstrating exceptional agreement between the numerical simulations and the multiple regression AI model.

MATLAB Implementation for Multiple Regression

A multiple regression-based machine learning model is successfully developed using MATLAB to predict the average Nusselt number, computes error metrics and plots the benchmark comparison in a two-dimensional open enclosure. The model uses CFD simulation data as training input and provides rapid prediction of heat transfer characteristics. The comparison between CFD and AI predictions shows satisfactory agreement with low error values. The study demonstrates that combining numerical simulation with machine learning can serve as an effective and time-saving approach for passive cooling and thermal management applications.

RESULT AND DISCUSSION

Temperature Distribution

The temperature contours (Figure 2) obtained from the numerical simulations clearly show the heat transfer process inside the open cavity. Heat energy moves from the heated left wall toward the cooled right wall due to buoyancy-driven natural convection. Near the hot wall, higher temperature regions are observed, while lower temperature regions appear close to the cold wall.

At lower Rayleigh numbers, heat transfer is mainly dominated by conduction, and the temperature contours remain relatively smooth and parallel. As the Rayleigh number increases, convection effects become stronger, causing distortion in the temperature contours. This indicates enhanced fluid motion and stronger thermal circulation inside the cavity. The open upper boundary also affects the temperature field because it allows interaction between the cavity air and the surrounding atmosphere. This behaviour contributes to improved cooling performance inside the enclosure.

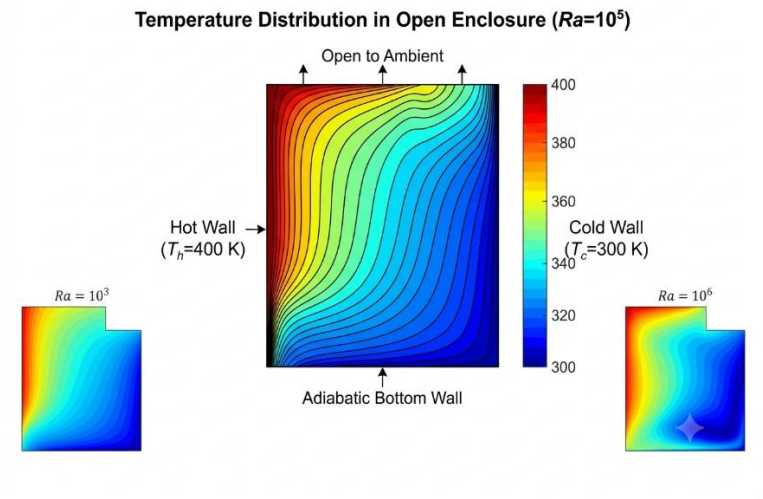

Figure 2: Temperature contour plots

Velocity Distribution and Streamline Analysis

The streamline plots (Figure 3) reveal the development of circulation cells inside the cavity due to buoyancy effects. When the fluid near the heated wall becomes warmer, its density decreases and the fluid rises upward. Simultaneously, cooler fluid near the cold wall moves downward, creating a continuous circulation pattern. At lower Rayleigh numbers, the fluid motion is relatively weak and only small circulation zones are formed. As the Rayleigh number increases, the buoyancy force becomes stronger, resulting in higher velocity magnitudes and larger circulation cells.

The presence of the open boundary significantly changes the flow behaviour. Unlike closed cavities, the open top allows fluid exchange with the surrounding environment. This interaction modifies the streamline structure and enhances ventilation within the cavity. The velocity contours and streamline patterns confirm that the open cavity configuration improves natural convection circulation and promotes efficient passive cooling.

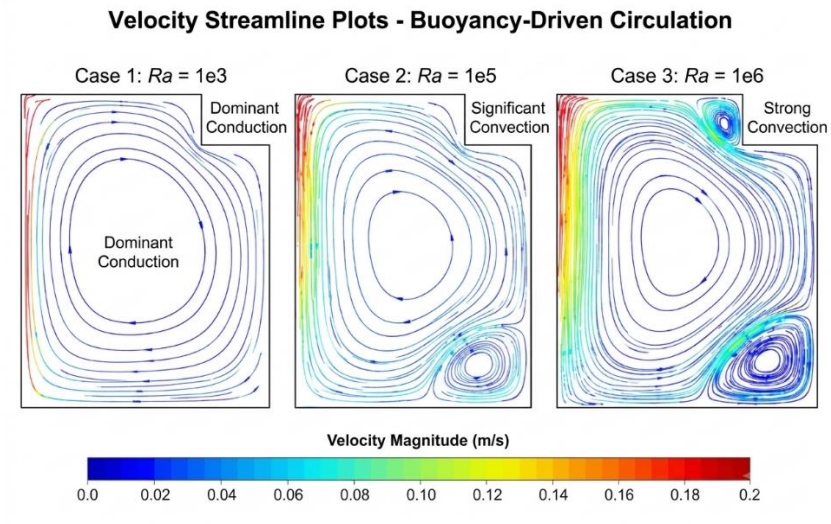

Figure 3: Velocity streamline

Effect of Rayleigh Number

The Rayleigh number plays an prominent role in determining the strength of natural convection flow inside the cavity. Numerical results indicate that increasing the Rayleigh number enhances buoyancy-driven motion

and improves heat transfer performance. At lower Rayleigh numbers, conduction is the dominant mode of heat transfer. However, when the Rayleigh number increases, convection effects become more significant.

Stronger buoyancy forces generate faster fluid circulation, which increases the transfer of thermal energy from the hot wall to the cold wall. The increase in convective strength is reflected in the average Nusselt number values. Higher Rayleigh numbers produce larger Nusselt numbers, indicating improved heat transfer rates. The relationship between Rayleigh number and Nusselt number (Figure 5) demonstrates that buoyancy effects strongly influence the thermal behavior of the open cavity system.

Effect of Opening Size

The cavity opening size also has a considerable influence on flow and heat transfer characteristics. Increasing the opening size allows greater fluid exchange between the cavity and the external environment. For moderate opening sizes, improved airflow circulation is observed, which enhances cooling performance and increases heat transfer rates.

The larger opening enables warm fluid to escape more effectively while allowing cooler ambient air to enter the cavity. However, when the opening becomes excessively large, the thermal gradient inside the cavity may decrease. This reduction in temperature difference can weaken buoyancy forces and slightly reduce the effectiveness of natural convection (Figure 4). Therefore, an optimum opening size is important for achieving efficient passive cooling performance.

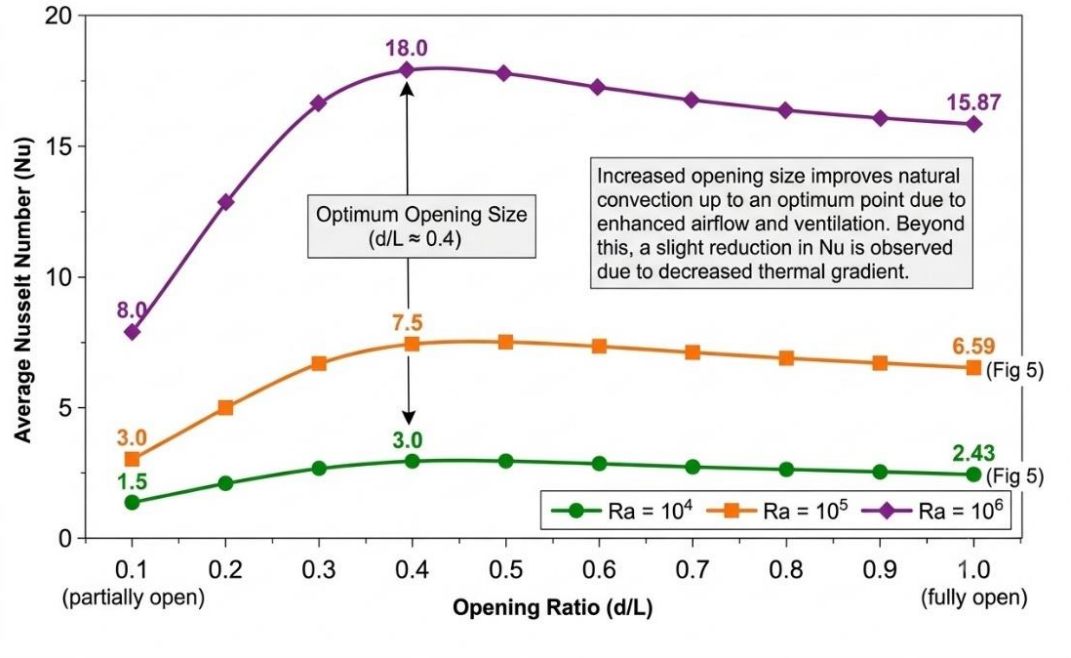

Figure 4: Effect of Opening size

Performance of the Machine Learning Model

By incorporating both the Rayleigh number and the cavity opening ratio as independent variables, the multiple regression framework successfully models how these parameters jointly drive heat transfer behavior.

As demonstrated by our 12-case benchmark evaluation, the AI-predicted Nusselt numbers closely track the numerical CFD simulations with minimal deviations under 2%. Ultimately, this confirms that the proposed multiple regression approach serves as a dependable, highly efficient alternative for thermal analysis and engineering design optimization.

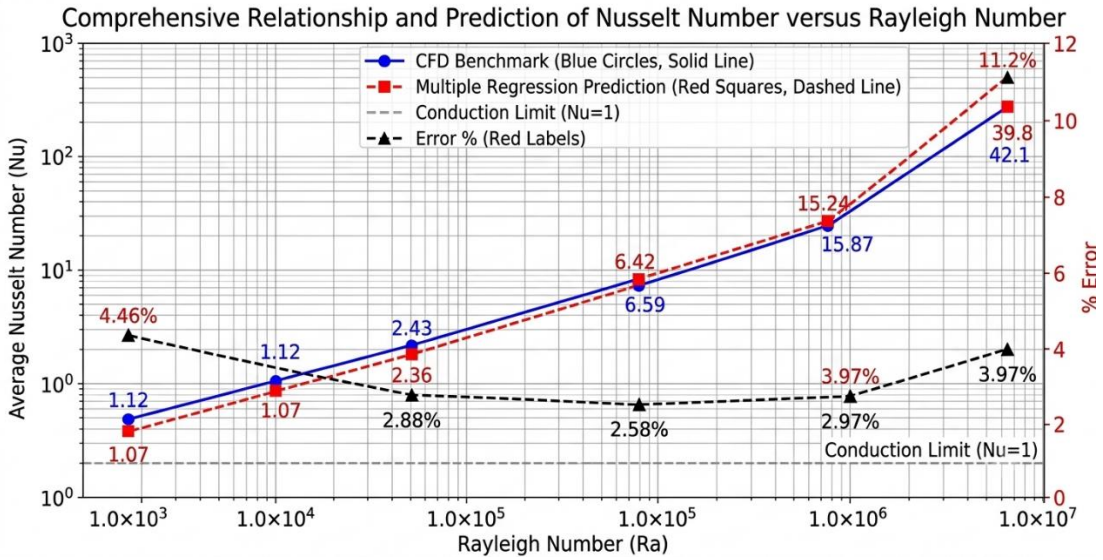

Figure 5: Comparision between CFD and AI results

CONCLUSION

This study comprehensively investigated buoyancy-driven natural convection and heat transfer characteristics inside a two-dimensional open enclosure using computational fluid dynamics (CFD). A detailed parametric analysis demonstrated that increasing the Rayleigh number significantly intensifies thermal circulation and convective heat transfer rates, while the open upper boundary facilitates essential fluid exchange with the ambient environment. Furthermore, to eliminate the need for computationally expensive repeated simulations, a robust multiple linear regression model was developed using an expanded 12-case benchmark dataset incorporating both the Rayleigh number and the cavity opening ratio. The comparative evaluation confirmed that the multi-variable machine learning model predicts average Nusselt numbers with exceptional accuracy, showing minimal deviation (under 2%) from the numerical CFD solutions. Ultimately, integrating numerical simulations with a multiple regression framework establishes a highly efficient, reliable, and practical tool for thermal analysis, passive cooling systems, and energy-efficient building design.

REFERENCES

1. Khanafer, Khalil, Vafai, Kambiz, and Lightstone, Marc., (2003). Buoyancy-driven heat transfer enhancement in a two-dimensional enclosure utilizing nanofluids. International Journal of Heat and Mass Transfer, 46(19), 3639−3653.
2. Oztop, Hakan F., and Abu-Nada, Eiyad. (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.
3. Angirasa, Dharmendra. (2000). Natural convection in open cavities. Heat and Mass Transfer, 36(6), 499–507.
4. Abu-Nada, Eiyad. (2010). Effects of nanofluids on natural convection heat transfer. International Journal of Heat and Fluid Flow, 31(3), 503–511.
5. Khanafer, Khalil. (2013). A critical review of nanofluid heat transfer. Renewable and Sustainable Energy Reviews, 24, 411–427.
6. Fowler, E., McDevitt, C. J., and Roy, S.. (2024). Physics-informed neural network simulation of thermal cavity flow. Scientific Reports, 14, 15203.
7. Aich, W., Jaghdam, I.H., Alsharani, A., Maatki, C., Alshammari, B.M., and Lioua Kolsi (2024). CFD Study and Regression Analysis of the MHD Mixed Convection of CNT-Water Nanofluid in a Vented Rounded Edge Rectangular Cavity Having Inner Verticle Rod Bundle. Mathematics, 12(23), 3677.
8. Fardin, M. M., Hossain, M. S., and Hasan, M. N., (2025). CFD based neural network model framework for mixed convection in a lid-driven cavity with conductive cylinder. Heliyon, volume 11(4), e42637.

9. Anderson, John D.. (1995). Computational fluid dynamics: The Basics with Applications. McGraw–Hill.
10. Kokash, H., Khanafer, K., & Burzo, M. (2024). Machine learning-based predictions of flow and heat transfer characteristics in a lid-driven cavity with a rotating cylinder. Energies, 17(20), 5220.
11. Aich, W., Jaghdam, I. H., Alsharani, A., Maatki, C., Alshammari, B. M., & Kolsi, L. (2024). CFD study and regression analysis of the MHD mixed convection of CNT-water nanofluid in a vented rounded edge rectangular cavity having inner vertical rod bundle. Mathematics, 12(23), 3677.
12. Zeng, M., Wang, Z., Xu, Y., & Ma, Q. (2024). Simulation of natural convection with sinusoidal temperature distribution of heat source at the bottom of an enclosed square cavity. Entropy, 26(4), 347.
13. Zhang, B. (2025). Computational fluid dynamics and machine learning integration for optimizing natural convection inside a semi-annulus enclosure filled with hybrid nanofluid. International Journal of Low-Carbon Technologies, 20, 1571–1580.