Journal of Information Systems Engineering and Management
2025, 10(4)
e-ISSN: 2468-4376
https://www.jisem-journal.com/

Research Article

Mixed Convection in Square Cavity with Constant Heat Source on the Bottom Wall and Isothermal Sidewalls Moving in Opposite Directions

Fatima Zemmouri[1,2], Farid Gaci[1], Azeddine Belhamri[2]
[1] *Laboratory of Applied Energetics and Pollution, Frères Mentouri University-Constantine 1, Algeria*
[2] *Laboratoire de Génie Climatique de Constantine, Algeria*
* *Corresponding Author: zemmouri66@yahoo.fr*

ARTICLE INFO

Received: 18 Mar 2025

Revised: 02 Jul 2025

Accepted: 20 Jul 2025

ABSTRACT

Laminar mixed convection in a square cavity, with a heat source located on the horizontal bottom wall and two vertical isothermal sidewalls moving in opposite directions, is numerically investigated. The heat source represents a heater or an electronic component, located at the bottom in such an enclosure. The influence of reversal of the direction of movement of the side walls (mixed convection contrary) with Richardson number, on the heat transfer in the cavity, is described by the Nüsselt number. Results are obtained for several values of the Richardson number Ri (0.1 to 10) with a Prandtl number Pr = 0.71 at Re =100. Different heat source lengths with different positions are considered.

Keywords: laminar mixed convection, square cavity, moving side walls, reversal direction.

1. INTRODUCTION

Mixed convection in cavities represents a topic of confirmed practical interest. Many technological applications are concerned, particularly in the case of cooling electronic components. The review of the literature concerning mixed convection in ventilated cavities shows that the subject still remains to be explored. The interaction of shear flow due to wall motion and natural flow due to buoyancy effect is a fundamental research area so far and requires comprehensive analysis to understand the physics of the resulting flow and heat transfer. Currently, mixed convection in closed cavities has been the subject of several theoretical, experimental and especially numerical studies. Ghia et al. [1], Shaw [2], Aydin and Yang [3], Hsu and Wang [4] and Guo and Sharif [5] have carried out numerical studies on mixed convective heat transfer in enclosure. Emphasis is placed on the influence of the governing parameters, such as Reynolds number, Re, buoyancy parameter, Gr/Re2, location of the heat sources, and the conductivity ratio on the thermal phenomenon in the enclosure. V. Sivakumar et al. [6] Cheng and Liu [7] have studied numerically several different values of the heat source length, the aspect ratio of the cavity, as well as symmetric and asymmetric placement of the heat source were considered. Kareem and Gao [8], Yang and Du [9] were interested in the use of nano-fluids to improve heat transfer and ensure better cooling. For this same purpose, the present study considers a square cavity with constant heat source on the bottom and isothermal sidewalls moving in opposite directions. It is then, a question of determining the relative influence of the two modes of force (buoyant force and shear force) on the heat transfer induced and to evaluate the improvements generated.

2. MATHEMATICAL FORMULATION

2.1. Problem description

The physical model considered here is shown in Figure (1). The enclosure represents a practical system such as an air-cooled electronic device, where the moving sidewalls are an idealization of the cold airflow along the sides of the cavity. It consists of a square cavity with length L, whose sidewalls are moving in the opposite direction with a uniform velocity, V0, and are kept at a constant temperature (Tc) The lower wall has an embedded symmetrical heat source (Th), and length l. The remaining parts of the bottom wall and the entire upper wall are adiabatic. The flow in this cavity is induced by the shear force resulting from the movement of the side walls combined with the buoyancy force resulting from the heat source.

Journal of Information Systems Engineering and Management
2025, 10(4)
e-ISSN: 2468-4376
https://www.jisem-journal.com/

Research Article

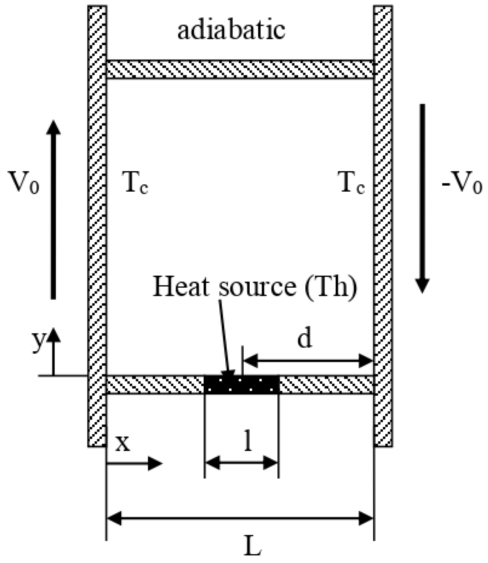

Figure 1. Schematic diagram of the physical system

2.2. Mathematical model

The following assumptions are considered: The flow is two-dimensional, laminar, the fluid is Newtonian and incompressible (Pr = 0.71). The heat transfer by radiation, the work induced by the viscous and pressure forces are negligible and the physical properties of the fluid are constant except the density, which obeys the Boussinesq approximation in the term of the Archimedes thrust. The governing equations can be expressed in the dimensionless form as:

$$\partial U/\partial X + \partial V/\partial Y = 0 \quad \text{(Eq. 1)}$$

$$U\,\partial U/\partial X + V\,\partial U/\partial Y = -\partial P/\partial X + 1/Re\,(\partial^2 U/\partial X^2 + \partial^2 U/\partial Y^2) \quad \text{(Eq. 2)}$$

$$U\,\partial V/\partial X + V\,\partial V/\partial Y = -\partial P/\partial Y + 1/Re\,(\partial^2 V/\partial X^2 + \partial^2 V/\partial Y^2) + (Gr/Re^2)\,\theta \quad \text{(Eq. 3)}$$

$$U\,\partial T/\partial X + V\,\partial T/\partial Y = 1/Pr\,Re\,(\partial^2 T/\partial X^2 + \partial^2 T/\partial Y^2) \quad \text{(Eq. 4)}$$

All distances are normalized by L, all velocities are normalized by U0, and pressure is normalized by $\rho U0^2$; ρ being the fluid density and heat source length $\varepsilon = l/L$.

$X = \frac{x}{L}$; $Y = \frac{y}{L}$; $U = u/U0$; $V = v/V0$;

The boundary conditions for the present problem are specified as follows:

On the upper wall

$\partial T/\partial y = 0$; $U = V = 0$ $\quad$ *for*: $0 < x < L$ *et* $y = L$

At the right and left wall levels

$T = Tc$; $U = 0$ *and* $VL = V0, VR = -V0$ $\quad$ *for*: $= 0, x = L$ *and* $0 < y < L$

At the heat source part on the bottom wall

$T = Th$ *and* $U = V = 0$ $\quad$ *for*: $0.4\,L \leq x \leq 0.6\,L$ *and* $y = 0$

At the levels of the adiabatic parts of the lower wall

$\partial T/\partial y = 0$ *and* $U = V = 0$ $\quad$ *for*: $0 < x < 0.4\,L$, $0.6\,L < x < L$ *and* $y = 0$

Where: V_L, V_R: represent the velocity of the left and right walls respectively

The local heat transfer coefficient is defined by $h(x) = q/(Th(x) - Tc)$ at a given point on the heat source surface where $Th(x)$ is the local temperature on the surface.

2.3. Numerical resolution

The commercial finite volume-based code "ANSYS-FLUENT" is used to solve the mathematical model. The normalized length of the constant flux heat source at the bottom wall, "ε" is varied to be 1/5, 2/5, 3/5, and 4/5. Firstly, computations were performed at Ri = 0.1, 0.5, 1, 2 and 10 and keeping the Reynolds number, Re, fixed at 100. To

Journal of Information Systems Engineering and Management
2025, 10(4)
e-ISSN: 2468-4376
https://www.jisem-journal.com/
Research Article

allow grid independent examination, a numerical procedure has been conducted for different grid resolutions. A comparison is made with the numerical results found by Ghia et al. [1] and Aydin and Yang [2] for a velocity relative to the median plane and dimensionless vertical velocity profiles, respectively. It was concluded that the grid system of (150 X 150) is fine enough to obtain accurate results. Moreover, a non-uniform grid mesh is used, which is finer in vicinity of the source and vertical walls in order to increase the accuracy of the results.

3. RESULTS AND DISCUSSION

3.1. Effect of Richardson number and heat source length on streamlines

The governing parameters are Richardson number and heat source length. Investigations through the cavity are made for ranges of Richardson number from 0.1 to 10 at Re=100. In each case, the flow rises on the left side rubbing the left movable side wall and turns horizontally after being blocked at the level of the adiabatic upper wall to then follow the direction of movement of the right side wall hence the formation of a single vortex where the direction of recirculation is that clockwise. Figure (2) shows examples of representative results. Mixed convection regime becomes more discernible with the shift of isotherms to the left. Beyond Ri = 0.5, the buoyancy starts to show its effect on the transport and become comparable to shear effects up to values lower than 2 [3]. For all dimensionless source lengths "ε", the geometry of the flow field is not affected considerably by changing Ri and heat source length; however the temperature field is. For this reason, the evaluation for all the cases is based on the isotherms.

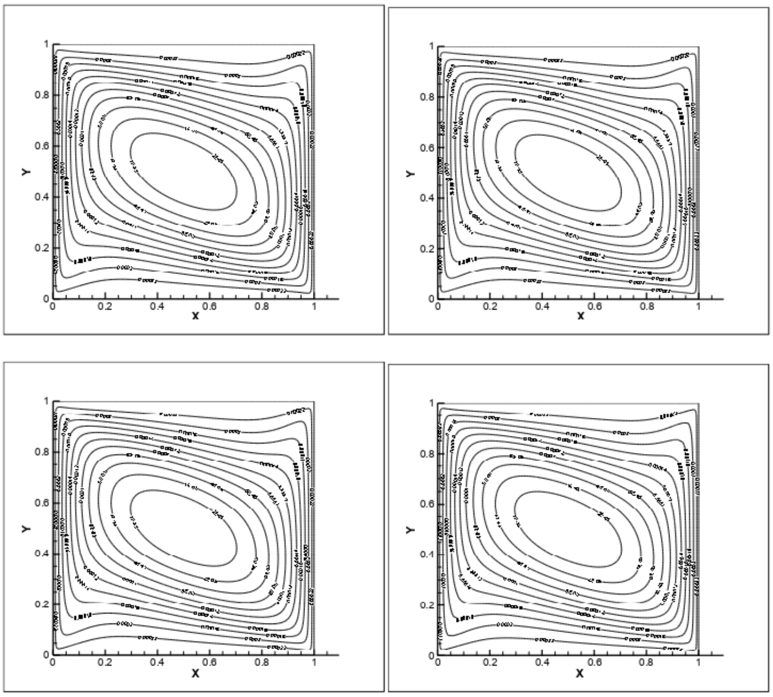

Figure 2. Streamlines for mixed convection case with, respectively
(ε = 1/5, Ri = 0.5), (ε = 1/5, Ri = 2), (ε = 4/5, Ri = 0.5), (ε = 4/5, Ri = 2)

3.2. Effect Of Richardson Number And Heat Source Length On Isotherms

The convection region adjacent to the heat source becomes thinner and denser producing higher temperature gradients with increasing Ri (figure 3). Similar behavior is also observed for cases with other values of "ε" and Ri. Isotherms become denser on the right side of the source following the crossing of the two forces which induced a movement of the latter towards the left side.

Journal of Information Systems Engineering and Management
2025, 10(4)
e-ISSN: 2468-4376
https://www.jisem-journal.com/

Research Article

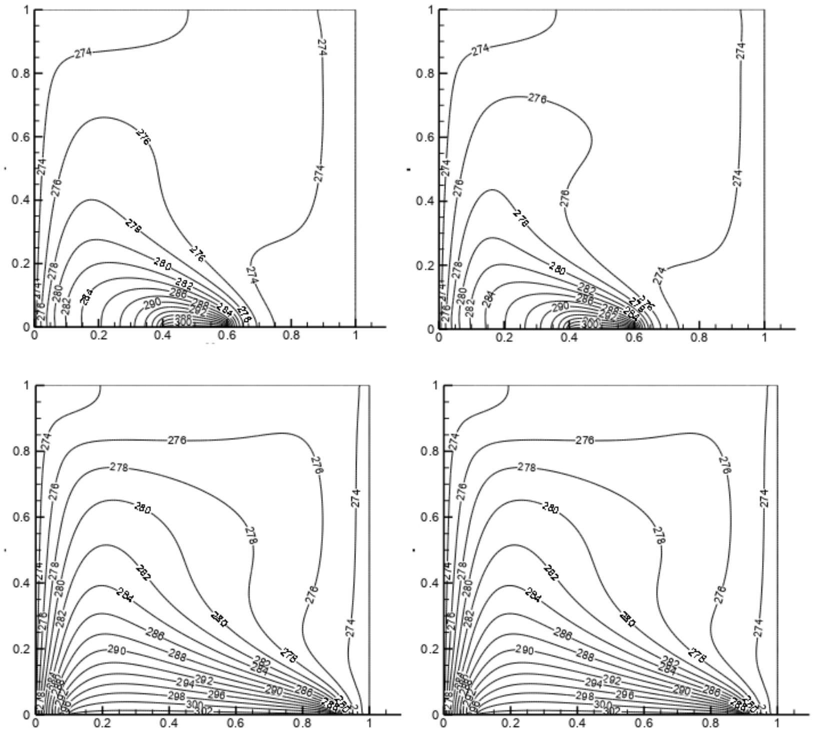

Figure 3. Isotherms for mixed convection case with, respectively
(ε = 1/5, Ri = 0.5), (ε = 1/5, Ri = 2), (ε = 4/5, Ri = 0.5), (ε = 4/5, Ri = 2)

3.3. Effect of Richardson number and heat source length on Nüsselt number

The effect of increasing Richardson number and the heat source length on the Nüsselt number for the different simulations are shown in figure (4). The Nüsselt number along the bottom heated wall of the cavity increases continuously with increasing Ri and heat source length.

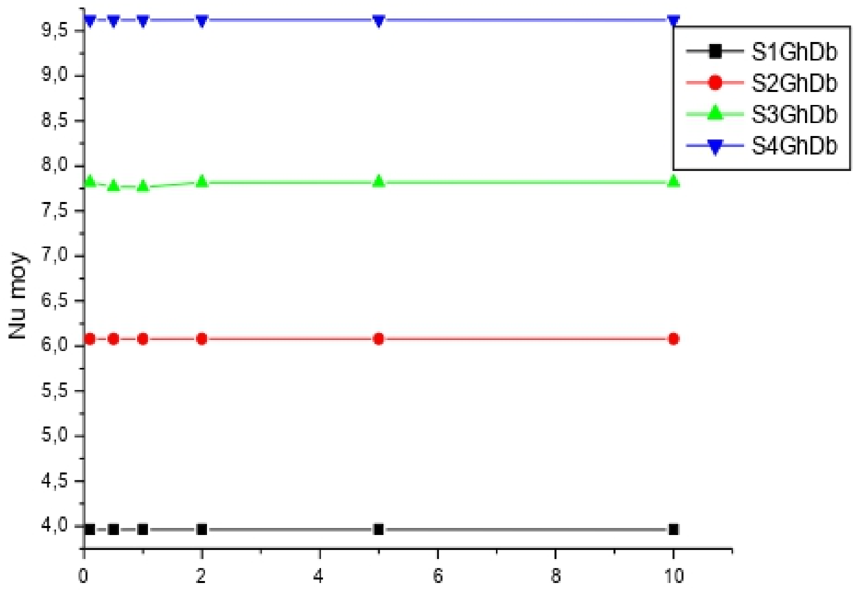

Figure 4. Nüsselt number for different Ri.

Journal of Information Systems Engineering and Management
2025, 10(4)
e-ISSN: 2468-4376
https://www.jisem-journal.com/

Research Article

3.4. Asymmetry

Figure (5) shows a case of walls with opposite directions (left upwards and the right downwards) with representative values: Ri=0.1 and Re=100, in the case of asymmetric placement of the heat source. The streamlines form cells rotating in a single direction, which are favored by the opposing movement of the walls. Asymmetry is obvious, depending on the position of the heat source.

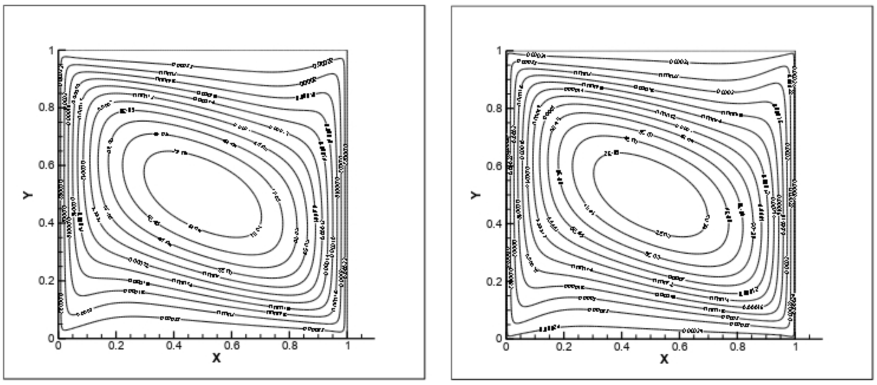

Figure 5. Asymmetry effect of the heating element on streamlines (d/L=0.4, d/L=0.2)

The isotherms are denser on the right side of the source and move towards the left (figure 6), Nusselt number increases as the source moves more and more towards the side walls (Figure 7); the cooling process becomes more efficient.

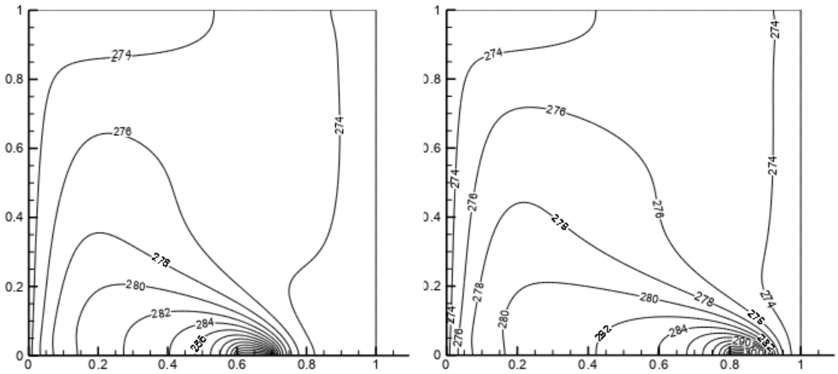

Figure 6. Asymmetry effect of the heating element on isotherms (d/L=0.4, d/L=0.2)

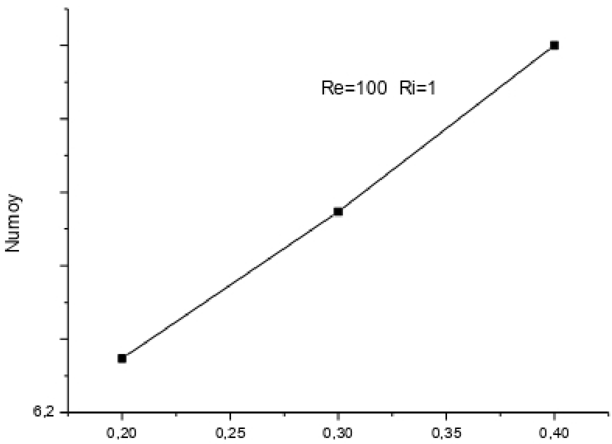

Figure 7. Asymmetry effect (d/L) of the heating element on the Nüsselt number

Journal of Information Systems Engineering and Management
2025, 10(4)
e-ISSN: 2468-4376
https://www.jisem-journal.com/

Research Article

4. CONCLUSION

This study focuses on mixed convection in a square cavity, with heat source at the bottom and isothermal vertical sidewalls moving in opposite directions. Generally, the resulting flow consists of a single vortex, where the direction of recirculation is that clockwise. The geometry of the flow field is not affected considerably by changing Ri and heat source length, however the temperature field is. The isotherms become denser on the right side of the source following the crossing of the two forces, which induced a movement of the latter towards the left side of the cavity, which also explains the propagation and orientation of the heat from the source towards the left, compensating with the bottom corner. Of course, the heat-affected region becomes larger with the increasing heat source length. For the asymmetric configuration, a change in the flow inside the cavity is observed, the maximum temperature decreases and Nu increases as the heat source becomes more and more asymmetric. The closer an electronic component is located adjacent to the sidewalls, the better the cooling effect or heat removal is achieved.

REFRENCES

[1] Ghia U, Ghia K N, Shin C T, High-Re solutions for incompressible flow using the Navier-Stokes equations and a multigrid Method, Journal of Computational Physics 48, 387-411 (1982), https://doi.org/10.1016/0021-9991(82)90058-4.

[2] Heiu-Jou Shaw, Laminar mixed convection heat transfer in three-dimensional horizontal channel with a heated bottom, Numerical Heat Transfer, Part A, vol. 23, pp. 445-461, (1993), https://doi.org/10.1080/10407789308913682.

[3] O. Aydin, Wen-Jei Yang, Mixed convection in cavities with a locally heated lower wall and moving sidewalls, Numerical Heat Transfer, Part A: Applications, 695–710, (2000), https://doi.org/10.1080/104077800274037.

[4] H. Hsu and S. G. Wang, Mixed convection in a rectangular enclosure with discrete heat sources, Numerical Heat Transfer, Part A, 38:627-652, (2000), https://doi.org/10.1080/104077800750021170.

[5] G; Guo, M. A.R. Sharif, Mixed convection in rectangular cavities at various aspect ratios with moving isothermal sidewalls and constant flux heat source on the bottom wall, International Journal of Thermal Sci., 43 (2004) 465–475, https://doi.org/10.1016/j.ijthermalsci.2003.08.008.

[6] V. Sivakumar , S. Sivasankaran, P. Prakash , Jinho Lee, Effect of heating location and size on mixed convection in lid-driven cavities, Computers and Mathematics with Applications, (2010), https://doi.org/10.1016/j.camwa.2010.02.025.

[7] T.S. Cheng et W.H. Liu, Effect of Temperature gradient orientation on the characteristics of mixed convection flow in a lid-driven square cavity, Computers and Fluids, Vol. 39, 965–978, (2010), DOI: 10.4236/ojfd.2017.72016.

[8] A. K. Kareem, S. Gao, Computational study of unsteady mixed convection heat transfer of Nano-fluids in a 3D closed lid-driven cavity, International Communications in Heat and Mass Transfer, Vol. 82, pp. 125–138, (2017), https://doi.org/10.1016/j.icheatmasstransfer.2017.02.009.

[9] L. Yang, K. Du, A comprehensive review on the natural, forced, and mixed convection of non-Newtonian fluids (Nano-fluids) inside different cavities, Journal of Thermal Analysis and Calorimetry, Vol. 140, pp. 2033–2054, (2020), https://doi.org/10.1007/s10973-019-08987-y.