ScienceDirect

International Journal of Thermal Sciences
Volume 203, September 2024, 109137

Mixed convection in a variable size cavity: Effect of flexible wall characteristics

Duna T. Yaseen [a], Salah M. Salih [b], Muneer A. Ismael [c d]

Show more

Share Cite

https://doi.org/10.1016/j.ijthermalsci.2024.109137
Get rights and content

Abstract

Boosting the mixed convection in vented cavities using a moving wall results in a disturbed strong circulation. This study investigates the mitigation of the disturbed circulation to promote the transfer of heat by providing additional size to the circulation. The use of deformable walls with different thicknesses, lengths and modulus of elasticity is suggested for this problem. The top wall of the cavity is adiabatic and moves in the positive direction, and four speeds were studied; 0.5, 1, 2 and 4 times the fluid velocity at the inlet vent. The ratio of the natural to forced convection is controlled by ranging Richardson number by 0.01 ≤ Ri≤100. Three aspect ratios are studied; AR=0.5 (horizontal cavity), 1 and 2 (tall cavity). The problem is treated numerically using the finite element method. Results indicate that the fluid flow inside the vented square cavity changes from major single vortices into multi vortices at high (Ri=100). Results affirm that a flexible wall yields a higher Nusselt number than the rigid wall, it's about 12%, and 20% for Ri=0.01 and 10, respectively. A thicker flexible wall augments the Nusselt number for Ri≤10. For specified conditions, the Nusselt number of the horizontal cavity is 17.7% greater than the tall cavity. The lid speed of $u_{Lid}=4\ u_{in}$ exhibits a competent role in raising the Nusselt number for low Richardson number.

Introduction

The management of energy transportation is a critical function in many engineering systems encountered in room cooling and ventilation, electronic apparatus cooling, PC cooling, etc., and when this issue is resolved, heat and mass transfer in such systems can be optimized [[1], [2], [3]]. Several methods for heat control are available such as the use of baffles, porous insertions, solid obstacles, moving walls and so on. The use of active methods such as moving and rotating objects is always fraught with disturbance, which inherently generates additional heat losses. Passive methods such as the use of baffles or nano-additives may be limited in action. For a comprehensive understanding of the main focus and purpose of the research being undertaken, an examination of the survey of various papers that have been conducted on the subject matter is conducted. Alsabery et al. [4] studied water-based nanofluid energy transmission in a heated enclosure with moving vertical walls and an inner solid cylinder. The authors found that the size of the inside cylinder, the roof waviness, and the nanofluid properties offer good options for controlling the heat transfer inside the cavity. The effect of a moving wall and magnetic field on the mixed convection circulation and thermal flow within a wavy cavity containing heat sources was studied by Oztop et al. [5]. They observed that the heat transfer rate decreases with a higher Hartmann number, while the concentration of nanoparticles has a significant effect on the heat transfer rate. Researchers discovered that changing the concentration of nanoparticles has a significant effect on heat transmission.

Mohamed and Viskanta [6] examined how a moving lid affected the thermal and fluid dynamics in a lid-driven cavity. They indicated that the heat transfer rate increases as the Reynolds (Re) number increases or the Richardson number (Ri) decreases. In a square-lid-driven enclosure with a heated bottom, Moallemi and Jang [7] scrutinized the mixed convective flow conditions numerically. They addressed how the Prandtl number affects the process of flow and heat transfer. Results discovered that the effects of buoyancy become noticeable at larger Prandtl numbers. Also, the averaged *Nu* is significantly influenced by Reynolds, Richardson, and Prandtl numbers. Ismael et al. [8] considered the heat transfer in a channel connected with a lid-driven trapezoidal cavity. They used different directions of side wall cavities with various parameters such as Richardson number and Reynolds number ratio (Rer) on the thermal-fluid flow. Their results found that the heat transfer increment becomes noticeable at larger Ri and Rer numbers, and the higher *Nu* enhancement is 163% was obtained when the velocity of the lid is four times the air inlet velocity (Rer=4).

Al-Amiri et al. [9] executed the buoyancy influence of heat and mass diffusion in a square cavity with a sliding top wall. They discovered that, for low values of the Richardson number, the moving lid's mechanical action increases the features of heat and mass transfer through the cavity. Furthermore, the complexity of heat and flow is increased when obstructions or cylinders are placed inside the cavity created by the lid. This interaction between shear and buoyancy-driven flows increases complexity. Esfe et al. [10] discussed the transfer of heat from a block attached to the bottom of a horizontal cavity with a moving lid. To enhance the process of mixed convection, they charged the cavity with Al_2O_3 nanoparticles-water based and imposed the variation of the nanofluid properties by the temperature as well as with the concentration. His results were not so huge thus his conclusions were limited to the adverse impact of the Richardson number on the Nusselt number, while the nanoparticles concentration increased. In another study, Esfe et al. [11] promoted their previous study [10] by imposing more parameters these are the position of the hot block on the cavity base and the inclination of the cavity. The adverse impact of the Richardson number held out while the inclining of the cavity did not give hoped enhancement. Random variations of the Nusselt number were observed with the inclination of the cavity.

Islam et al. [12] investigated a variety of parameters to determine how heated "square" blockage affects mixed convection characteristics inside a lid-driven cavity, such as block placement. They showed that the Nusselt number diminishes by increasing the ratio of obstruction and reaches the lowest value at a blockage ratio of larger than 1/2, and as a further increase of the blockage ratio, *Nu* rises again and becomes independent of the Ri number. Gangawane et al. [13] scrutinized the moving lid of a square cavity involving a triangular blockage. They focused their assessments on the size of the blockage and its vertical position. Their outcomes led them to recommend the central position of the blockage to get the highest Nusselt number. As in Refs. [10,11], the Nusselt number was a decreasing function to the Richardson number. Apart from the Richardson number, they predicted two correlations relating the Nusselt number with the Reynolds number and the size and position of the block. Çiçek and Baytaş [14] considered the mixed convection and entropy generation inside a square-vented cavity. They found that the minimum overall entropy generation at $Ri=5$ for given ranges of Re number excepting Re=50. Mixed convective flow aligned with two horizontal cylinders under different conditions was researched by Al-Omar and Ali [15]. They observed that increasing the space between the cylinders enhances the heat transfer increment around the hot cylinder, where the increment in heat transfer rate reached from 10 % to 38 %. Zhang et al. [16] investigated the inverse problem of mixed convective flow in a square enclosure under unknown wall heat fluxes. They imposed several profiles of heat flux on the vertical walls to ascertain boundary heat flux by solving the temperature distribution within the circulated fluid. The central vortex strengthening by the Reynolds number imparts homogeneous temperature within the cavity. They noticed a decrease in the accuracy of the inverse problem when the Reynolds number goes higher.

Radhakrishnan et al. [17] conducted an experimental and numerical investigation of nine plate heaters distributed alternatively inside a vented cavity. To optimize the maximum heat dissipation from the heaters, the authors followed two approaches namely; the trial and error approach and the response surface approach. They proposed the last approach as it gave accurate results. They stated that heaters that are in the air stream can be loaded more than the other heaters. Rodrigues and de Lemos [18] addressed the mixed convection in a filtration process, where hot air enters downward a cavity with a porous layer insert. The outer wall of the cavity is subjected to an external convection. They considered the turbulence flow and the local thermal non-equilibrium between the air and the matrix of the filter. They explained a reduction in the heat exchange with a high Darcy number because of the high velocity within the filter. Hence, they got higher heat exchange efficiency at a lower Darcy number. Prakash and Singh [19] built an experimental rig to test the thermal aspects in a chamber comprising a heater along the right wall. Air is sucked from the exit opening located above the heater and enters from an inlet opening located at the lower of the left wall. A numerical solution approximating the problem to two-dimension was conducted to support the experimental results. Radiation was taken into account in the numerical solution. The radiation from the hot surface was the key parameter in governing the flow circulation and hence the heat transfer inside the chamber.

Regarding the indoor comfort demands, Xaman et al. [20] constructed a thorough numerical study simulating the process of ventilating a space heated from the left by a finite conductive wall. Cold air permeates the space with a specified velocity. To get a better distribution of the temperature in the space, four exit locations were inspected, one on the left wall and three on the top wall. Two materials and three thicknesses of the right wall were investigated, while it was subjected to a constant heat flux. The results proposed an exit location on the top wall and closest to the right vertical wall. Bennacer and Ma [21] considered the homogeneity of temperature in an indoor space subdivided into four domains by three cold partitions. The space was charged with air from an opening position in the hot wall. The study was performed numerically. The results implied to temperature homogeneity with a higher Rayleigh number and equally distributed flow throughout the indoor space at fully dominated forced convection.

Additional techniques to enhance the heat removal process include the use of a flexible wall or a flexible baffle called fluid-structure interaction (FSI) [[22], [23], [24], [25], [26], [27]]. A common feature among these papers is that they contain either a flexible fin or a flexible wall and these flexible segments deform based on the force of the fluid. The main remark that can be drawn from these papers is that the flexible walls serve favorably in elevating the Nusselt number as they provide extra spaces for fluid circulation [24,26], while the flexible fins [22,23,25,27] are not better than the rigid fins in enhancing the Nusselt number. However, the pressure drop mitigates with these flexible parts. Alternatively, in a vented chamber, Ismael and Jasim [28] placed a flexible fin in a strategical location namely facing to the inlet port, and found perceptible enhancement in the Nusselt number. A 230 % increase in *Nu* was obtained in comparison to a cavity devoid of a flexible fin due to the passive fin's oscillations with the flow input to the cavity. The fluid-structure interaction between the flexible bottom wall of a two-dimensional vented lid-driven cavity was numerically explored by Amani et al. [29]. Three types of viscoelastic fluid were implemented. The noted deformation in the flexible bottom was lesser in the case of viscoelastic fluid than the Newtonian fluid. Their study was focused on the numerical characteristics of the fluid-structure interaction rather than the heat aspects, where they found irregular deformation in the bottom when the relaxation time was raised from 0.1 to 0.15. Alsabery et al. [30] employed flexible wall segments postulated in the upper wall of a heated abdominal aortic aneurysm. The shear-thinning Carreau model assumes blood to be a non-Newtonian fluid. The result showed that in sick arteries, blood viscosity should be reduced since a greater power-law fluid index increases shear stress. In another study, Alsabery et al. [31] investigated transient entropy production as well as mixed convection due to the rotation of a hot cylinder in a square cavity with flexible sidewalls. They demonstrated that when the circular cylinder is turned counter-clockwise and the modulus of elasticity for the flexible wall is smaller, the maximum average Nusselt number has emerged.

With the exception of the recent work of Yaseen et al. [32], no other study has been published for mixed convection in a vented square lid-driven cavity with a deformable sidewall. They employed a flexible side wall vented lid-driven cavity and three different conditions of lid-driven speed with varied directions. The result demonstrated that the flexible wall alone boosts the Nusselt number by around 4.5%, but when the top wall moves in a certain direction, the enhancement rises to 28%. Hence, the literature survey showcases ↗ a gap in studying the impact of the variable size of the vented cavities. This gap, as such, motivated us to delve with the augmentation of heat transfer of lid-driven vented cavities by considering a flexible wall that inflates and then increases (or decreases) the size of the cavity. The enlarged space of the cavity is expected to strengthen the heat exchange. To promote more understanding of the suggested study, the speeds of the moving lid, the thickness of the flexible wall and the aspect ratio of the cavity are also investigated. It is sought that such results are new and contribute notably to the field of promoting heat exchange in cooling electric equipment or in ventilating indoor spaces.

Section snippets

Mathematical model

Fig. 1 interprets the geometry of the problem, which consists of a cavity of length *L* and width *H*. The ratio of *L*/*H* is defined as the aspect ratio *AR*. The cold fluid (T_c) is supplied into the cavity through a port located on the left flexible wall (FW), with a width of 0.05*H* at a uniform velocity of $u=u_{in}$ and is exhausted from a port located on the adiabatic right wall. The cavity's top wall is adiabatic and moves with $u_{Lid} = \{0.5, 1, 2, 4\}\ u_{in}$ in the same direction. A segment of 0.6H is set on…

Numerical method and validation

The use of FSI is very important to enhance the heat transfer between fluids entering the open cavity and the flexible wall. The dimensionless governing equations (6), (7), (8), (9), (10) are treated using the arbitrary Lagrangian-Eulerian (ALE) formulation and computed using the Galerkin method of Weighted Residuals Finite Element (FEM). These equations have undergone a weak form transformation [33,34] and the moving mesh caused by the deformation of the flexible wall is firmly taken into…

Results and discussion

This section provides a detailed description of the results of the FSI and pertinent variables on fluid flow and heat transmission via the streamlines, isotherms and the average Nusselt number. The pertinent variables are; $0.01 \le Ri \le 100$, modulus of elasticity $E=10^3–10^{15}$, AR=0.5, 1, 2 and the thickness of the flexible wall $t = 0.01H - 0.04H$ with fixed values of Re_{in}=200 and Pr=0.71. The Reynolds number ratio of lid-driven cavity is set as $Re_r = \{0.5, 1, 2, 4\}$. The case of the rigid wall RW,…

Conclusions

A flexible wall has been used in the numerical analysis of coupled a lid-driven cavity with mixed convection and FSI approach. The cavity involves two ports, inlet at the left wall and two outlets at the top wall. The characteristics of the flexible wall is varied and compared with another rigid one. From this research, it is possible to conclude the following.

1 The cavity with a flexible wall (FW) has a higher Nusselt number than the cavity with a rigid wall (RW). Specifically↗, for Ri=0.01 and …

…

CRediT authorship contribution statement

Duna T. Yaseen: Visualization, Software, Methodology, Investigation, Formal analysis, Data curation. **Salah M. Salih:** Writing – review & editing, Writing – original draft, Methodology, Investigation, Funding acquisition, Conceptualization. **Muneer A. Ismael:** Writing – review & editing, Validation, Resources, Project administration, Investigation, Data curation, Conceptualization.…

Declaration of competing interest

The authors declare that there are no conflicts of interest.…

Recommended articles

References (35)

A. Alsabery
Impact of nonhomogeneous nanofluid model on transient mixed convection in a double lid-driven wavy cavity involving solid circular cylinder
Int. J. Mech. Sci. (2019)

H.F. Öztop
Mixed convection of MHD flow in nanofluid filled and partially heated wavy walled lid-driven enclosure
Int. Commun. Heat Mass Transfer (2017)

A. Mohamad *et al.*
Flow and heat transfer in a lid-driven cavity filled with a stably stratified fluid
Appl.Math. Modelling (1995)

M. Moallemi *et al.*
Prandtl number effects on laminar mixed convection heat transfer in a lid-driven cavity
Int. J. Heat Mass Transfer (1992)

A.M. Al-Amiri *et al.*
Numerical simulation of combined thermal and mass transport in a square lid-driven cavity
Int. J. Therm. Sci. (2007)

A.W. Islam *et al.*
Mixed convection in a lid driven square cavity with an isothermally heated square blockage inside
Int. J. Heat Mass Transfer (2012)

K.M. Gangawane *et al.*
Mixed convection in a lid-driven cavity containing triangular block with constant heat flux: effect of location of block
Int. J. Mech. Sci. (2019)

S. Saha *et al.*
Comment on" mixed convective flow in a multiple port ventilation square cavity with insulated baffle"[case stud. Therm. Eng. 30,(2022) 101785]
Case Stud. Therm. Eng. (2023)

S.O.O. Al-Omar *et al.*
Mixed convection from two horizontally aligned hot and cold circular cylinders in a vented square enclosure
Ain Shams Eng. J. (2023)

T. Radhakrishnan *et al.*

Optimization of multiple heaters in a vented enclosure–A combined numerical and experimental study

Int. J. Therm. Sci. (2010)

View more references

Cited by (0)

View full text