PAPER FOR THE YOUNG SCIENTIST AWARD

Numerical Analysis of Vortex Structures in a Square Cavity with Inline and Staggered Heat Sources

ABSTRACT

Unsteady mixed convection in a square cavity filled with air and containing two heat sources is investigated. Each heat source is located on the horizontal top and bottom walls, which are stationary and adiabatic. The vertical sidewalls assumed to be moving in the downward direction. The heat sources are maintained at higher temperature (T_h) in comparison with the vertical walls, which are maintained at cold temperature (T_c). Two different configurations, namely, inline and staggered arrangements, of heat sources are considered. The time dependent incompressible fluid flow governing equations are solved using the finite volume method. The effect of temperature distributions and fluid flow patterns are accomplished for different Richardson numbers ($0.0001 \leq Ri \leq 1$) and Rayleigh numbers ($10^3 \leq Ra \leq 10^5$). Convergence of results get declined for $Ra = 10^4$. Proper Orthogonal Decomposition (POD) method has been used to analyze the spatio–temporal dynamics of vortex structures. It is also observed that the average Nusselt number (*Nu*) for both top and bottom heat sources are increased with increasing *Ri* and *Ra*.

INTRODUCTION & OBJECTIVE

The present study finds application in industrial and technological applications such as heat exchangers, efficient thermal management, cooling electronic devices and nuclear reactors, air-conditioning systems and ventilation. Hidki et al. [1] focused on the thermal performance of the insulated square cavity containing two heat blockages one as the hot block and other as the cold block filled with mixture of Nano-Encapsulated Phase Change Material (NEPCM) and water as a base fluid. Solutions are obtained with several parameters like *Ra* and Darcy number to improve the heat transfer rate within the cavity. The study revealed a remarkable thermal enhancement when NEPCM particles is mixed with base fluid water. Lam and Prakash [2] analysed the fluid flow pattern, entropy generation and temperature distribution of the natural convective heat transfer in a porous cavity containing two heat sources placed on top and bottom walls in an inline and staggered arrangements. The results show that maximum entropy generation is observed (i) in the nearby heat sources due to the high thermal gradient and (ii) heat transfer rate is significantly improved for both inline and staggered arrangements. It is also observed that heat transfer is maximum in inline arrangement rather than the staggered arrangement. Zisan et al. [3] examined the entropy generation and natural convection in a cubic air-filled cavity with a discretely heated sources located at the bottom wall in different configurations. The results show that among all the considered configurations, heat sources placed near the cold wall show the best heat transfer characteristics. Isotherm and stream function plots are smoother at low *Ra* than severe mixing of fluid at high *Ra*. Ghia et al. [4] modeled the driven cavity flow and formulated the vorticity function of two-dimensional incompressible Navier-Stokes equations by using the coupled strongly implicit multigrid method. Accurate results have been obtained for $Re = 10^{10}$ flow with fine-mesh refinements upto grid 257 x 257. Lam and Prakash [5] modeled the impingement cooling system with discrete heat sources to analyse the fluid flow, thermal characteristics and entropy generation by simulating for different values of *Re*, velocity ratio and channel height. The authors examined that the improvement in *Nu* and global entropy generation. They performed spectral analysis and POD to analyse the spatio-temporal dynamics of unsteady configurations.

The objective of the present study is to analyse the thermal characteristics and fluid flow patterns in a square cavity containing internal heat sources arranged in an inline and staggered arrangement on the horizontal walls. The Second order linear upwind discretization technique based finite volume method has been employed to solve Navier-Stokes equations for different values of *Ri*, Aspect Ratio (*AR*) and *Ra*. The effects of orientation (inline and staggered) of heat sources for maximum heat transfer rate is analysed by the contours of isotherm and stream function. For unsteady configurations, spectral and POD analyses are used to investigate the spatio-temporal dynamics. Furthermore, *Nu* and Local Nusselt number (Nu_{loc}) are calculated to analyse the improvement of heat transfer.

PROBLEM STATEMENT

The physical domain of square cavity contains air (Pr = 0.71) and two solid heat sources placed on top and bottom horizontal walls in a two different (inline and staggered) configurations. The left and right walls are maintained at non-dimensional temperature θ = 0 whereas the heat blocks are maintained at θ = 1. The other surfaces are stationary and adiabatic. The computational domain with no-slip boundary conditions are illustrated in the Fig. (1). The governing equations are:

$$\nabla . \boldsymbol{U} = 0 \tag{1}$$

$$\frac{\partial \mathbf{U}}{\partial t} + (\mathbf{U} \cdot \nabla)\mathbf{U} = -\nabla P + \frac{1}{Re}\nabla^2 \mathbf{U} + Ri\, T \tag{2}$$

$$\frac{\partial \mathrm{T}}{\partial t} + (\mathrm{U} \cdot \nabla)T = \frac{1}{RePr}\nabla^2 T \tag{3}$$

where $\boldsymbol{U}$, P and T denote the non-dimensional velocity vector (u, v), pressure and temperature and Re is the Reynolds number.

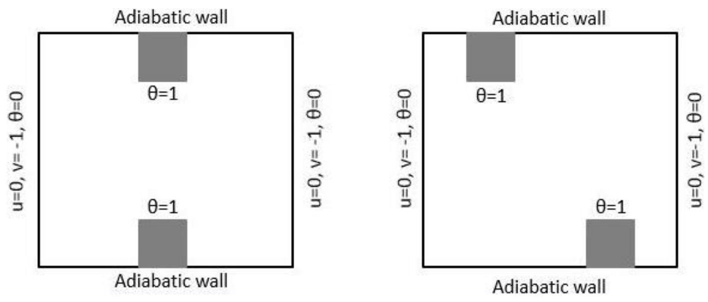

Fig. 1: Schematic diagram of computational domain

RESULTS & HIGHLIGHTS

A computational study using second order linear upwind based finite volume method of Ansys fluent CFD code has been accomplished to determine the thermal characteristics and fluid flow patterns associated with the mixed convection in a square enclosure containing isothermal heat sources. It has been observed that the magnitude of overall Nusselt number has increased with increasing Ra, AR and Ri.

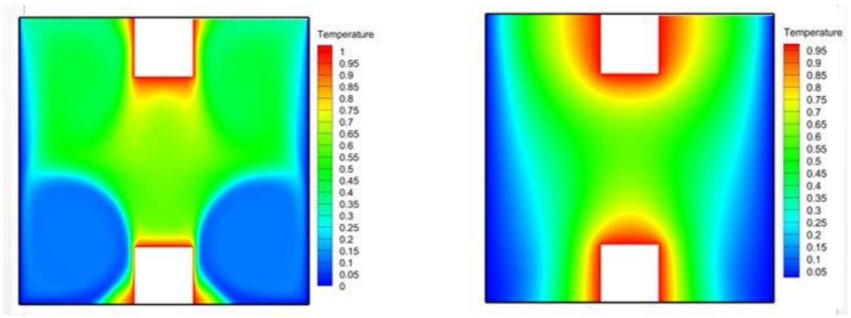

Fig.2: Temperature contours when $Ra = 10^3$ for different (a) $Ri = 0.0001$ (b) $Ri = 1$

References

1. Hidki R., Moutaouakil L. El., Boukendil M., Charqui Z., Zrikam Z., Abdelbaki A., *"Natural convection heat transfer analysis of nano encapsulated phase change material (NEPCM) confined in a porous square chamber with two heat sources",* Journal of Energy Storage., 2023, 73.

2. Prasanth Anand Kumar Lam., and Arul Prakash K., *"A numerical study on natural convection and entropy generation in a porous enclosure with heat sources"*, International Journal of Mass and Heat transfer., 2014, 69, 390-407.

3. HM Toufik Ahmed Zisan., Tahmidul Haque Ruva., and Sumon Saha., *"Entropy generation and natural convection on a cubic cavity with a pair of heat source at different configurations"*, International Communications in Heat and Mass Transfer. 2022, 134.

4. Ghia U., Ghia K. N., Shin C. T.,*"High-Re solutions for incompressible flow using the Navier-Stokes equation and a multigrid method "*, Journal of Computational Physics, 1982, 48.

5. Prasanth Anand Kumar Lam., and Arul Prakash K., *"A numerical investigation and design optimization of impingement cooling system with an array of air jets"* International Journal of Mass and Heat transfer., 2017, 108, 880-900.