A NUMERICAL STUDY OF MIXED CONVECTION IN A SQUARE CAVITY WITH A HEAT CONDUCTING SQUARE CYLINDER AT DIFFERENT LOCATIONS

Md. Mustafizur Rahman[1*], M. A. Alim[1], Sumon Saha[2] and M. K. Chowdhury[1]
[1]Department of Mathematics, Bangladesh University of Engineering and Technology, Dhaka-1000, Bangladesh
[2]Department of Energy and Environmental Engineering, Interdisciplinary Graduate School of Engineering Sciences, Kyushu University, 6-1 Kasuga-koen, Kasuga-shi, Fukuoka 816-8580, Japan
*E-mail: mmustafizurrahman@math.buet.ac.bd

Abstract: *Numerical simulations are carried out for mixed convection flow in a vented cavity with a heat conducting horizontal square cylinder. A two-dimensional solution for steady laminar mixed convection flow is obtained by using the finite element scheme based on the Galerkin method of weighted residuals for different Richardson numbers varying over the range of 0.0 to 5.0. The study goes further to investigate the effect of the inner cylinder position on the fluid flow and heat transfer in the cavity. The location of the inner cylinder is changed horizontally and vertically along the centerline of the cavity. The effects of both Richardson numbers and cylinder locations on the streamlines, isotherms, average rate of heat transfer from the hot wall, the average temperature of the fluid inside the cavity and the temperature at the cylinder center inside the cavity are investigated. The results indicate that the flow field and temperature distributions inside the cavity are strongly dependent on the Richardson numbers and the position of the inner cylinder.*

Keywords: *Finite element method, square cylinder, vented cavity, mixed convection.*

INTRODUCTION

Mixed convection in a cavity is relevant to many industrial and environmental applications such as in heat exchangers, nuclear and chemical reactors and cooling of electronic equipments etc. In engineering applications, the geometries that arise however are more complicated than simple cavity configurations filled with a convective fluid. The geometric configuration of interest is of the presence of cylinder entrenched within the cavity. Several investigators have dealt conjugate heat transfer inside an enclosure with the presence of a body. House *et al.*[1] numerically examined the effect of a centered, square, heat conducting body on natural convection in a vertical square enclosure. They found that heat transfer across the cavity might be enhanced or reduced by a body with a thermal conductivity ratio less or greater than unity. Oh *et al.*[2] numerically studied the natural convection in a vertical square enclosure containing a conducting body generating heat, when a temperature difference existed across the enclosure. They analyzed the variation of streamlines, isotherms and average Nusselt number at the hot and cold walls with respect to temperature difference ratios for each Rayleigh number. However, Lacroix and Joyeux[3] performed a numerical study of natural convection heat transfer from two vertically separated heated cylinder to a rectangular cavity cooled from above. Later on, Lacroix and Joyeux[4] conducted a numerical study of natural convection heat transfer from two horizontal heated cylinders confined to a rectangular enclosure having finite wall conductances. They indicated that wall heat conduction reduced the average temperature differences across the cavity, partially stabilized the flow and decreased natural convection heat transfer around the cylinders.

Nomenclature

d	Dimensional cylinder length (m)
D	Non dimensional cylinder length
g	Gravitational acceleration (ms^{-2})
k	Thermal conductivity of fluid (Wm^{-1}k^{-1})
k_s	Thermal conductivity of cylinder (Wm^{-1}k^{-1})
K	Solid fluid thermal conductivity ratio
L	Length of the cavity (m)
l_x	Dimensional distance between y-axis and the cylinder center (m)
l_y	Dimensional distance between x-axis and the cylinder center (m)
L_x	Dimensionless distance between y-axis and the cylinder center
L_y	Dimensionless distance between x-axis and the cylinder center
Nu	Nusselt number
p	Dimensional pressure (Nm^{-2})
P	Dimensionless pressure
Pr	Prandtl number
Re	Reynolds number
Ra	Rayleigh number
Ri	Richardson number
T	Dimensional temperature (K)
u, v	Dimensional velocity components (ms^{-1})
U, V	Dimensionless velocity components
$\bar{V}$	Cavity volume (m^3)
w	Height of the opening (m)
x, y	Cartesian coordinates (m)
X, Y	Dimensionless Cartesian coordinates

Greek Symbols

α	Thermal diffusivity (m^{2}s^{-1})
β	Thermal expansion coefficient (k^{-1})
υ	Kinematic viscosity (m^{2}s^{-1})
Θ	Non dimensional temperature
ρ	Density of the fluid (kgm^{-3})

Subscripts

av	Average
h	Heated wall
i	Inlet state
c	Cylinder center
s	Solid

Abbreviation

CBC	Convective boundary conditions

Shuja *et al.*[5] numerically studied mixed convection in a square cavity due to heat generating rectangular body and investigated the effect of exit port locations on the heat transfer characteristics and irreversibility generation in the cavity. They showed that the normalized irreversibility increased as the exit port location number increased and the heat transfer from the solid body enhanced while the irreversibility reduced. Also, the influence of vortex shedding on the heat transfer characteristics of the rectangular protruding body was conducted numerically by Shuja *et al.*[6] considering heat transfer enhancement due to flow over a two-dimensional rectangular protruding bluff body. Roychowdhury *et al.*[7] analyzed the natural convective flow and heat transfer features for a heated cylinder placed in a square enclosure with different thermal boundary conditions. Dong and Li[8] studied conjugate effect of natural convection and conduction in a complicated enclosure. They observed the influences of material character, geometrical shape and Rayleigh number on the heat transfer in the overall concerned region. They finally concluded that the flow and heat transfer increased with the increase of thermal conductivity in the solid region and besides, both geometric shape and Rayleigh number also affected the overall flow and heat transfer greatly. The problem of laminar natural convection heat transfer in a square cavity with an adiabatic arc shaped baffle was numerically analyzed by Tasnim and Collins[9], they identified that flow and thermal fields were modified by the blockage effect of the baffle and the degree of flow modification due to blockage was enhanced by increasing the shape parameter of the baffle. At the same time, Braga and de Lemos[10] investigated steady laminar natural convection within a square cavity filled with a fixed volume of conducting solid material consisting of either circular or square obstacles. They used finite element method with a collocated grid to solve governing equations. They showed that the average Nusselt number for cylindrical rods was slightly lower than those for square rods. Recently, Das and Reddy[11] investigated conjugate natural convection heat transfer inside an inclined square cavity with an internal conducting block. At the same time Xu *et al.*[12] experimentally observed the thermal flow around a square obstruction on a vertical wall in a differentially heated cavity and Zhao *et al.*[13] numerically investigated conjugate natural convection in enclosures with external and internal heat sources.

However, there is little information about mixed convection processes when a heat-conducting cylinder exists within a vented cavity and the location of the inner cylinder is moved along the horizontal and vertical centerline of the cavity. In this situation, the flow and heat transfer in the cavity are largely affected by the locations of the inner cylinder for different Richardson numbers. The objective of the present study is to present comprehensive numerical results for the configuration as shown in Figure 1. Finally, the effect of the locations of the inner cylinder for different Richardson numbers on the flow and heat transfer within the cavity is present and explained briefly.

PROBLEM FORMULATION

The physical model considered here is shown in Figure 1, along with the important geometric parameters. It consists of a square cavity with sides of length L, within which a square solid cylinder with size, d and thermal conductivity, k_s is located. A Cartesian co-ordinate system is used with origin at the lower left corner of the computational domain. The top, bottom and left vertical walls of the cavity are kept adiabatic and the right vertical wall is kept at a uniform constant temperature, T_h. The inflow opening located on the bottom of the left wall and the outflow opening of the same size is placed at the top of the opposite heated wall as shown in Figure 1. For simplicity, the size of the two openings, w is set equal to the one-tenth of the cavity length (L). Cold air flows through the inlet inside the cavity at a uniform velocity, u_i. It is also assumed that the incoming flow is at the ambient temperature, T_i and the outgoing flow is assumed to have zero diffusion flux for all dependent variables i.e. convective boundary conditions (CBC). All solid boundaries are assumed to be rigid no-slip walls.

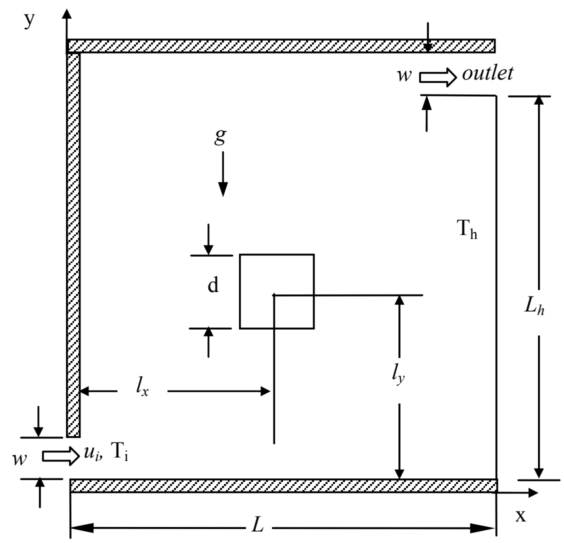


Figure 1: Schematic of the Problem with the Domain and Boundary Conditions

MATHEMATICAL MODEL

The flow within the cavity is assumed to be two-dimensional, steady and laminar with constant fluid properties. The radiation effects are neglected and the Boussinesq approximation is considered. The dimensionless equations describing the flow are as follows:

$$\frac{\partial U}{\partial X}+\frac{\partial V}{\partial Y}=0 \tag{1}$$

$$U\frac{\partial U}{\partial X}+V\frac{\partial U}{\partial Y}=-\frac{\partial P}{\partial X}+\frac{1}{Re}\left(\frac{\partial^2 U}{\partial X^2}+\frac{\partial^2 U}{\partial Y^2}\right) \tag{2}$$

$$U\frac{\partial V}{\partial X}+V\frac{\partial V}{\partial Y}=-\frac{\partial P}{\partial Y}+\frac{1}{Re}\left(\frac{\partial^2 V}{\partial X^2}+\frac{\partial^2 V}{\partial Y^2}\right)+Ri\,\Theta \tag{3}$$

$$U\frac{\partial \Theta}{\partial X}+V\frac{\partial \Theta}{\partial Y}=\frac{1}{Re\,Pr}\left(\frac{\partial^2 \Theta}{\partial X^2}+\frac{\partial^2 \Theta}{\partial Y^2}\right) \tag{4}$$

For heat conducting cylinder, the energy equation is

$$\frac{\partial^2 \Theta_s}{\partial X^2}+\frac{\partial^2 \Theta_s}{\partial Y^2}=0 \tag{5}$$

In the above equations, the dimensionless variables are defined by

$$X=\frac{x}{L},\ Y=\frac{y}{L},\ U=\frac{u}{u_i},\ V=\frac{v}{u_i},\ P=\frac{p}{\rho u_i^{\,2}},\ D=\frac{d}{L},$$

$$L_x=\frac{l_x}{L},\ L_y=\frac{l_y}{L},\ \Theta=\frac{(T-T_i)}{(T_h-T_i)},\ \Theta_s=\frac{(T_s-T_i)}{(T_h-T_i)}$$

The governing parameters i.e., Reynolds number (Re), Richardson number (Ri), Prandtl number (Pr) and the solid fluid thermal conductivity ratio (K) are included in the preceding equations and boundary conditions are defined as

$$\mathrm{Re} = \frac{u_i L}{\upsilon}, Ri = \frac{g\beta(T-T_i)L}{u_i^2}, \mathrm{Pr} = \frac{\upsilon}{\alpha} \; and \; K = \frac{k_s}{k}$$

The appropriate dimensionless boundary conditions (as shown in Figure 1) used to solve Eqs (1)-(5) inside the cavity are given as follows:

At the inlet: U = 1, V = 0, Θ = 0

At the outlet: Convective Boundary Condition (CBC), P = 0

At all solid boundaries: U = 0, V = 0

At the heated right vertical wall: Θ = 1

At the left, top and bottom walls: $\left.\frac{\partial \Theta}{\partial X}\right|_{X=0} = \left.\frac{\partial \Theta}{\partial Y}\right|_{Y=1,0} = 0$

At the solid-fluid vertical interfaces of the block: $\left(\frac{\partial \Theta}{\partial X}\right)_{fluid} = K\left(\frac{\partial \Theta_s}{\partial X}\right)_{solid}$

At the solid-fluid horizontal interfaces of the block: $\left(\frac{\partial \Theta}{\partial Y}\right)_{fluid} = K\left(\frac{\partial \Theta_s}{\partial Y}\right)_{solid}$

The average Nusselt number (Nu) at the hot wall is defined as

$$Nu = \frac{L}{L_h}\int_0^{L_h/L} \left.\frac{\partial \Theta}{\partial X}\right|_{X=1} dY \qquad (6)$$

and the bulk average temperature in the cavity is defined as

$$\Theta_{av} = \frac{1}{\overline{V}}\int \Theta \, d\overline{V} \qquad (7)$$

where, L_h is the length of the hot wall and $\overline{V}$ is the cavity volume.

METHOD OF SOLUTION

The numerical procedure used in this work is based on the Galerkin weighted residual method of finite element formulation. The application of this technique is well described by Taylor and Hood[14] and Dechaumphai[15]. In this method, the solution domain is discretized into finite element meshes, which are composed of non-uniform triangular elements. Then the nonlinear governing partial differential equations i.e., mass, momentum and energy equations are transferred into a system of integral equations by applying Galerkin weighted residual method. Gauss quadrature method performs the integration involved in each term of these equations. The nonlinear algebraic equations thus obtained are modified by imposition of boundary conditions. These modified nonlinear equations are transferred into linear algebraic equations by using Newton's method. Finally, these linear equations are solved by using triangular factorization method.

GRID REFINEMENT CHECK

Five different grid sizes of 3976, 4798, 6158, 6278 and 7724 elements are chosen for the present simulation to test the independency of the results with the grid size variations. Average Nusselt number at the heated surface, average temperature of the fluid inside the cavity and the solution time are monitored at Ri = 1.0, $L_x = L_y = 0.5$, D = 0.2 and K = 5.0 for these grid elements (Table 1). The magnitude of average Nusselt number at the heated surface and average temperature of the fluid inside the cavity for 6278 elements shows a very little difference with the results obtained for the other denser grids. Hence, for the rest of the calculation in this study, a grid size of 6278 elements is chosen for optimum results.

Table 1: Grid Sensitivity Check at Ri = 1.0, K = 5.0, D = 0.2 and $L_x = L_y = 0.5$

Elements	3976	4798	6158	6278	7724
Nu	4.84242	4.84221	4.83259	4.83287	4.83245
Tav	0.19719	0.19720	0.19723	0.19722	0.197223
Time(s)	385.219	493.235	682.985	698.703	927.359

Table 2: Comparison of Average Nusselt Number with House *et al.*[1]

Ra	K	Nu	
		Present study	House *et al.*[1]
0	0.2	0.7071	0.7063
0	1.0	1.0000	1.0000
0	5.0	1.4142	1.4125
10^5	0.2	4.6237	4.6239
10^5	1.0	4.5037	4.5061
10^5	5.0	4.3190	4.3249

CODE VALIDATION

The present code was extensively validated based on the problem of House *et al.*[1] We present here some results obtained by our code in comparison with those reported in House *et al.*[1] for Ra = 0.0 and 10^5 and three values of K = 0.2, 1.0 and 5.0. The physical problem studied by House *et al.*[1] was a vertical square enclosure with sides of length L. The vertical walls were isothermal and differentially heated; where as the bottom and top walls were adiabatic. A square heat conducting body with sides of length equal to L/2 was placed at the center of the enclosure. For the same parameters used by House *et al.*[1]; the comparison of average Nusselt number at the hot wall is shown in Table 2. The present results have an excellent agreement with the results obtained by House *et al.*[1]

RESULT AND DISCUSSION

Mixed convection flow and temperature fields in a vented square cavity filled with a horizontal square solid cylinder are examined. The numerical model developed in the present investigation is used to carry out a number of simulations for the parametric variation of L_x, L_y and Ri. The range of Ri for this investigation is varied from 0 to 5.0 by changing Gr while keeping Re fixed at 100. In this simulation, the values of K and D are assigned 5.0 and 0.2 respectively. Air is chosen as working fluid with Pr = 0.71.

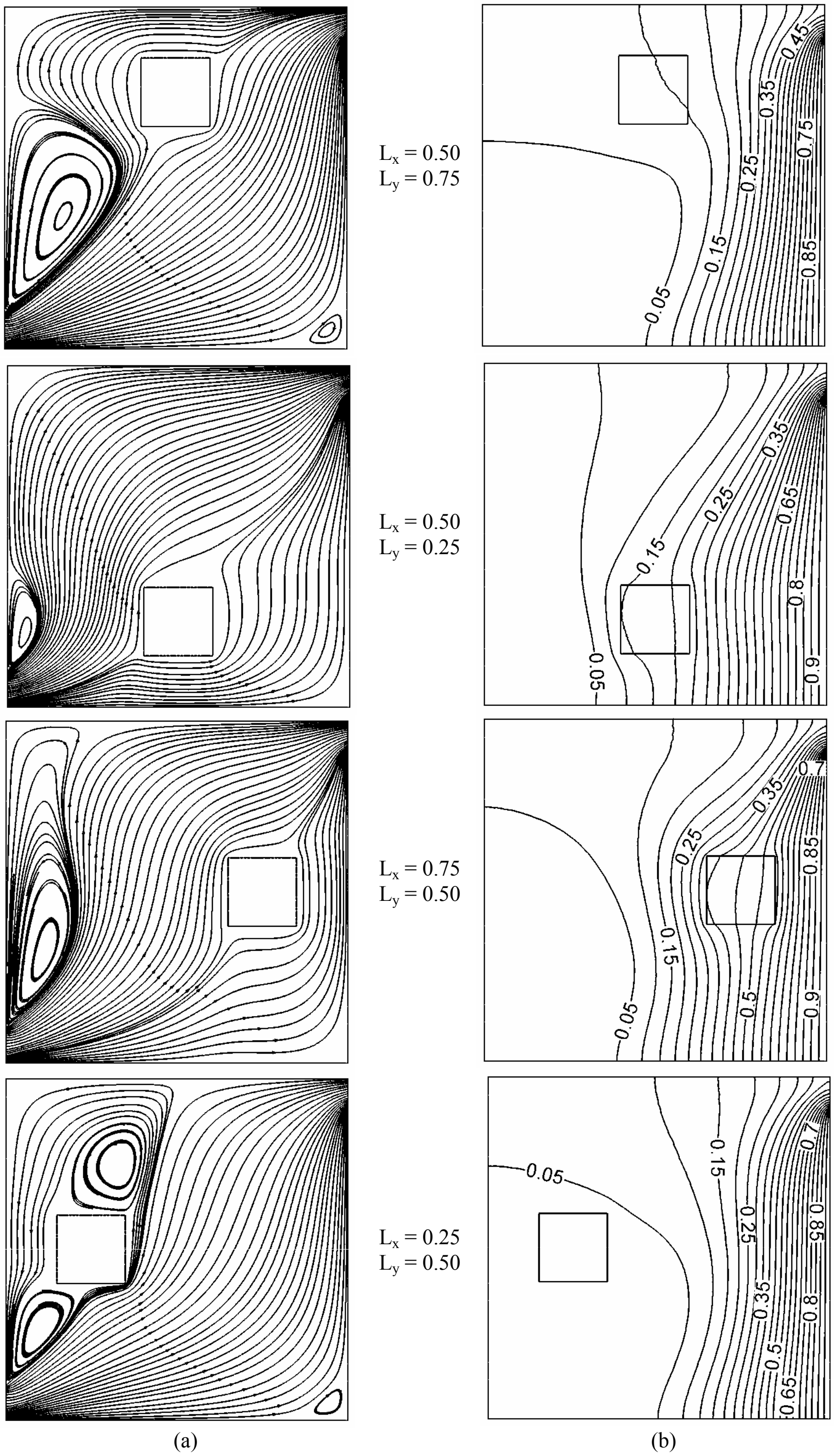
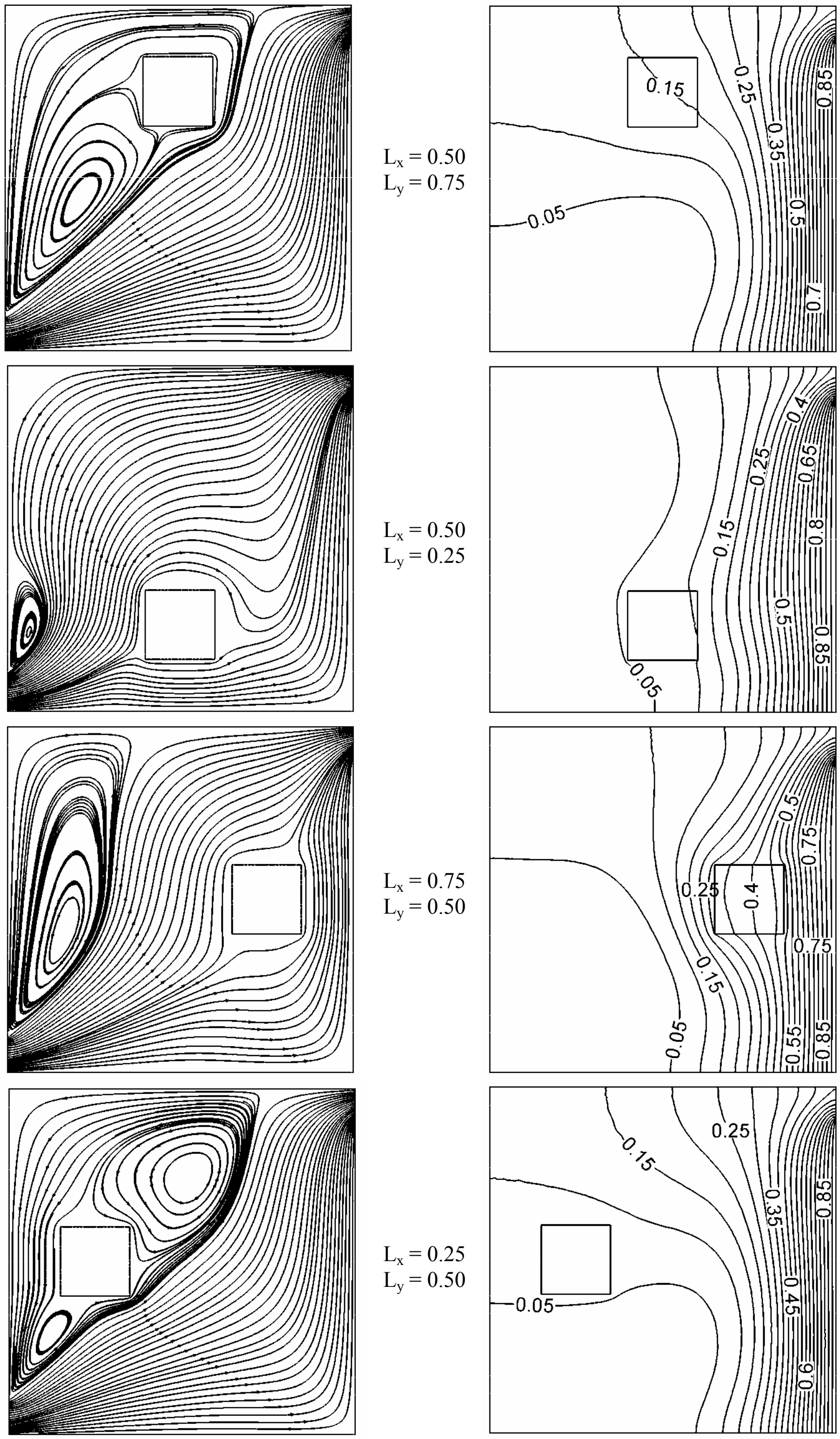


Figure 2: (a) Streamlines and (b) Isotherms for Different Locations of the Cylinder at Ri = 0.0.

Flow and thermal field

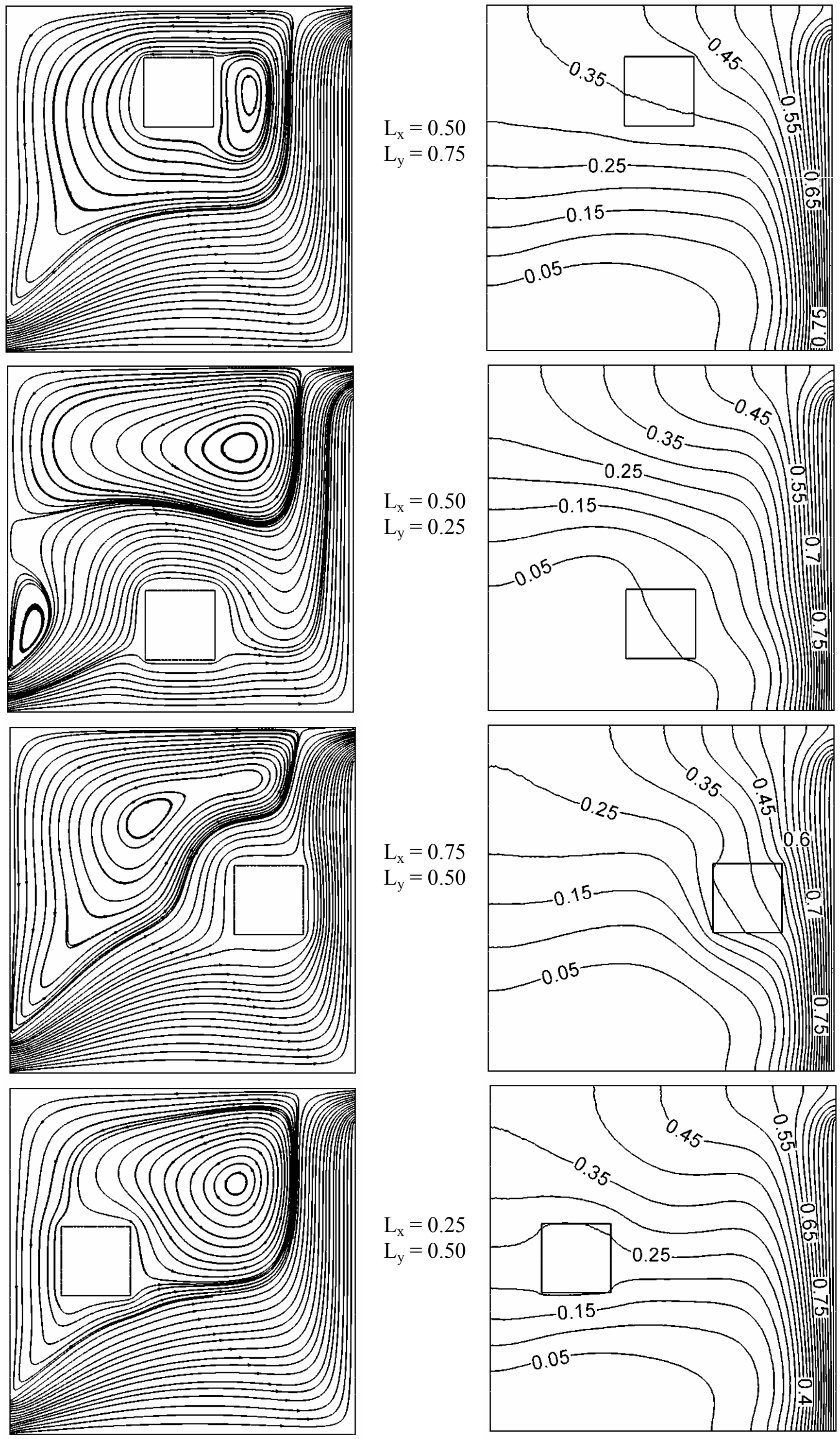
Figures 2-4 show the distribution of streamlines and isothermal lines for various locations of the cylinder at Ri = 0.0, 1.0 and 5.0 in the cavity. Figure 2(a) shows the distribution of streamlines for different locations of the cylinder at Ri = 0.0. When the inner cylinder moves closer to the left wall along the mid-horizontal plane (L_x = 0.25, L_y = 0.50), the major flow is diagonal from the inlet to the



(a) (b)

Figure 3: (a) Streamlines and (b) Isotherms for Different Locations of the Cylinder at Ri = 1.0.

exit and an eddy with two inner vortices is developed near the left top corner of the cavity. Also, a very small vortex is appear at the right bottom corner in the cavity. Further, if the cylinder moves closer to the heated wall along the mid-horizontal plane (L_x = 0.75, L_y = 0.50), the eddy changes its pattern from bi-cellular vortices to a uni-cellular vortex and the small vortex becomes disappears in the cavity. On the other hand, the uni-cellular vortex squeezes and thereby



(a) (b)

Figure 4: (a) Streamlines and (b) Isotherms for Different Locations of the Cylinder at Ri = 5.0.

spreads the induced flow path. When the inner cylinder moves closer to the bottom wall along the mid-vertical plane (L_x = 0.50, L_y = 0.25), the size of the vortex is reduced sharply. As a result, the induced flow is spreads and almost covers the cavity. Moreover, if the inner cylinder moves closer to the top wall along the mid-vertical plane (L_x = 0.50, L_y = 0.75), the uni-cellular vortex near the left wall further spreads and a very small vortex is also appears right bottom corner of the cavity. The distribution of isotherms inside the cavity for the four various locations of the cylinder and fixed Ri = 0.0 is shown in the Figure 2(b). As the inner cylinder moves closer to the left wall along the mid-horizontal plane (L_x = 0.25, L_y = 0.50), the uniformly distributed isotherms around the heat source display that the heat is mainly transported by diffusion due to zero buoyancy force. The isothermal lines surrounding the heat source seem to have no significant difference as the cylinder moves closer to the right wall along the mid-horizontal plane (L_x = 0.75, L_y = 0.50) and closer to the top wall along the mid-vertical plane (L_x = 0.50, L_y = 0.75). In addition, more vertical isotherms near the hot wall generates when the inner cylinder moves closer to the bottom wall along the mid-vertical plane (L_x = 0.50, L_y = 0.25).

However, Figure 3 shows the distribution of streamlines and isotherms for different locations of the cylinder at Ri = 1.0. If we compare these figures with the Figures 2, it is found that as Ri increases from 0.0 to 1.0, the effect of convection on heat transfer becomes larger. As a result, the intensity of the vortices in the cavity increases and the isotherms become nonlinear. Further, the distribution of streamlines and isotherms in the cavity at Ri = 5.0 is significantly different from that at the lower Richardson numbers, because the buoyancy-induced convection becomes more predominant than conduction. Thus the vortex in the cavity spreads and thereby squeezes the induced flow path, and nonlinearity of the isotherms becomes higher and plume formation is profound, indicating the well-established natural convection heat transfer in the cavity.

Heat transfer

Figure 5(i) shows the average Nusselt number (Nu) at the heated surface of the cavity as a function of Richardson numbers and for the four different locations of the cylinder. Nu increases generally with increasing Ri due to the increasing effect of convection. A carefully attention on Figure 5(i) shows that Nu is slightly higher when the inner cylinder moves closer to the top wall at Ri ≤ 1.0, but at Ri > 1.0 it is slightly higher when the inner cylinder moves closer to the bottom wall. Figures 5 (ii-iii) show the average temperature (Θ_{av}) of the fluid and the temperature (Θ_c) at the cylinder center in the cavity as a function of Richardson numbers for the four different locations of the cylinder. From these figures, it is seen that the average temperature of the fluid and the temperature at the cylinder center in the cavity are not monotonic with increasing Ri.

CONCLUSION

A numerical investigation is performed for laminar mixed-convection in a square cavity with a heat conducting horizontal square cylinder. A detailed analysis for the distribution of streamlines, isotherms, average Nusselt number at the hot wall, average temperature of the fluid in the cavity and the centerline temperature at the cylinder is carried out to investigate the effect of the locations of the conducting cylinder on the fluid flow and heat transfer in the square cavity for different Richardson numbers in the range of 0.0 ≤ Ri ≤ 5.0. Cylinder locations have significant effect on the flow and thermal fields. The value of average Nusselt number is the highest in the forced convection dominated area when the cylinder is located near the top wall along the mid-vertical plane and in the free convection dominated area when the cylinder moves closure to the left vertical wall along the mid-horizontal plane. The average temperature of the fluid and the temperature at the cylinder center in the cavity are not monotonic with increasing Ri.

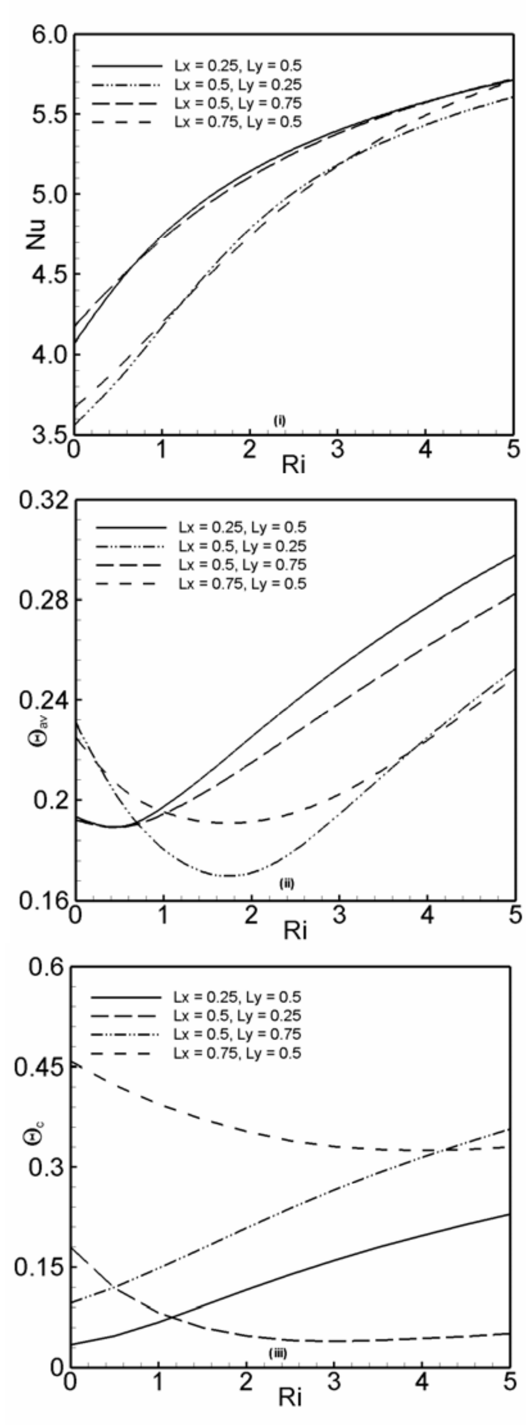


Figure 5: Effect of Various Locations of the Cylinder on (i) Average Nusselt Number, (ii) Average Temperature and (iii) Temperature at the Cylinder Center for Different Richardson Numbers, while K = 5.0 and D = 0.2.

REFERENCES

[1] House, J. M., Beckermann, C., Smith, T. F., 1990, "Effect of a Centered Conducting Body on Natural Convection Heat Transfer in an Enclosure," Numer. Heat Transfer, Part A, Vol. 18, pp. 213-225.

[2] 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," Numer. Heat Transfer, Part A, Vol. 31, pp. 289-304.

[3] Lacroix, M., Joyeux, A., 1995, "Natural Convection Heat Transfer around Heated Cylinders inside a Cavity with Conducting Walls," Numer. Heat Transfer, Part A, Vol. 27, pp. 335-349.

[4] Lacroix, M., Joyeux, A., 1996, "Coupling of Wall Conduction with Natural Convection from Heated Cylinders in a Rectangular Enclosure," Int. Comm. Heat Mass Transfer, Vol. 23(1), pp. 143-151.

[5] Shuja, S.Z., Yilbas, B.S., Iqbal, M.O., 2000, "Mixed Convection in a Square Cavity due to Heat Generating Rectangular Body," Int. J. Numer. Methods Heat Fluid Flow, Vol. 10 (8), pp. 824-841.

[6] Shuja, S.Z., Yilbas, B.S., Iqbal, M.O., 2000, "Heat Transfer Characteristics of Flow past a Rectangular Protruding Body," Numer. Heat Transfer, Part A, Vol. 37, pp. 307-321.

[7] Roychowdhury, D.G, Das, S.K., Sundararajan, T.S., 2002, "Numerical Simulation of Natural Convection Heat Transfer and Fluid Flow around a Heated Cylinder inside an Enclosure," Heat Mass Transfer, Vol. 38, pp. 565-576.

[8] Dong, S. F., Li, Y.T., 2004, "Conjugate of Natural Convection and Conduction in a Complicated Enclosure," Int. J. Heat Mass Transfer, Vol. 47, pp. 2233-2239.

[9] Tasnim, S. H., Collins, M. R., 2005, "Suppressing Natural Convection in a Differentially Heated Square Cavity with an Arc Shaped Baffle," Int. Comm. Heat Mass Transfer, Vol. 32, pp. 94-106.

[10] Braga, E. J., de Lemos, M. J. S., 2005, "Laminar Natural Convection in Cavities Filed with Circular and Square Rods," Int. Comm. Heat Mass Transfer, Vol. 32, pp. 1289-1297.

[11] Das, M. K., Reddy, K. S. K., 2006, "Conjugate Natural Convection Heat Transfer in an Inclined Square Cavity Containing a Conducting Block," Int. J. Heat Mass Transfer, Vol. 49, pp. 4987-5000.

[12] Xu, F., Patterson, J. C., Lei, C., 2006, "Experimental Observations of the Thermal Flow around a Square Obstruction on a Vertical Wall in a Differentially Heated Cavity," Experiments in Fluids, Vol. 40, pp. 364-371.

[13] Zhao, F. Y., Tang, G. F., Lin, D., 2006, "Conjugate Natural Convection in Enclosures with External and Internal Heat Sources," Int. J. Engg. Science, Vol. 44, pp. 148-165.

[14] Taylor, C. Hood, P., 1973, "A Numerical Solution of the Navier-Stokes Equations using Finite Element Technique," Computer and Fluids, Vol. 1, pp.73-89.

[15] Dechaumphai, P., 1999, "Finite Element Method in Engineering," 2nd Edn., Chulalongkorn University Press, Bangkok.