MIXED CONVECTION FLOW IN A RECTANGULAR VENTILATED CAVITY WITH A HEAT CONDUCTING SQUARE CYLINDER AT THE CENTER

Md. Mustafizur Rahman, Md. Elias and M. A. Alim
Department of Mathematics, Bangladesh University of Engineering and Technology, Dhaka, Bangladesh
E-Mail: mmustafizurrahman@math.buet.ac.bd, m71ra@yahoo.com

ABSTRACT

In this paper, a study of mixed convection flow inside a rectangular ventilated cavity in the presence of a heat conducting square cylinder at the center has been carried out. An external fluid flow enters the cavity through an opening in the left vertical wall and exits from another opening in the right vertical wall. The governing equations are transformed into non-dimensional form and the resulting partial differential equations are solved by the finite element method. Results are presented in the form of average Nusselt number of the heated wall, average temperature of the fluid in the cavity and temperature at the cylinder center for the range of Richardson number and cavity aspect ratio. The streamlines and isothermal lines are also presented.

Keywords: Mixed convection, finite element method, Richardson number, rectangular cavity and square cylinder.

INTRODUCTION

Heat transfer in flows in which the influence of forced convection and natural convection are of comparable magnitude (mixed convection flows) occurs frequently in engineering situations. The applications include the design of solar collectors, thermal design of building, air conditioning and cooling of electronic circuit boards etc. Analysis of a mixed convection flow usually requires an understanding of the two limiting regimes. The mixed convection transport is complex due to the interaction of buoyancy force with the shear force.

There have been many investigators in the past on mixed convective flow in ventilated cavities. Papanicolaou and Jaluria (1990) studied mixed convection from an isolated heat sources in a rectangular enclosure. Later on, Papanicolaou and Jaluria (1993) performed computations on mixed convection from a localized heat source in a cavity with conducting walls and two openings for application of electronic equipment cooling. Angirasa (2000) presented a numerical study for mixed convection of airflow in an enclosure having an isothermal vertical wall. Forced conditions were imposed by providing an inlet and a vent in the enclosure. Both, positive and negative temperatures potentials were considered; making the Grashof numbers vary from -10^6 to 10^6. In their study steady-state solutions could not be obtained for higher positive values of the Grashof number and for buoyancy-dominated flows. In general, forced flow enhanced heat transfer for both negative and positive Grashof numbers. Hsu and Wang (2000) studied the mixed convection of micropolar fluids in a square cavity with localized heat source. They indicated that the heat transfer coefficient is lower for a micropolar fluid in comparison with a Newtonian fluid. For mixed convection from a bottom heated open cavity, the stability of convective flow has been analyzed by Leong *et al.* (2005). Their analysis concluded that the transition to the mixed convection regime depends on the relative magnitude of Grashof and Reynolds numbers.

A numerical analysis of laminar mixed convection in an open cavity with a heated wall bounded by a horizontally insulated plate was presented in Manca *et al.* (2003). Three heating modes were considered: assisting flow, opposing flow, and heating from below. Results for Richardson number equal to 0.1 and 100, *Re* = 100 and 1000, and aspect ratio in the range 0.1-1.5 were reported. It was shown that the maximum temperature values decrease as the Reynolds and the Richardson numbers increase. The effect of the ratio of channel height to the cavity height was found to play a significant role on streamline and isotherm patterns for different heating configurations. The investigation showed that opposing forced flow configuration has the highest thermal performance, in terms of both maximum temperature and average Nusselt number. Later, similar problem for the case of assisting forced flow configuration was tested experimentally by Manca *et al.* (2006) and based on the flow visualization results, they pointed out that for *Re* = 1000 there were two nearly distinct fluid motions: a parallel forced flow in the channel and a recirculation flow inside the cavity and for *Re* = 100 the effect of a stronger buoyancy determined a penetration of thermal plume from the heated plate wall into the upper channel. Omri and Nasrallah (1999) and Singh and Sharif (2003) studied mixed convection in an air-cooled cavity with differentially heated vertical isothermal side walls having inlet and exit ports. Several different placement configurations of the inlet and exit ports were investigated. Best configuration was selected analyzing the cooling effectiveness of the cavity which suggested that injecting air through the cold wall was more effective in heat removal and placing inlet near the bottom and exit near the top produce effective cooling.

However, many authors have studied heat transfer in enclosures with heat-conducting body

ARPN Journal of Engineering and Applied Sciences

obstruction, thereby influencing the convective flow phenomenon. Hsu *et al.* (1997) numerically investigated mixed convection in a partially divided rectangular enclosure. They considered the divider as a baffle inside the enclosure with two different orientations and indicated that the average Nusselt number and the dimensionless surface temperature dependent on the locations and height of the baffle. Effect of exit port locations and aspect ratio of the heat generating body on the heat transfer characteristics and irreversibility generation in a square cavity was investigated by Shuja *et al.* (2000). They found that the overall normalized Nusselt number as well as irreversibility was strongly affected by both of the location of exit port and aspect ratios. Ha *et al*. (2002), in which they concluded that the transition of the flow from quasi-steady up to unsteady convection depends on the presence of bodies and aspect ratio effect of the cell. Lee *et al.* (2004) considered the problem of natural convection in a horizontal enclosure with a square body. Natural convection in a horizontal layer of fluid with a periodic array of square cylinder in the interior were conducted by However, in the previous literature the body was considered as a rigid wall but internal heat transfer was not calculated. Few numerical studies taking into account heat transfer in the interior of the body have been conducted over the past couple of decades. One of the systematic numerical investigations of this problem was conducted by House *et al.* (1990), who considered natural convection in a vertical square cavity with heat conducting body, placed on center in order to understand the effect of the heat conducting body on the heat transfer process in the cavity. They showed that for given *Ra* and *Pr* an existence of conducting body with thermal conductivity ratio less than unity leads to heat transfer enhancement.

To the best of the author's knowledge, no attention has been paid to the problem of mixed convection flow and heat transfer in a rectangular ventilated cavity with a heat conducting square cylinder at the center. The objective of the present study is to examine the effect on heat transfer process in a rectangular ventilated cavity with a heat conducting square cylinder at the center. The results are shown in terms of parametric presentations of streamlines and isotherms plots in the cavity. Also the effect of mixed convection parameter *Ri* and cavity aspect ratio (*AR*) on the heat transfer process are analyzed and the results are presented in terms of the average Nusselt number at the heated surface, average temperature of the fluid in the cavity and the temperature at the cylinder center.

MODEL DESCRIPTION

Details of the geometry for the configuration are shown in Figure-1. The model considered here is a rectangular ventilated cavity with a uniform constant temperature T_h, applied on the right vertical wall. The cavity has dimensions of $L \times H$. The other sidewalls including top and bottom of the cavity are assumed to be adiabatic. The inflow opening located on the left vertical wall is arranged as shown in the Figure-1. The outflow opening of the cavity is fixed at the top of the opposite heated wall and the size of the inlet port is the same size as the exit port which is equal to $w = 0.1H$. It is assumed that the incoming flow is at a uniform velocity, u_i 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). The cylinder, having a diameter of H/5, is placed at the center of the rectangular cavity and no slip boundary conditions are assumed on the cylinder surface and all solid boundaries.

MATHEMATICAL FORMULATIONS

Governing equations

Mixed convection is governed by the differential equations expressing conservation of mass, momentum and energy. The present flow is considered steady, laminar, incompressible and two-dimensional. The viscous dissipation term in the energy equation is neglected. The physical properties of the fluid in the flow model are assumed to be constant except the density variations causing a body force term in the momentum equation. The Boussinesq approximation is invoked for the fluid properties to relative density changes to temperature changes, and to couple in this way the temperature field to the flow field. The governing equations for steady mixed convection flow can be expressed in the dimensionless form as

$$\frac{\partial U}{\partial X}+\frac{\partial V}{\partial Y}=0 \quad (1)$$

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

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

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

For heat conducting cylinder, the energy equation is

$$\frac{\partial^2 \Theta_s}{\partial X^2}+\frac{\partial^2 \Theta_s}{\partial Y^2}=0 \quad (5)$$

Where X and Y are the coordinates varying along horizontal and vertical directions respectively, U and V are the velocity components in the X and Y directions respectively, Θ is the dimensionless temperature and P is the dimensionless pressure. The non-dimensional numbers seen in the above, *Re*, *Ri*, *Pr* and *K* are the Reynolds number, Richardson number, Prandtl number and solid fluid thermal conductivity ratio respectively, and are defined as

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

The dimensionless variables in the equations above are defined as follows:

$$X=\frac{x}{H},\ Y=\frac{y}{H},\ U=\frac{u}{u_i},\ V=\frac{v}{u_i},\ P=\frac{p}{\rho u_i^2},\ \Theta=\frac{(T-T_i)}{(T_h-T_i)},$$

$$\Theta_s=\frac{(T_s-T_i)}{(T_h-T_i)}$$

Where ρ, β, v, α and g are the fluid density, coefficient of volumetric expansion, kinematic viscosity, thermal diffusivity, and gravitational acceleration, respectively.

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

At the inlet: $U = 1$, $V = 0$, $\Theta = 0$

At the outlet: $P = 0$

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

At the heated right vertical wall: $\Theta = 1$

At the left, top and bottom walls in the cavity:

$$\left.\frac{\partial\Theta}{\partial X}\right|_{X=0}=\left.\frac{\partial\Theta}{\partial Y}\right|_{Y=1,0}=0$$

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

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

And the bulk average temperature in the cavity is defined as

$$\Theta_{av}=\int\frac{\Theta}{\overline{V}}\,d\overline{V}$$

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

Computational procedure

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 (1973) and Dechaumphai (1999). 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. Using Gauss quadrature method performs the integration involved in each term of these equations. The nonlinear algebraic equations so obtained are modified by imposition of boundary conditions. These modified nonlinear equations are transferred into linear algebraic equations by using Newton's method. Finally, using Triangular Factorization method solves these linear equations.

Grid sensitivity check

Geometry studied in this paper is an obstructed ventilated cavity; therefore several grid size sensitivity tests were conducted in this geometry to determine the sufficiency of the mesh scheme and to ensure that the solutions are grid independent. This is obtained when numerical results of the average Nusselt number Nu, average temperature Θ_{av} and solution time become grid size independent, although we continue the refinement of the mesh grid. Five different non-uniform grids with the following number of nodes and elements were considered for the grid refinement tests: 24545 nodes, 3788 elements; 29321 nodes, 4556 elements; 37787 nodes, 5900 elements; 38163 nodes, 5962 elements and 48030 nodes, 7516 elements as shown in Table-1. From these values, 38163 nodes and 5962 elements have been chosen throughout the simulation to optimize the relation between the accuracy required and the computing time.

Code validation

The present code was extensively validated based on the problem of House *et al.* (1990). We present here some results obtained by our code in comparison with those reported in House *et al.* (1990) for $Ra = 0.0$, 10^5 and two values of $K = 0.2$ and 5.0. The physical problem studied by House *et al.* (1990) 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 in House *et al.* (1990); the average Nusselt number (at the hot wall) comparison is shown in Table-2. The present results have an excellent agreement with the results obtained by House *et al.* (1990).

RESULTS AND DISCUSSIONS

Results are presented for mixed convection inside a rectangular ventilated cavity with a heat conducting square cylinder at the center, where the Ri has been varied from 0.0 to 5.0 keeping at $Re = 100$, $Pr = 0.71$ and $K = 5.0$. The aspect ratio (AR) considered is in the range of 0.5, 1.0, 1.5 and 2.0.

Flow and thermal fields' characteristics

The effect of the cavity aspect ratio on the flow structure and temperature distribution for various values of Ri is shown in Figures 2 to 4. The streamlines and isotherms are presented in Figure-2 for steady state flows obtained for $Ri = 0.0$ and the values of cavity aspect ratio ranging between 0.5 and 2.0. For $AR = 0.5$, the analysis of the streamlines in Figure-2a (iv) shows the bulk induced fluid flows throughout the cavity and the streamlines bifurcates near the cylinder. It is also seen that the streamlines are symmetrical about the line joining the inlet and outlet ports and a small recirculating cell is formed

ARPN Journal of Engineering and Applied Sciences

just at the above the inlet near the left insulated wall. Thus conduction and forced convection effects are dominant. However, at *AR* = 1.0, it is seen that the recirculating cell increases in size, but the open lines are almost similar as compared to the previous case. From the Figures-2a (i, ii) it is clearly seen that the recirculating cell gradually increases and the symmetry of the open lines is destroyed with the increase of the values of *AR*. The corresponding isotherm plots are presented in Figure-2b (i-iv). As expected, the thermal boundary decreases in thickness as *AR* increases. This is reflected by the denser dustering of isotherms close to the hot wall as *AR* increases.

Now for *Ri* = 1.0, it is seen from Figure-3a (i-iv) that the natural convection effect is present but remains relatively weak at the low *AR* = 0.5, since the open lines characterizing the imposed flow are still dominant. On the other hand, for the higher values of *AR* it clearly seen that the recirculating cell near the left wall gradually increases with the increase of *AR* and another recirculating cell is developed just in the front of the outlet port, above the open lines. The corresponding isotherm plots for the above cases are presented in Figures-3b (i-iv). As shown, the thermal boundary layer decreases in thickness slowly as the *AR* increases.

The effect of the cavity aspect ratio on the flow structure and temperature distribution inside the cavity is illustrated in Figure-4 (i-iv) for *Ri* = 5.0 and four selected values of *AR*. Making a comparison of streamlines and isotherms at *Ri* = 5.0 for various values of *AR* with the streamlines at *Ri* = 0.0, 1.0 for various values of *AR*, a significant difference is found.

Heat transfer characteristics

The effect of Richardson number on the average Nusselt number at the hot wall, average temperature of the fluid in the cavity, dimensionless temperature at the cylinder center for values of cavity aspect ratio is shown in the Figure-5. As *Ri* increases, average Nusselt number increases sharply for all values of *AR*. Average Nusselt number (*Nu*) is higher for the lower values of *AR*. On the other hand, the average temperature of the fluid in the cavity and temperature at the cylinder center increases gradually with increasing *Ri*. The average temperature of the fluid in the cavity and temperature at the cylinder center are lower for the higher values of *AR*.

Table-1. Grid sensitivity check at *AR* = 1.0, *Re* = 100, *Ri* = 1.0 and *Pr* = 0.71.

Nodes (elements)	**24545** (3788)	**29321** (4556)	**37787** (5900)	3**8163** (5962)	**48030** (7516)
Nu	4.167817	4.168185	4.168376	4.168394	4.168461
Θ_{av}	0.042974	0.042973	0.042973	0.042973	0.042973
Time(s)	323.610	408.859	563.203	588.390	793.125

Table-2. Comparison of average Nusselt number with House *et al.* (1990).

Ra	***K***	***Nu***	
		Present study	**House *et al*.** (1990)
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

ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

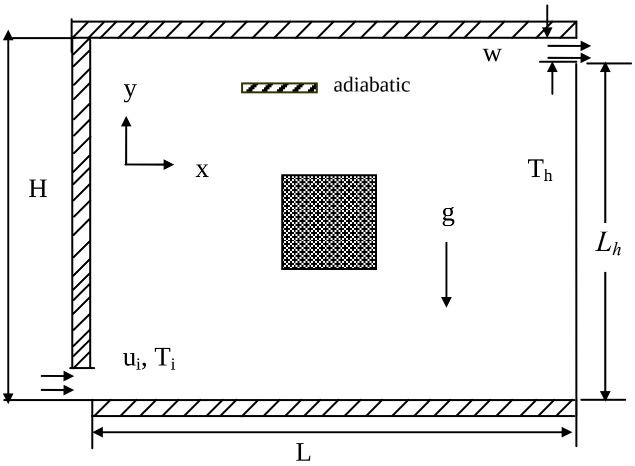
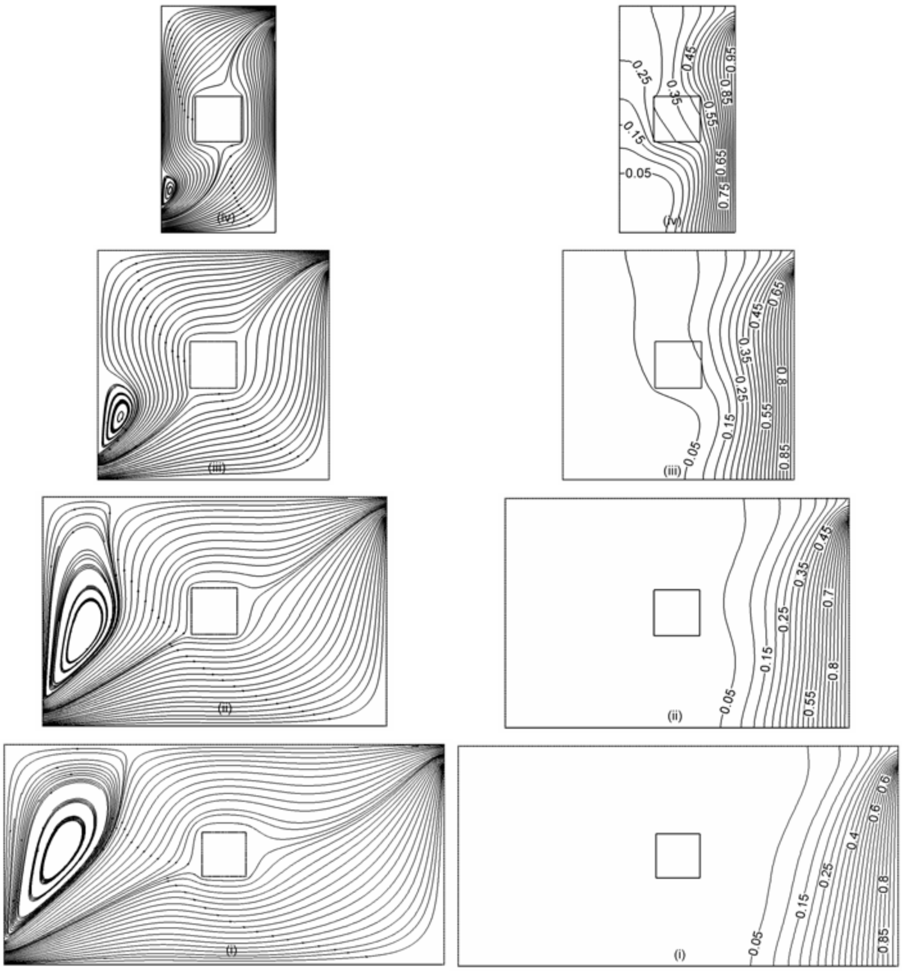


Figure-1. Schematic of the problem with the domain and boundary conditions.



Streamlines Isotherms

Figure-2. (a) Streamlines (left) and (b) Isotherms (right) for (i) AR = 2.0, (ii) AR = 1.5, (iii) AR = 1.0 and (iv) AR = 0.5 at Ri = 0.0.

ARPN Journal of Engineering and Applied Sciences

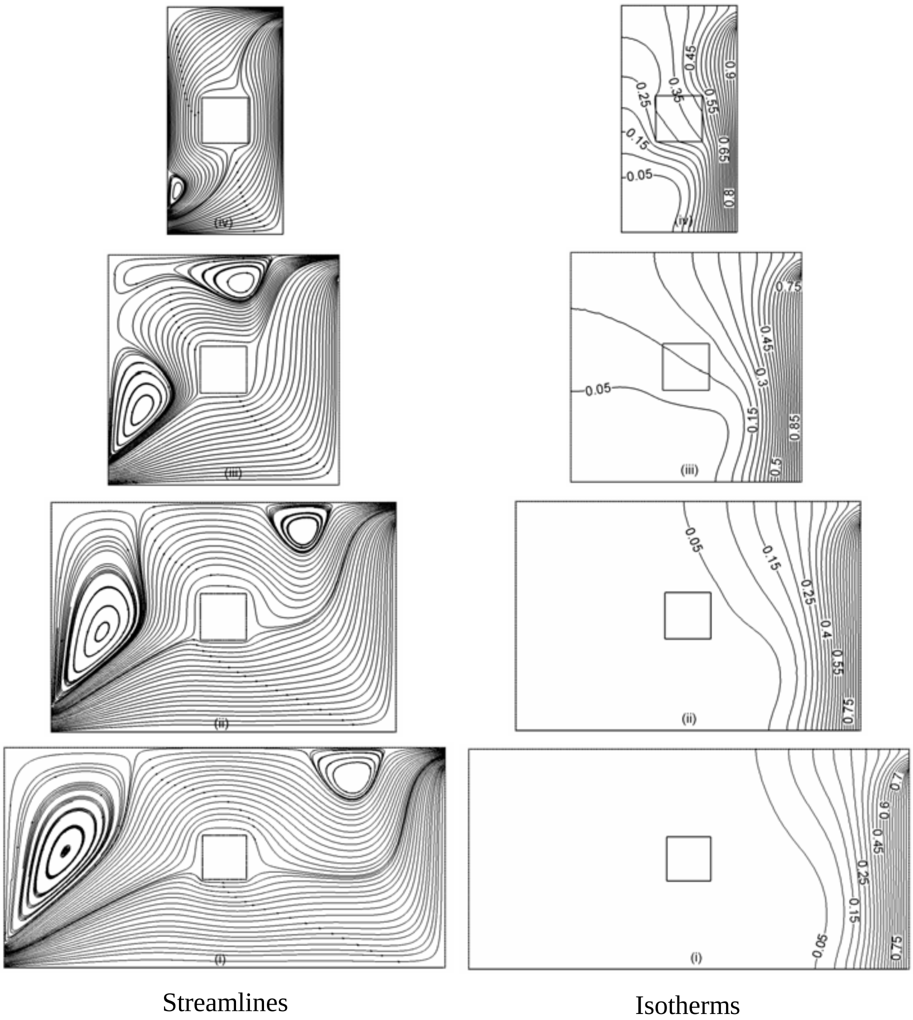
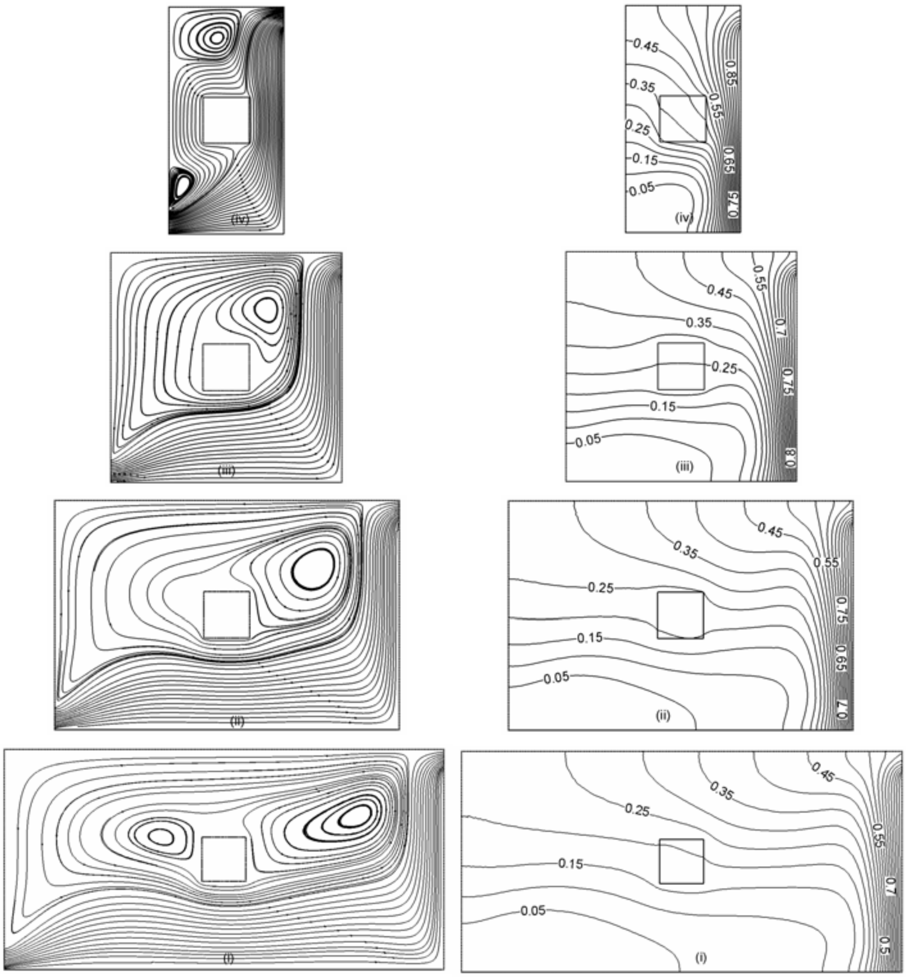


Figure-3. (a) Streamlines (left) and (b) Isotherms (right) for (i) *AR* = 2.0, (ii) *AR* = 1.5, (iii) *AR* = 1.0 and (iv) *AR* = 0.5 at *Ri* = 1.0.



Streamlines Isotherms

Figure-4. (a) Streamlines (left) and (b) Isotherms (right) for (i) AR = 2.0, (ii) AR = 1.5, (iii) AR = 1.0 and (iv) AR = 0.5 at Ri = 5.0.

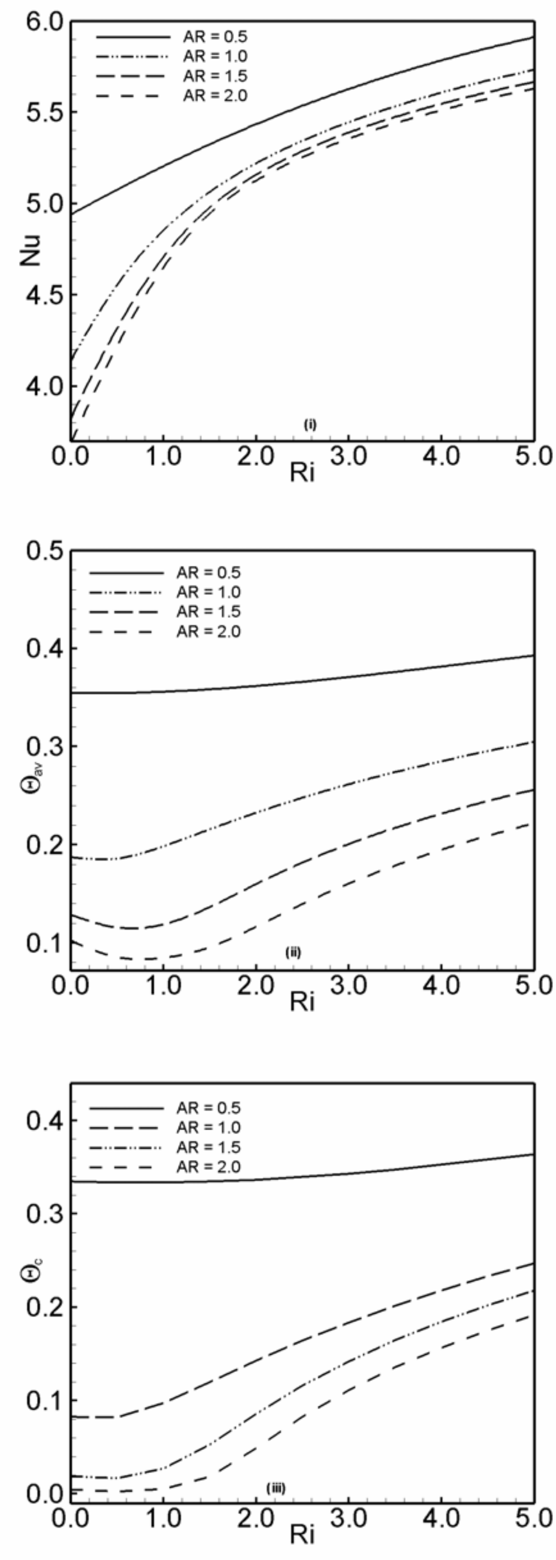


Figure-5. Effect of cavity aspect ratio on (i) average Nusselt number, (ii) average temperature and (iii) temperature at the cylinder center for different *Ri*.

CONCLUSIONS

A numerical investigation on mixed convection in a rectangular ventilated cavity with a heat conducting square cylinder at the center was carried out using a finite element method. The present study examined and explained the complex interaction between buoyancy and force flow in a ventilated rectangular cavity. The results of the study show that the flow structure and temperature distribution are considerably influenced by the interaction between natural convection and forced convection in the cavity as well as the cavity aspect ratio. The average Nusselt number at the heated surface is the highest for the lowest value of *AR*, but the average temperature of the fluid in the cavity and temperature at the cylinder center are the lowest for the highest value of *AR*.

Nomenclature

AR	cavity aspect ratio
g	gravitational acceleration (ms^{-2})
Gr	Grashof number
h	convective heat transfer coefficient
H	Height of the cavity (m)
w	height of the inflow and outflow openings (m)
k	thermal conductivity of the fluid ($Wm^{-1}K^{-1}$)
k_s	thermal conductivity of the solid square cylinder ($Wm^{-1}k^{-1}$)
K	Solid fluid thermal conductivity ratio
L	length of the cavity (m)
L_h	length of the heated wall
Nu	Average Nusselt number
N	non-dimensional distance
p	pressure (Nm^{-2})
P	non-dimensional pressure
Pr	Prandtl number
Ra	Rayleigh number
Re	Reynolds number
Ri	Richardson number
T	temperature (K)
u, v	velocity components (ms^{-1})
U, V	non-dimensional velocity components
$\overline{V}$	cavity volume (m^3)
x, y	Cartesian coordinates (m)
X, Y	non-dimensional Cartesian coordinates

Greek symbols

α	thermal diffusivity (m^2s^{-1})
Θ	non-dimensional temperature
β	thermal expansion coefficient (K^{-1})
ρ	density of the fluid (kgm^{-3})
υ	kinematic viscosity of the fluid (m^2s^{-1})

Subscripts

av	average
h	heated surface
i	inlet state

ACKNOWLEDGEMENTS

The authors wish to acknowledge Department of Mathematics, Bangladesh University of Engineering and Technology, Dhaka, Bangladesh, for support and technical help throughout this work.

REFERENCES

Papanicolaou E. and Jaluria Y. 1990. Mixed Convection from an Isolated Heat Source in a Rectangular Enclosure. Numerical Heat Transfer, Part A. 18: 427-461.

Papanicolaou E. and Jaluria Y. 1993. Mixed Convection from a Localized Heat Source in a Cavity with Conducting Walls: A Numerical Study. Numerical Heat Transfer. Part A 23: 463-484.

Angirasa D. 2000. Mixed convection in a vented enclosure with an isothermal vertical surface. Fluid Dynamics Research. 26: 219-223.

Hsu T-H. And Wang, S-G. 2000. Mixed convection of micropolar fluids in a cavity. International Journal of Heat and Mass Transfer. 43: 1563-1572.

Leong J.C., Brown, N.M., and Lai F.C. 2005. Mixed convection from an open cavity in a horizontal channel. International Communications in Heat and Mass Transfer. 32: 583-592.

Manca O., Nardini, S., Khanafer, K., Vafai K. 2003. Effect of Heated Wall Position on Mixed Convection in a Channel with an Open Cavity. Numerical Heat Transfer, Part A. 43: 259-282.

Manca O., Nardini, S., and Vafai K. 2006. Experimental investigation of mixed convection in a channel with an open cavity. Experimental Heat Transfer. 19: 53-68.

Omri A. and Nasrallah S. B. 1999. Control Volume Finite Element Numerical Simulation of Mixed Convection in an Air-Cooled Cavity. Numerical Heat Transfer, Part A. 36: 615-637.

Singh S. and Sharif M. A. R. 2003. Mixed Convective Cooling of a Rectangular Cavity with Inlet and Exit Openings on Differentially Heated Side Walls. Numerical Heat Transfer, Part A. 44: 233-253.

Hsu, T.H., Hsu, P.T. and How S.P. 1997. Mixed Convection in a Partially Divided Rectangular Enclosure. Numerical Heat Transfer, Part A. 31: 655-683.

Shuja S.Z., Yilbas, B.S., and Iqbal M.O. 2000. Mixed convection in a square cavity due to heat generating rectangular body: effect of cavity exit port locations, International Journal of Numerical Methods for Heat and Fluid Flow. 10(8): 824-841.

Ha M. Y. Yoon, H.S, Yoon, K.S, Balachandar, S., Kim, I, Lee, J.R. Chun H.H. 2002. Two-dimensional and unsteady natural convection in a horizontal enclosure with a square body. Numerical Heat Transfer A. 41: 183-210.

Lee J. R., Ha, M. Y. Balachandar, S., Yoon, H.S. and Lee S.S. 2004. Natural Convection in a Horizontal Layer of Fluid with a Periodic Array of Square Cylinders in the Interior. Phys. Fluids. 16: 1097-1117.

House J. M., Beckermann, C. and Smith T. F. 1990. Effect of a Centered Conducting Body on Natural Convection Heat Transfer in an Enclosure. Numerical Heat Transfer, Part A. 18: 213-225.

Taylor C. and Hood P. 1973. A numerical solution of the Navier-Stokes equations using finite element technique. Computer and Fluids. 1(1): 73-89.

Dechaumphai P. 1999. Finite Element Method in Engineering. 2nd Ed. Chulalongkorn University Press, Bangkok.