

## EFFECT OF A HEATED HOLLOW CYLINDER ON COMBINED FREE AND FORCED CONVECTION IN A VENTILATED CAVITY

M. M. Rahman, M. A. Alim and M. A. H. Khan

Department of Mathematics, Bangladesh University of Engineering and Technology (BUET),  
Dhaka, Bangladesh

### ABSTRACT

Effect of a heated hollow cylinder on combined free and forced convection in a ventilated cavity is studied numerically. The wall of the cavity is assumed to be adiabatic. Flows are imposed at the bottom of the left wall and exited at the top of the right wall of the cavity. The heated cylinder is placed at the center of the cavity. The present study simulates a practical system such as an air-cooled electronic equipment with a heat component or an oven with heater. Emphasis is sited on the influences of the cylinder diameter and thermal conductivity of the cylinder in the cavity. The consequent mathematical model is governed by the coupled equations of mass, momentum and energy and is solved by employing Galerkin weighted residual method of finite element formulation. A wide range of pertinent parameters such as Reynolds number, Richardson number, cylinder diameter and the solid-fluid thermal conductivity ratio are considered in the present study. Various results such as the streamlines, isotherms, heat transfer rates in terms of the average Nusselt number and average fluid temperature in the cavity are presented for different parameter. It is observed that the cylinder diameter has significant effect on both the flow and thermal fields but the solid-fluid thermal conductivity ratio has insignificant effect on the flow field.

**Keywords:** Hollow Cylinder, Ventilated Cavity, Combined Free And Forced Convection.

### 1. INTRODUCTION

Convection in enclosures containing blocks has gained recent research significance as a means of heat transfer enhancement. One of the systematic numerical investigations of this problem was conducted by House et al. [1], the authors 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 found that the heat transfer across the enclosure enhanced by a body with thermal conductivity ratio less than unity. Lacroix [2] 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 [3] conducted a numerical study of natural convection heat transfer from two horizontal heated cylinders confined to a rectangular enclosure having finite wall conductance's. They indicated that wall heat conduction reduces the average temperature differences across the cavity, partially stabilizes the flow and decreases natural convection heat transfer around the cylinders. Ha et al. [4] conducted a comprehensive numerical study to investigate the transient heat transfer and flow characteristics of the natural convection of three different fluids in a vertical square enclosure within which a centered, square, heat conducting body generates heat. Later on, Ha and Jung [5] conducted a comprehensive

numerical study to investigate three dimensional steady conjugate heat transfers of natural convection and conduction in a differentially heated in a vertical cubic enclosure within which a centered, cubic, heat-generating cubic conducting body. They concluded that for the presence of a conducting body in the enclosure, a larger variation of the local Nusselt number at the hot and cold walls in the z-direction is seen. Roychowdhury et al. [6] analyzed the natural convective flow and heat transfer features for a heated cylinder kept in a square enclosure with different thermal boundary conditions. Dong and Li [7] studied conjugate of natural convection and conduction in a complicated enclosure. They investigated the influences of material character, geometrical shape and Rayleigh number on the heat transfer in overall concerned region and concluded that the flow and heat transfer increase with the increase of thermal conductivity in the solid region; both geometric shape and Rayleigh number affect the overall flow and heat transfer greatly. Buoyancy induced flow and heat transfer inside a square cavity due to a thin baffle on the hot wall was analyzed numerically in Tasnim and Collins [8]. They investigated the effects of baffle height, length and Rayleigh number on heat transfer performance. It was found that adding baffle on the hot wall increased the rate of heat transfer. Braga and Lemos [9] numerically studied steady laminar natural convection within a square cavity filled with a fixed amount of conducting solid

material consisting of either circular or square obstacles. They showed that the average Nusselt number for cylindrical rods is slightly lower than those for square rods. Bhave et al. [10] were investigated the effect on the steady-state natural convection heat transfer enhancement of a centrally-placed adiabatic block within a differentially heated square cavity with a fixed temperature drop between the vertical walls. Kumar and Dalal [11] studied natural convection around a tilted heated square cylinder kept in an enclosure in the range of  $103 \leq Ra \leq 106$ . They reported detailed flow and heat transfer features for two different thermal boundary conditions and found that the uniform wall temperature heating is quantitatively different from the uniform wall heat flux heating.

The studies of mixed convection in a partially divided rectangular enclosure were respectively carried out by Hsu et al. [12], How and Hsu [13]. The simulation was conducted for wide range of Reynolds and Grashof numbers. They indicated that the average Nusselt number and the dimensionless surface temperature depended on the location and height of the divider. Combined free and forced convection in a square enclosure with heat conducting body and a finite-size heat source was simulated numerically by Hsu and How [14]. They concluded that both the heat transfer coefficient and the dimensionless temperature in the body center strongly depend on the configurations of the system. Recently Rahman et al. [15] studied of mixed convection in a square cavity with a heat conducting square cylinder at different locations. At the same time Rahman et al. [16] studied mixed convection in a vented square cavity with a heat conducting horizontal solid circular cylinder. Very recently Rahman et al. [17] analyzed mixed convection in a rectangular cavity with a heat conducting horizontal circular cylinder by using finite element method.

Most of the previous studies were done on natural convection in a closed cavity with a heat conducting body. There has been little study on mixed convection in an obstructed vented cavity. In the present study, a numerical simulation of flow and temperature fields in a square cavity with a heated hollow cylinder is carried out. We investigate the flow and thermal characteristics of the system by observing variations in streamlines and isotherms for different values of the cylinder diameter and solid fluid thermal conductivity ratio at the three convective regimes. We also investigate the heat transfer characteristics by calculating the average Nusselt number on the hot surface.

## 2. MODEL SPECIFICATION

The schematic of the system considered in this paper is shown in Fig. 1. The system consists of a square cavity with sides of length  $L$ , within which a heated hollow cylinder with diameter of  $d$  and thermal conductivity of  $k_s$  is centered. The sidewalls of the cavity are assumed to be adiabatic. It is assumed that the incoming flow is at a uniform velocity,  $u_i$ , and at the ambient temperature,  $T_i$ . An inflow opening located on the bottom of the left vertical wall, whereas the out flow opening at the top of the opposite side wall and the size of the inlet port is the

same size as the exit port which is equal to  $w = 0.1L$ . The outgoing flow is assumed to have zero diffusion flux for all variables i.e. convective boundary conditions (CBC). All solid boundaries are assumed to be rigid no-slip walls.

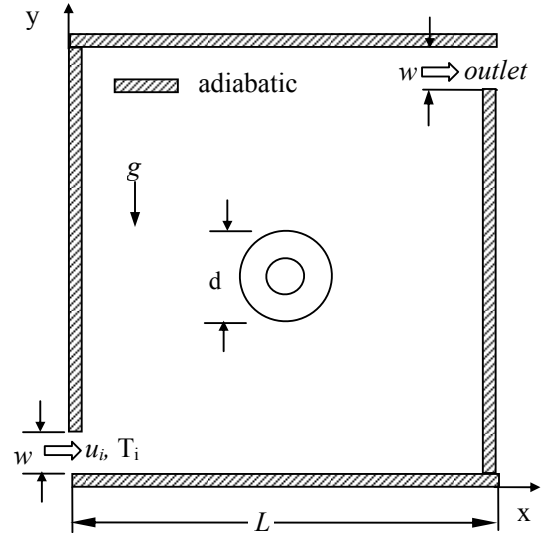


Fig 1. Schematic diagram of the problem considered and coordinate system

## 3. MATHEMATICAL FORMULATION

The equations describing the problem under consideration are based on the laws of mass, momentum and energy. All physical properties of are assumed to be constant except the density variation in body force term of the  $v$ -momentum equation according to the Boussinesq approximation. The flow within the cavity is assumed to be steady, laminar and two-dimensional incompressible. The radiation effects, viscous dissipation and pressure work are taken as negligible. Taking into consideration the above mentioned assumptions the equations can be written in dimensionless form as follows:

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

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

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

$$U \frac{\partial \theta}{\partial X} + V \frac{\partial \theta}{\partial Y} = \frac{1}{RePr} \left( \frac{\partial^2 \theta}{\partial X^2} + \frac{\partial^2 \theta}{\partial Y^2} \right) \quad (4)$$

For solid cylinder the energy equation is

$$0 = \frac{K}{RePr} \left( \frac{\partial^2 \theta_s}{\partial X^2} + \frac{\partial^2 \theta_s}{\partial Y^2} \right) \quad (5)$$

Here the Reynolds number, Grashof number, cylinder diameter, Prandtl number, Richardson number and solid fluid thermal conductivity ratio are given respectively by Eq. (6).

$$Re = u_i L / \nu, Gr = g \beta \Delta T L^3 / \nu^2, D = d / L \quad (6)$$

$$Pr = \nu / \alpha, Ri = Gr / Re^2 \text{ and } K = k_s / k$$

Using the following variables the above equations were non dimensionalized

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

$$\theta = \frac{(T - T_i)}{(T_h - T_i)}, \theta_s = \frac{(T_s - T_i)}{(T_h - T_i)}$$

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,  $\theta$  is the dimensionless temperature and P is the dimensionless pressure.

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

At the Inlet:  $U = 1, V = 0, \theta = -0.5$

At the outlet: Convective boundary condition  $P = 0$

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

At the cavity walls:  $\frac{\partial \theta}{\partial N} = 0$

At the inner surface of the cylinder:  $\theta = 1.0$

At the outer surface of the cylinder:  $\left(\frac{\partial \theta}{\partial N}\right)_{fluid} = K \left(\frac{\partial \theta_s}{\partial N}\right)_{solid}$

Where  $N$  is the non-dimensional distances either  $X$  or  $Y$  direction acting normal to the surface.

The average Nusselt number at the heated surface is calculated as

$$Nu = -\frac{1}{L_h} \int_0^{L_h} \frac{\partial \theta}{\partial N} dS$$

and the average temperature of the fluid is defined as

$$\theta_{av} = \int \theta d\bar{V} / \bar{V}$$

where  $L_h$  is the surface area of the of the heated wall and  $\bar{V}$  is the cavity volume.

#### 4. COMPUTATIONAL PROCEDURE

The solution of the governing equations along with boundary conditions are solved through the Galerkin finite element formulation. The continuum domain is divided into a set of non-overlapping regions called elements. Six node triangular elements with quadratic interpolation functions for velocity as well as temperature and linear interpolation functions for pressure are utilized to discretize the physical domain. Moreover, interpolation functions in terms of local normalized element coordinates are employed to approximate the dependent variables within each element. Substitution of the obtained approximations into the system of the governing equations and boundary conditions yields a residual for each of the conservation equations. These residuals are reduced to zero in a weighted sense over each element volume using the Galerkin method. More details are available in and Rahman *et al.* [17] and Zienkiewicz Taylor [18]

Grid independency test is important for this study due to complexity of the computational domain. We made

several test on Grid independency by using the following grid dimensions: 30049 nodes, 4738 elements; 37248 nodes, 5876 elements; 38821 nodes, 6118 elements and 48495 nodes, 7634 elements. The results are obtained for average Nusselt numbers and average fluid temperature at  $Re = 100, D = 0.2, Ri = 1.0, K = 5.0$  and  $Pr = 0.71$  and listed in Table 1. The table shows that average Nusselt numbers and average fluid temperature do not change significantly with grid dimensions. Based on the results from the table 38821 nodes and 6118 elements can be chosen throughout the simulation.

Table 1: Grid Sensitivity Check at  $Re = 100, D = 0.2, Ri = 1.0, K = 5.0$  and  $Pr = 0.71$

Nodes (elements)	30049 (4738)	37248 (5876)	38821 (6118)	48495 (7634)
$Nu$	0.736641	0.737751	0.745781	0.754781
$\theta_{av}$	0.088298	0.087330	0.087296	0.086298
Time(sec)	408.859	563.203	588.390	793.125

For the purpose of code validation, the natural convection problem in an enclosure with a square heat conducting body in the center of the enclosure was tested for Rayleigh number,  $Ra = 0.0, 10^5$  and two values of  $K = 0.2$  and  $5.0$ . The calculated average Nusselt numbers at the hot wall for the test cases were compared with the values calculated by House *et al.* [1]. The calculated average Nusselt numbers as shown in the Table 2 have an excellent agreement with the results obtained by House *et al* [1].

Table 2: Nusselt Number Comparison for  $Pr = 0.71$

$Ra$	$K$	$Nu$		
		Present work	House <i>et al.</i> [1]	Error (%)
0	0.2	0.7071	0.7063	0.11
0	1.0	1.0000	1.0000	0.00
0	5.0	1.4142	1.4125	0.12
$10^5$	0.2	4.6237	4.6239	0.00
$10^5$	1.0	4.5037	4.5061	0.00
$10^5$	5.0	4.3190	4.3249	0.14

#### 5. RESULTS AND DISCUSSION

The effect of a heated hollow cylinder on mixed convection flow in a ventilated square cavity is tested using a numerical technique. Inspection of the foregoing analysis indicates that the flow and heat transfer characteristics in the present study depend on 5 parameters. These are the Reynolds number  $Re$ , Richardson number  $Ri$ , Prandtl number  $Pr$ , solid fluid thermal conductivity ratio  $K$  and dimensionless cylinder diameter  $D$ . Since a number of governing dimensionless parameters are required to characterize the system, a comprehensive analysis of all combinations of parameters is not practical. So the objective here is to present the effects of  $D, K$  and  $Ri$  on the convective heat transfer in the cavity. In particular, air ( $Pr = 0.71$ ) flowing through the cavity with  $Re = 100$ .

The effect of cylinder diameter on the flow field at  $K = 5.0$  and three different values of  $Ri$  is displayed in fig. 2. The flow structure in the absence of free convection effect ( $Ri = 0.0$ ) and for the four different values of  $D$  is shown in the left column of the fig.2. Now at  $Ri = 0.0$  and  $D = 0.2$ , it is seen that a comparatively small uni-cellular vortex appears just at the top of the inlet in the cavity, due to the effect of buoyancy driven flow. Further with the increase of  $D$  at fixed  $Ri$  ( $Ri = 0.0$ ) the size of the vortex decreases. This is due to increasing the size of the cylinder which gives rise to a decrease in the space available for the flow induced by the heat source. Next for  $Ri = 1.0$  and the different values of  $D$  ( $D = 0.2, 0.3, 0.4$  and  $0.5$ ), it is clearly seen from the figure that the natural convection effect is present, but remains relatively weak at the higher values of  $D$ , since the open lines characterizing the imposed flow are still dominant. As  $Ri$  increase from 1.0 to 5.0, the size of the vortex increases sharply and another vortex is appear just at the below of the outlet port for the lower values of  $D$  ( $D = 0.2$  and  $0.3$ ). On the other hand, the uni-cellular vortex become two cellular for the higher values of  $D$  ( $D = 0.4$  and  $0.5$ ). The effect of cylinder diameter on the thermal field at  $K = 5.0$  and three different values of  $Ri$  is displayed in fig. 3. From these figures it is seen that the size of the cylinder influences the shapes of the isotherms at the three convective regimes. A thermal boundary layer near the bottom of the cylinder is found for the selected values of  $D$  and  $Ri$ . However, a careful observation indicates that the thermal boundary layer on the bottom part of the cylinder become thinner with increasing the values of  $D$  at the three convective regimes. Also, a plume shape isotherms are found at the top of the cylinder.

The effect of the solid-fluid thermal conductivity ratio  $K$  on streamlines at  $D = 0.2$  and various  $Ri$  ( $= 0.0, 1.0$  and  $5.0$ ) are presented in fig. 4. At low  $Ri$  ( $Ri = 0.0$ ) and relatively small values of  $K$  ( $K = 0.2$ ), a small recirculating cell is located just at the top of the inlet port of the cavity. The formation of circulation cell is because of the mixing of the fluid due to the buoyancy driven and convective currents. From the left column of fig. 4, it is also be seen that the streamlines for the different values of  $K$  ( $K = 0.2, 1.0, 5.0, 10.0$ ) at  $Ri = 0.0$  are almost identical. This is because thermal conductivity ratios have insignificant influence on velocity distribution. When  $Ri$  increases from 0.0 to 1.0, i.e. the natural convection effect is comparable with forced convection effect, then the size of the recirculating cell increases, compared with that for  $Ri = 0.0$  and different values of  $K$ . Further when  $Ri$  increases to 5.0, the effect of natural convection is far more compared to the forced convection effect. In this case, conditions are strongly favoring the phenomena of natural convection and significant increase in recirculating cell is found. The effect of the solid-fluid thermal conductivity ratio  $K$  on isotherms for various  $D$  ( $= 0.2, 0.3, 0.4$  and  $0.5$ ) and  $Ri$  ( $= 0.0, 1.0$  and  $5.0$ ) is presented in fig. 5. From the fig. 5, it is clearly seen that the thermal conductivity of the inner cylinder affects strongly on the isotherm structures in the cavity. Now at different values of  $Ri$  ( $= 0.0, 1.0$  and  $5.0$ ) and large  $K$  ( $K = 10.0$ ), the isotherms move out of the inner cylinder and a thermal boundary layer is formed

near the bottom part of the cylinder. On the other hand, plume shape isotherms are observed at the top of the cylinder. When we compare the results for the distribution of isotherms for the case of  $K = 10.0$  with those for  $K = 5.0$  and  $1.0$ , the distribution of isotherms for  $K = 10.0$  is generally similar to that for  $K = 5.0$  and  $1.0$ , for all the Richardson number considered, except slight difference around the solid body due to the difference in the thermal conductivity of the solid. However, at small  $K$  ( $K = 0.2$ ) various values of  $Ri$ , the isothermal lines are crowded in the cylinder, due to lower thermal conductivity of the solid. Moreover, the isotherms are qualitatively similar for different Richardson number considered.

The effects of cylinder diameters on average Nusselt number  $Nu$  at the heated surface and average temperature  $\theta_{av}$  of the fluid in the cavity as a function of  $Ri$  are shown in fig.6. From these figures it is seen that the values of average Nusselt number  $Nu$  decreases monotonically with increasing  $Ri$  for the higher values of  $D$ , but the values of  $Nu$  decreases in  $Ri \leq 0.5$  and beyond these values of  $Ri$  it is increases with  $Ri$ . On the other hand, average Nusselt number  $Nu$  is always highest for  $D = 0.5$ . Average temperature  $\theta_{av}$  of the fluid in the cavity increases slowly with increasing  $Ri$  for all values of  $D$ .

The effect of thermal conductivity ratio  $K$  on average Nusselt number  $Nu$  at the heated surface and average temperature  $\theta_{av}$  of the fluid in the cavity with  $D = 0.2$  is shown in fig. 7. From these figures it is clearly observed that as  $Ri$  increases, average Nusselt number  $Nu$  at the hot surface slowly increases for all values of  $K$ . Maximum average Nusselt number is always found for the higher values of  $K$ . However, as  $Ri$  increases average temperature  $\theta_{av}$  of the fluid decreases slowly in the forced convection dominated region and increases sharply in the free convection dominated region with increasing  $Ri$  for all values of  $K$  in the cavity. On the other hand, the average temperature  $\theta_{av}$  of the fluid in the cavity is the lowest for the highest values of  $K$ .

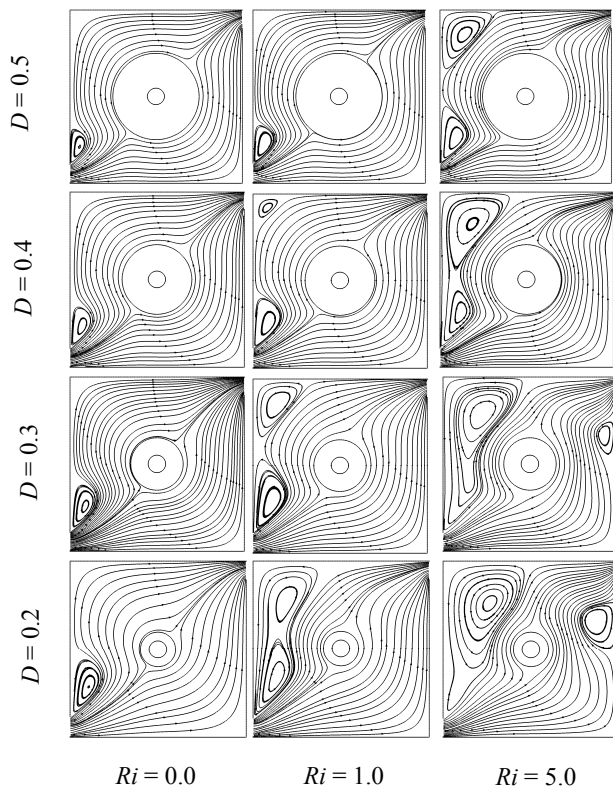


Fig 2. Streamlines for different values of cylinder diameter  $D$  and Richardson number  $Ri$  while  $K = 5.0$ .

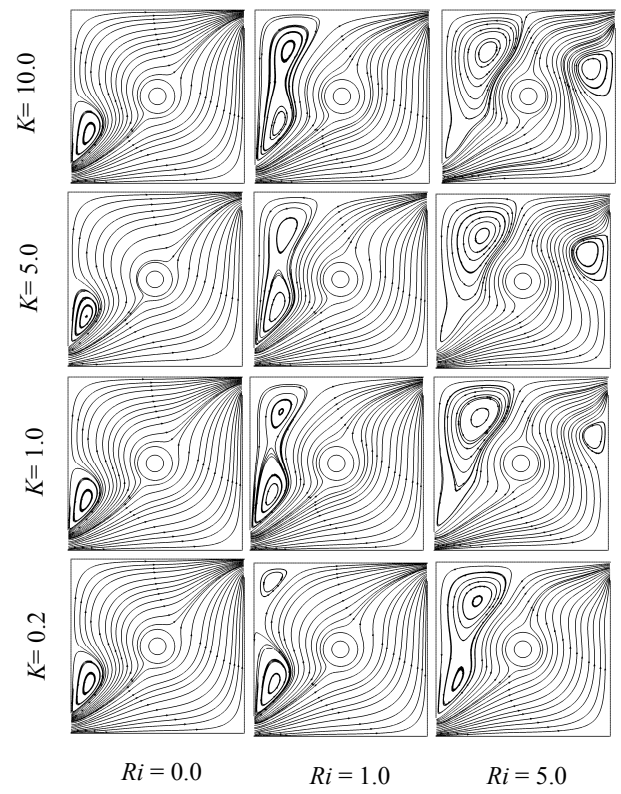


Fig 4. Isotherms for different values of solid fluid thermal conductivity ratio  $D$  and Richardson number  $Ri$  while  $D = 0.2$

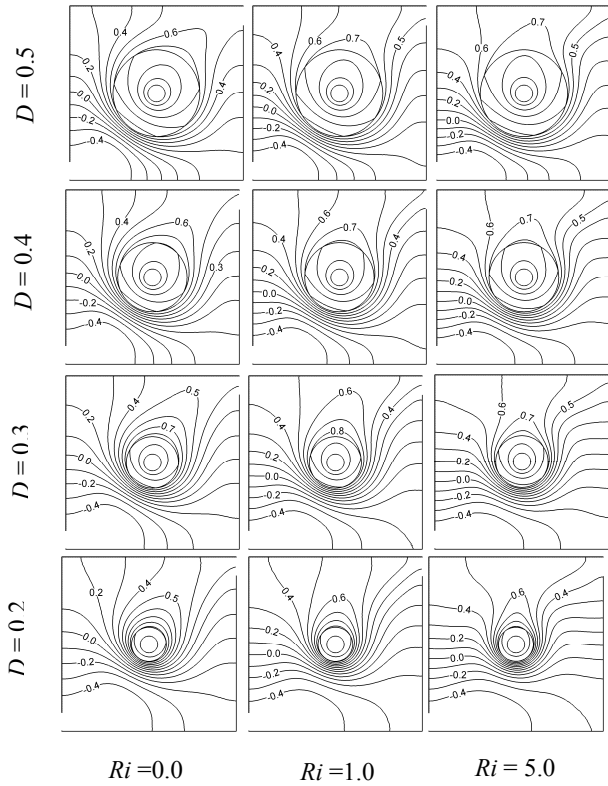


Fig 3. Isotherms for different values of cylinder diameter  $D$  and Richardson number  $Ri$  while  $K = 5.0$ .

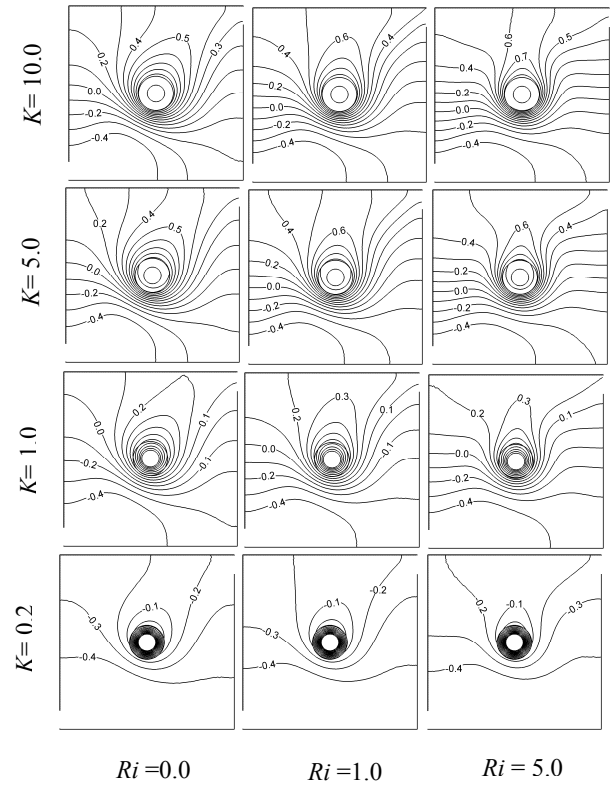


Fig 5. Isotherms for different values of solid fluid thermal conductivity ratio  $D$  and Richardson number  $Ri$  while  $D = 0.2$ .

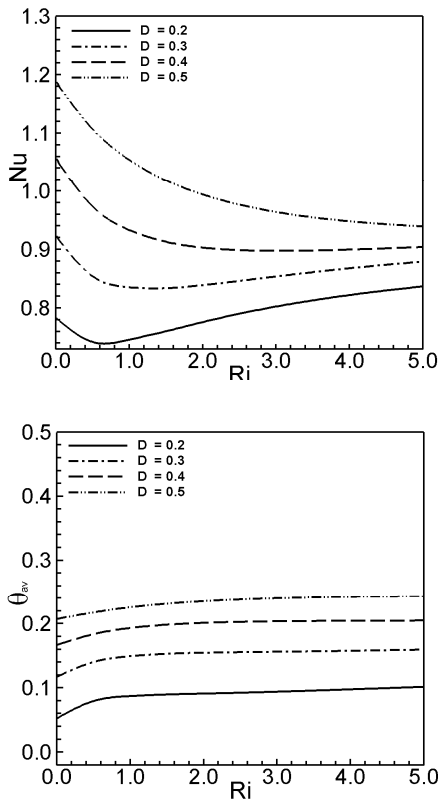


Fig 6. Effect of  $D$  on average Nusselt number (top) and average fluid temperature in the cavity (bottom), while  $K = 5.0$ .

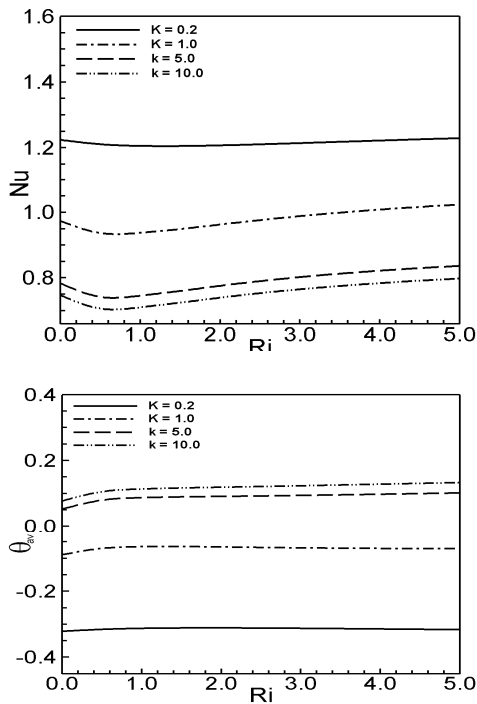


Fig 7. Effect of  $K$  on (i) average Nusselt number and (ii) average fluid temperature in the cavity, while  $D = 0.2$ .

## 6. CONCLUSION

The present study investigates numerically the characteristics of a two dimensional mixed convection problem in a ventilated cavity with an inner heated hollow circular cylinder. A detailed analysis of the distribution of streamlines, isotherms, average Nusselt number and average fluid temperature in the cavity was carried out to investigate the diameter of the cylinder and solid fluid thermal conductivity ratio on the fluid flow and heat transfer in the mentioned cavity for different Richardson number  $Ri$ . The following conclusions may be drawn from the present investigations

- Cylinder diameter has significant effect on the flow and thermal fields at the three convective regimes. Maximum average Nusselt number and average fluid temperature are found for the largest value of the Cylinder diameter.
- Solid fluid thermal conductivity ratio has insignificant effect on the flow and has significant effect on thermal fields at the three convective regimes. Maximum average Nusselt number and minimum average fluid temperature is found for the largest value of  $K$ .

## 7. REFERENCES

1. 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.
2. Lacroix, M., 1992, "Natural convection heat transfer around two heated horizontal cylinders inside a rectangular cavity cooled from above", Numer. Heat Transfer, Part A, 21: 37-54.
3. Lacroix, M. and Joyeux, A., 1995, "Natural convection heat transfer around heated cylinders inside a cavity with conducting walls", Numer. Heat Transfer, Part A, 27: 335-349.
4. Ha, M. Y., Jung, M. J. and Kim, Y.S., 1999, "Numerical study on transient heat transfer and fluid flow of natural convection in an enclosure with a heat-generating conducting body", Numer. Heat Transfer, Part A, 35: 415-434.
5. Ha, M. Y. and Jung, M. J., 2000, "A numerical study on three-dimensional conjugate heat transfer of natural convection and conduction in a differentially heated cubic enclosure with a heat-generating cubic conducting body", Int. J. of Heat and Mass Transfer, 43: 4229-4248.
6. Roychowdhury, D.G, Das, S.K. and Sundararajan, T.S., 2002, "Numerical simulation of natural convection heat transfer and fluid flow around a heated cylinder inside an enclosure", Heat and Mass Transfer, 38: 565-576.
7. Dong, S. F., and Li, Y.T., 2004, "Conjugate of natural convection and conduction in a complicated enclosure", Int. J. of Heat and Mass Transfer, 47: 2233-2239.

8. Tasnim, S. H. and Collins, M. R., 2004, "Numerical analysis of heat transfer in a square cavity with a baffle on the hot wall", *Int. Commun. Heat Transfer*, 31 (5): 639-650.
9. Braga, E. J., and de Lemos, M. J. S., 2005, "Laminar natural convection in cavities filled with circular and square rods", *Int. Commun. in Heat and Mass Transfer*, 32: 1289-1297.
10. Bhave, P., Narasimhan, A. and Ress, D. A. S., 2006, "Natural convection heat transfer enhancement using adiabatic block: Optimal block size and Prandtl number effect", *Int. J. of Heat and Mass Transfer*, 49: 3807-3818.
11. Kumar De, A. and Dalal, A., 2006, "A numerical study of natural convection around a square, horizontal, heated cylinder placed in an enclosure", *Int. J. of Heat and Mass Transfer*, 49: 4608-4623.
12. Hsu, T.H., Hsu, P.T., How, S.P., 1997, "Mixed convection in a partially divided rectangular enclosure", *Numerical Heat Transfer, Part A*, 31: 655-683.
13. How, S.P. and Hsu, T.H., 1998, "Transient mixed convection in a partially divided enclosure", *Comput. Math. Appl.*, 36: 95-115.
14. Hsu, T.H., and How, S. P., 1999 "Mixed convection in an enclosure with a heat-conducting body", *Acta Mechanica*, 133: 87-104.
15. Rahman, M. M., Alim, M. A., Saha, S. and Chowdhury, M. K., 2008a, "A Numerical Study of Mixed Convection in A Square Cavity with A Heat Conducting Square Cylinder at Different Locations", *J. of Mechanical Engineering, The Institution of Engineers, Bangladesh*, ME 39 (2): 78 - 85.
16. Rahman, M. M., Alim, M. A., Saha, S. and Chowdhury, M. K., 2008b, "Mixed Convection in a vented Square Cavity with a Heat Conducting Horizontal Solid Circular Cylinder", *Journal of Naval Architecture and Marine Engineering*, 5 (2): 37 - 46.
17. Rahman, M. M., Alim, M. A. and Mamun, M. A. H., 2009, "Finite element analysis of mixed convection in a rectangular cavity with a heat-conducting horizontal circular cylinder", *Nonlinear analysis: Modeling and Control*, 14 (2): 217-247.
18. Zienkiewicz, O. C. and Taylor, R. L., 1991, "The finite element method", Fourth Ed., McGraw-Hill.

## 8. NOMENCLATURE

Symbol	Meaning	Unit
d	Dimensional cylinder diameter	(m)
D	Nondimensional cylinder diameter	
g	gravitational acceleration	( $ms^{-2}$ )
Gr	Grashof number	
h	Convective heat transfer coefficient	
$K_f$	Thermal conductivity of fluid	$Wm^{-1}K^{-1}$
$K_s$	Thermal conductivity of solid	$Wm^{-1}K^{-1}$
K	Solid fluid thermal conductivity ratio	
L	Length of the cavity	(m)
$L_h$	Surface area of the of the heated wall	( $m^2$ )
Nu	Nusselt number	
p	Dimensional pressure	( $Nm^{-2}$ )
P	Dimensionless pressure	
Pr	Prandtl number	
Re	Reynolds number	
Ra	Rayleigh number	
Ri	Richardson number	
S	Distance along the square enclosure	(m)
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	
<b>Greek symbols</b>		
$\alpha$	Thermal diffusivity	( $m^2s^{-1}$ )
$\beta$	Thermal expansion coefficient	( $K^{-1}$ )
$\nu$	Kinematic viscosity	( $m^2s^{-1}$ )
$\theta$	Non dimensional temperature	
$\rho$	Density of the fluid	( $kgm^{-3}$ )
<b>Subscri</b>		
pts		
av	Average	
i	Inlet state	
s	Solid	

## 9. MAILING ADDRESS

M. M. Rahman  
 Department of Mathematics,  
 Bangladesh University of Engineering and Technology  
 (BUET), Dhaka, Bangladesh