



## MIXED CONVECTION IN A VENTED SQUARE CAVITY WITH A HEAT CONDUCTING HORIZONTAL SOLID CIRCULAR CYLINDER

Md. Mustafizur Rahman<sup>1</sup>, M. A. Alim<sup>1</sup>, Sumon Saha<sup>2</sup> and M. K. Chowdhury<sup>1</sup>

<sup>1</sup>Department of Mathematics, <sup>2</sup>Department of Mechanical Engineering,

Bangladesh University of Engineering and Technology (BUET), Dhaka-1000, Bangladesh Tel.: +88-02-9665650, E-mail: mmustafizurrahman@math.buet.ac.bd

### Abstract

A finite element method based computational study of steady laminar mixed convection flow inside a vented square cavity with a heat conducting horizontal solid circular cylinder placed at the center of the cavity is carried out in this paper. The developed mathematical model is governed by the coupled equations of continuity, momentum and energy. The present work simulates practical systems such as cooling of electronic devices, ventilation of building etc. The effects of cylinder size and Richardson number on fluid flow and heat transfer performance are investigated. Richardson number is varied from 0.0 to 5.0 and the cylinder diameter is varied from 0.0 to 0.6. The results for the case of without cylinder are compared to those with cylinder to observe the effects cylinder on the flow and heat transfer inside the cavity. The phenomenon inside the cavity for the case of with and without cylinder is analyzed through streamline and isotherm patterns. It is found that the streamlines, isotherms, average Nusselt number at the heated surface, average temperature of the fluid in the cavity and dimensionless temperature at the cylinder center strongly depend on the Richardson number as well as the diameter of the cylinder.

**Keywords:** Mixed convection, finite element method, Richardson number, cylinder diameter, vented cavity and diffusion.

### NOMENCLATURE

$d$	dimensional cylinder diameter ( $m$ )	$\bar{V}$	cavity volume ( $m^3$ )
$D$	non dimensional cylinder diameter	$w$	height of the opening ( $m$ )
$G$	gravitational acceleration ( $ms^{-2}$ )	$x, y$	cartesian coordinates ( $m$ )
$H$	convective heat transfer coefficient	$X, Y$	dimensionless cartesian coordinates
$K$	thermal conductivity of fluid ( $Wm^{-1}K^{-1}$ )	<i>Greek symbols</i>	
$k_s$	thermal conductivity of solid ( $Wm^{-1}K^{-1}$ )	$\alpha$	thermal diffusivity ( $m^2s^{-1}$ )
$K$	solid fluid thermal conductivity ratio	$\beta$	thermal expansion coefficient ( $K^{-1}$ )
$L$	length of the cavity ( $m$ )	$\nu$	kinematic viscosity ( $m^2s^{-1}$ )
$Nu$	Nusselt number	$\theta$	non dimensional temperature
$P$	dimensional pressure ( $Nm^{-2}$ )	$\rho$	density of the fluid ( $kgm^{-3}$ )
$P$	dimensionless pressure	<i>Subscripts</i>	
$Pr$	Prandtl number	$av$	average
$Re$	Reynolds number	$h$	heated wall
$Ra$	Rayleigh number	$i$	inlet state
$Ri$	Richardson number	$c$	cylinder center
$T$	dimensional temperature ( $K$ )		
$u, v$	dimensional velocity components ( $ms^{-1}$ )		
$U, V$	dimensionless velocity components		

### 1. Introduction

Mixed convection flow and heat transfer in a vented cavity have been the topic of many research in engineering studies. These studies consist of various technological applications such as in electronic cooling, ventilation of building and design of solar collectors etc. Analysis of above phenomena incorporating a heat conducting solid cylinder extends its usability to many other practical situations such as any projections on a motherboard of a computer and a conductive material in an inert atmosphere inside a furnace with a constant flow of gas from

outside constitute practical application for the present study. A brief review of the relevant literature is presented in the following section.

House *et al.* (1990) studied the effect of a centered, square, heat conducting body on natural convection in a vertical enclosure. They found that heat transfer across the cavity enhanced or reduced by a body with a thermal conductivity ratio less or greater than unity. The same geometry was considered in the numerical study of Oh *et al.* (1997), where the conducting body generated heat within the cavity. Under these situations, they showed that the flow was driven by a temperature difference across the cavity and a temperature difference caused by the heat-generating source. Later on, Roychowdhury *et al.* (2002) analyzed the natural convective flow and heat transfer features for a heated body placed in a square enclosure with different thermal boundary conditions. Dong and Li (2004) 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. Moreover, Bilgen and Yamane (2004) examined numerically the effect of conjugate heat transfer by laminar natural convection and conduction in two-dimensional rectangular enclosures with openings. A chimney inside the enclosure was simulated as a vertical rectangular body with a uniform heat flux on one side and insulation on the other. They investigated the effects of the various geometrical parameters and the thickness of the insulation layer on the fluid flow and heat transfer characteristics. Bhoite *et al.* (2005) studied numerically the problem of mixed convection flow and heat transfer in a shallow enclosure with a series of block-like heat generating components for a range of Reynolds numbers and Grashof numbers and block-to-fluid thermal conductivity ratios. They showed that higher Reynolds numbers created a recirculation region of increasing strength at the core region and the effect of buoyancy became insignificant beyond a Reynolds number of typically 600 and hence the thermal conductivity ratio had a negligible effect on the velocity fields. At the same time, Braga and de Lemos (2005) studied 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 volume 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. 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 (2005). 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. Recently, conjugate natural convection heat transfer inside an inclined square cavity with an internal conducting block was carried out by Das and Reddy (2006). At the same time, Xu *et al.* (2006) experimentally investigated the thermal flow around a square obstruction on a vertical wall in a differentially heated cavity. Zhao *et al.* (2006) numerically investigated conjugate natural convection in enclosures with external and internal heat sources.

The literature review shows that numerous studies have been published on a natural convection heat transfer in a closed cavity containing a heat conducting body. But we are not aware of any work that has been performed in a vented cavity with a solid circular cylinder. The objective of the present study is to investigate the effect of centered cylinder on fluid flow and heat transfer in a square cavity. The Richardson number ( $Ri$ ) is varied from 0.0 to 5.0 to simulate forced convection, pure mixed convection and free convection dominated flow in the cavity. The size of the cylinder is also varied from 0.0 to 0.6 to understand the effect of cylinder diameter on the flow and heat transfer in the cavity. In the present study the solid cylinder is assumed to be heat conducting keeping in view their applications in many practical situations.

## 2. Problem Definition

A schematic diagram of the system considered in the present study is shown in Fig. 1. The system consists of a square cavity with sides of length  $L$ , within which a heat conducting solid circular cylinder with diameter  $d$  and thermal conductivity  $k_s$  is located at the center of the cavity. A Cartesian co-ordinate system is used with origin at the lower left hand 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 the uniform constant temperature  $T_h$ . The inflow opening located on the bottom of the left vertical wall and the outflow opening on the top of the opposite heated wall is arranged as shown in the schematic figure. For simplicity, the size of the two openings is set equal to the one-tenth of the cavity length ( $w = 0.1 L$ ). Cold air flows through the inlet at a uniform velocity  $u_i$ . It is 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 variables i.e. convective boundary conditions (CBC). All solid boundaries are assumed to be rigid no-slip walls.

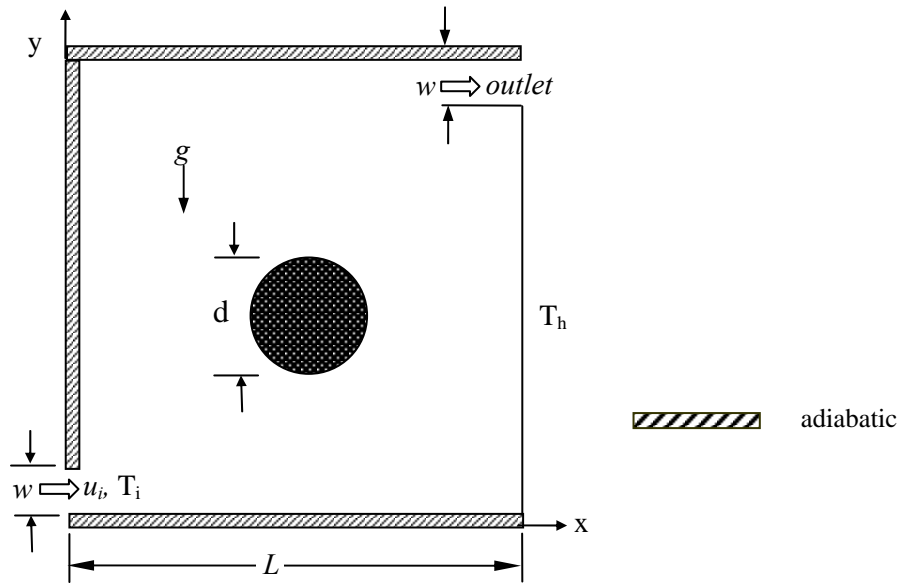


Figure 1: Schematic of the problem with the domain and boundary conditions

### 3. Governing Equations with Boundary Conditions

The following assumptions are made: the flow of fluid is steady, two-dimensional, laminar incompressible and Newtonian, there is no viscous dissipation, the gravity acts in the vertical downward direction, fluid properties are constant and fluid density variations are neglected except in the buoyancy term (Boussinesq approximation) and radiation heat exchange is negligible.

Using non-dimensional variables defined below, the non-dimensional governing equations are obtained 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 solid cylinder, the energy equation is

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

The non-dimensional variables used in the above equations are defined as

$$X = \frac{x}{L}, Y = \frac{y}{L}, U = \frac{u}{u_i}, V = \frac{v}{u_i}, D = \frac{d}{L}, P = \frac{p}{\rho u_i^2}, \theta = \frac{(T - T_i)}{(T_h - T_i)} \text{ and } \theta_s = \frac{(T_s - T_i)}{(T_h - T_i)}$$

and the parameters  $Re$ ,  $Ri$ ,  $Pr$  and  $K$  are defined as

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

The appropriate boundary conditions (also shown in Fig. 1) used to solve the equations (1)-(5) inside the cavity in dimensionless forms are given as

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

At the outlet: Convective boundary condition (CBC),  $P = 0$

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

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

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

At the solid-fluid vertical interfaces of the cylinder:  $\left(\frac{\partial \theta}{\partial X}\right)_{fluid} = K \left(\frac{\partial \theta_s}{\partial X}\right)_{solid}$

At the solid-fluid horizontal interfaces of the cylinder:  $\left(\frac{\partial \theta}{\partial Y}\right)_{fluid} = K \left(\frac{\partial \theta_s}{\partial Y}\right)_{solid}$

The average Nusselt number  $Nu$  is defined as

$$Nu = \frac{1}{L_h} \int_0^{L_h/L} \frac{\partial \theta}{\partial X} dY$$

and the bulk average temperature in the cavity is defined as  $\theta_{av} = \frac{1}{V} \int \theta d\bar{V}$ .

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

#### 4. Numerical Technique

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 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. The integration involved in each term of these equations is performed by using Gauss quadrature method. Then 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, these linear equations are solved by using Triangular Factorization method.

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

Elements (Nodes)	3818 (24740)	4516 (28998)	5838 (37402)	5918 (37886)	7512 (47986)
$Nu$	4.842422	4.842209	4.832593	4.832878	4.842449
$\theta_{av}$	0.197191	0.197204	0.197233	0.197224	0.197227
Time(s)	385.219	493.235	682.985	698.703	927.359

#### 4.1 Grid Refinement Check

Five different grid sizes of 3818, 4516, 5838, 5918 and 7512 elements with 24740, 28998, 37402, 37886 and 47986 nodes respectively are chosen for the present simulation to test the independency of the results with the grid variations. Average Nusselt number at the heated surface, average temperature of the fluid in the cavity and the solution time are monitored at  $Ri = 1.0, D = 0.2$  and  $K = 5.0$  for these grid elements. The magnitude of average Nusselt number at the heated surface and average temperature of the fluid in the cavity for the above grids shows a very little difference. However for the rest of the calculation in this study we choose a grid size of 37886 nodes 5918 elements for better accuracy.

## 4.2 Code Validation

The work has been validated against the work of House *et al.* (1990) for natural convection in a closed enclosure. We calculated the average Nusselt number at the heated wall for the square enclosure with a square heat conducting body at  $Ra = 0.0, 10^5$  and compared with the data from their results as presented in Table 2. The present average Nusselt numbers are in good agreement with these of House *et al.* (1990).

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

$Ra$	$K$	Present study $Nu$	House <i>et al.</i> (1990) $Nu$
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

## 5. Results and Discussion

A numerical study is performed through finite element method to analyze the laminar mixed convection flow and heat transfer in a vented square cavity filled with a horizontal solid circular cylinder. The mixed convection phenomenon inside the cavity is influenced by Richardson number ( $Ri$ ) Reynolds number ( $Re$ ), dimensionless form of the cylinder diameter ( $D$ ) and solid fluid thermal conductivity ratio ( $K$ ). Reynolds number and solid fluid thermal conductivity ratio are kept fixed at 100 and 5.0 respectively. Effect of the varying parameters such as Richardson number and cylinder diameter on fluid flow and heat transfer are analyzed. Air is chosen as working fluid with  $Pr = 0.71$ .

We presented the results in two sections. The first section will focus on flow and temperature fields, which contains streamlines and isotherms for the mentioned cases and the following section will discuss heat transfer including average Nusselt number at the heated surface, average temperature of the fluid and the temperature at the cylinder center in the cavity.

### 5.1 Flow and Temperature Fields

Flow and temperature fields are simulated using streamlines and isotherms for various values of cylinder diameter  $D$  and Richardson number  $Ri$ . Effects of cylinder diameter on streamlines (on the left) are presented in Fig. 2 for  $Ri = 0.0$ . It is seen from the figure that for  $D = 0.0$ , there develops two vortices of opposite directions. The vortex with anticlockwise direction is developed just at the top of the inlet port along the left wall, which is expected since the fresh stream enters the cavity at the bottom of the left wall and exit at the top of the right wall. Also a clockwise small vortex appears at the right bottom corner of the cavity. It is also seen that the size of vortex near the left wall decreases and the small vortex at the right bottom corner diminishes as cylinder diameter increases. This is because, the increasing cylinder size reduces the available space for the shear forced induced by the through stream at the inlet. The corresponding isotherms (on the right) of Fig. 2 are stratified near the heated surface indicating diffusion heat transfer in the cavity. As cylinder diameter ( $D$ ) increases the isotherms are increasingly distorted from the shapes corresponding to diffusion heat transfer in the cavity. At  $Ri = 1$ , it is seen from Fig. 3 that the natural convection effect is comparable to forced convection for both the streamlines and isotherms at the lower values of  $D$  ( $D = 0.0, 0.2$ ), but remains relatively weak at the higher values of  $D$  ( $D = 0.4, 0.6$ ), which is reasonable because, the increasing cylinder size reduces the available space for the buoyancy induced flow. Further for  $Ri = 5.0$ , the vortex spreads as shown in Fig. 4. As a consequence, the induced flow is squeezed and the vortex almost covers the cavity for the values of  $D$  ( $D = 0.0, 0.2$  and  $0.4$ ) indicating the dominance of the natural convection heat transfer in the cavity. On the other hand, the single vortex spreads into three vortices for the highest value of  $D$  ( $D = 0.6$ ). The corresponding isothermal lines become nonlinear and plume is formed which indicate the supremacy of natural convection heat transfer over forced convection and conduction in the cavity.

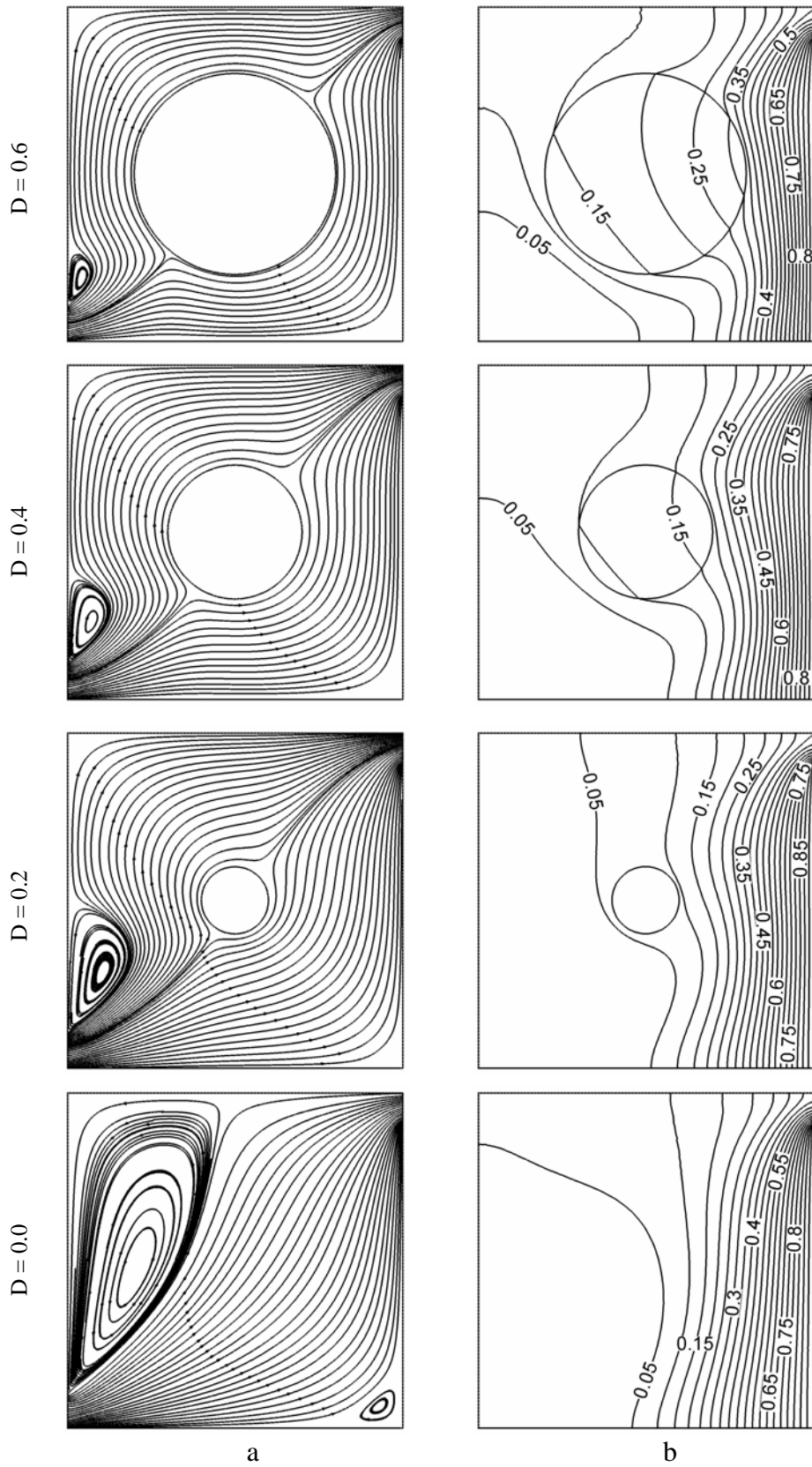


Figure 2: (a) Streamlines (left) and (b) Isotherms (right) for various diameters of the cylinder at  $Ri = 0.0$

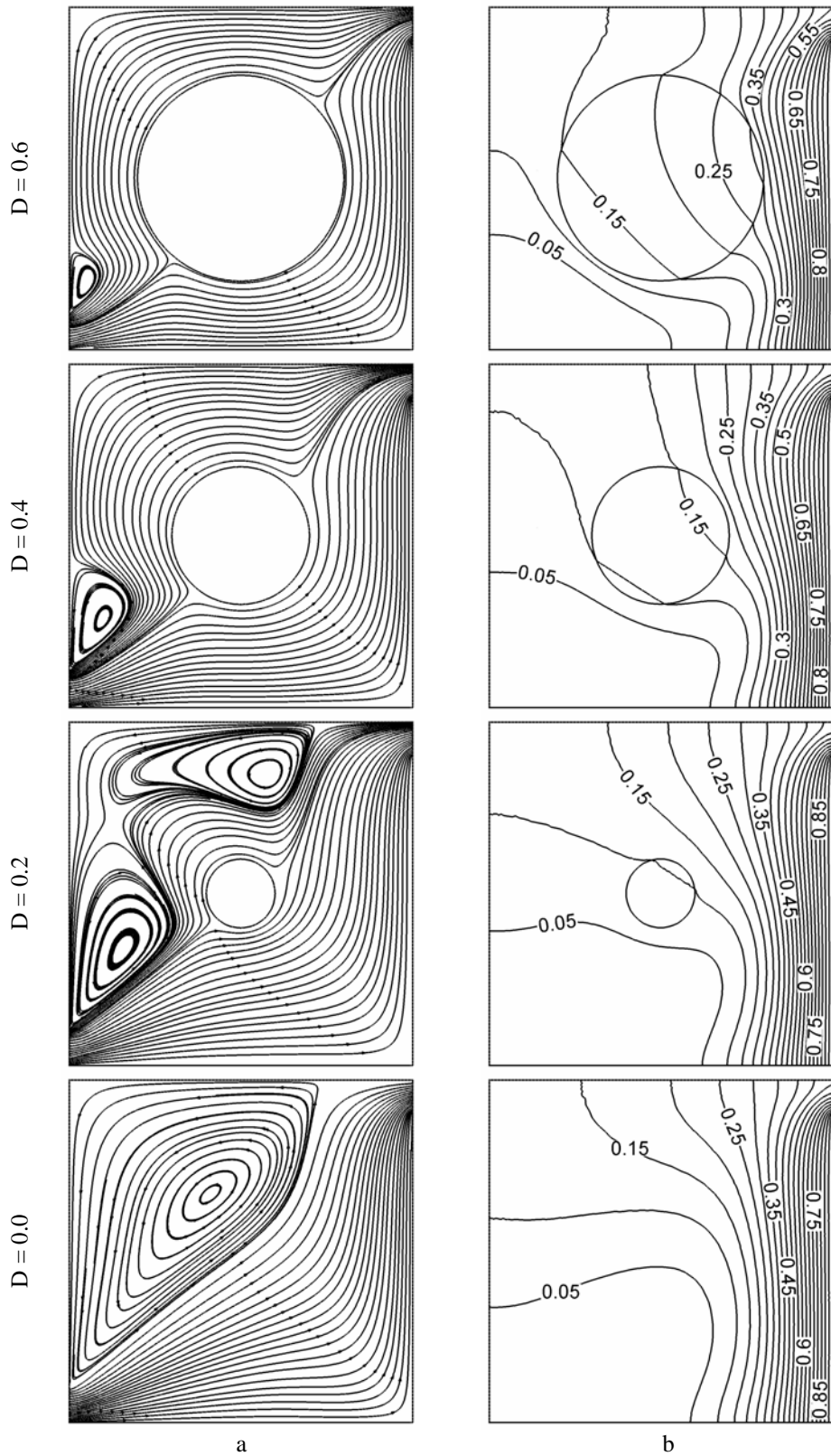


Figure 3: (a) Streamlines (left) and (b) Isotherms (right) for various diameters of the cylinder at  $Ri = 1.0$ .

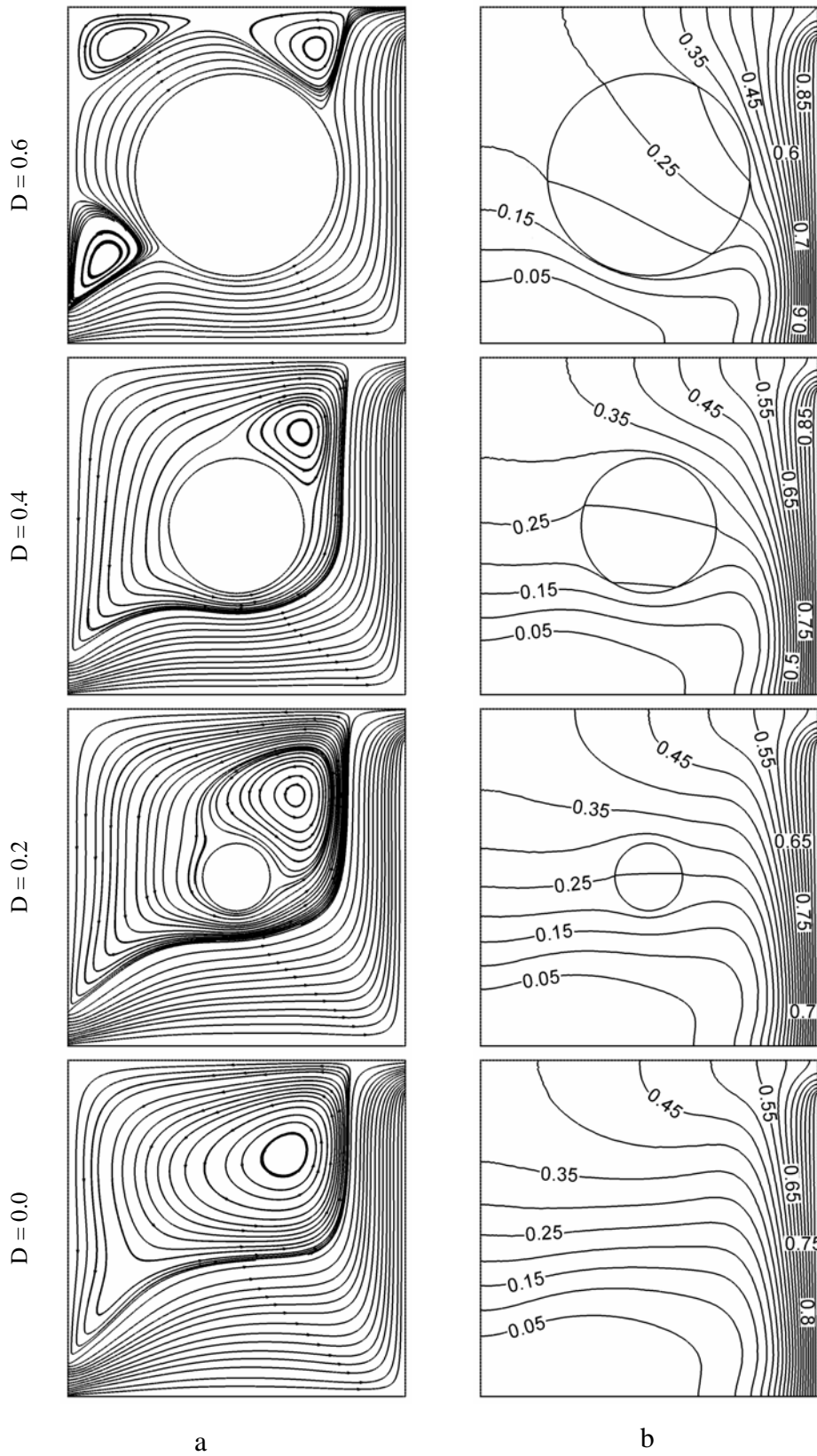


Figure 4: (a) Streamlines (left) and (b) Isotherms (right) for various diameters of the cylinder at  $Ri = 5.0$ .



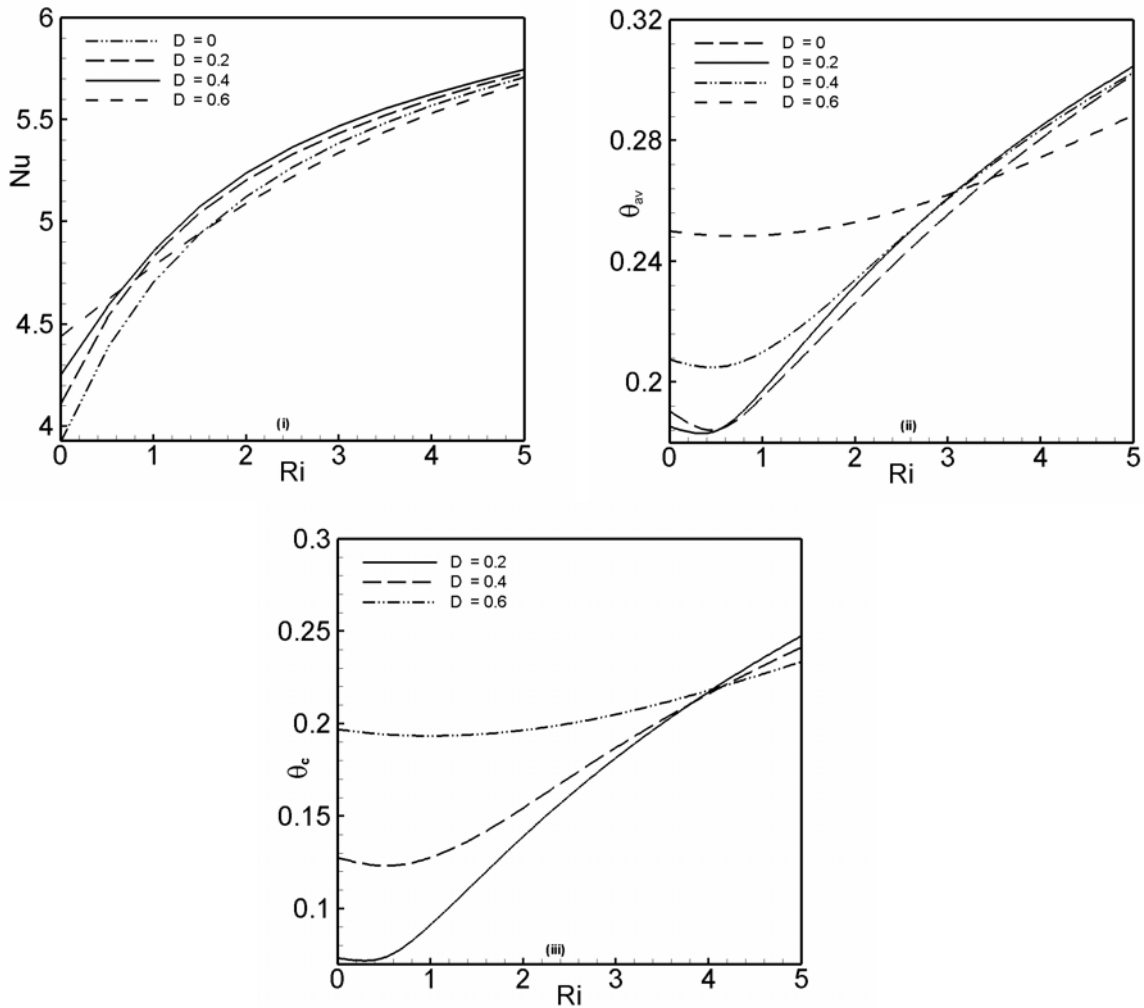


Figure 5: Effect of Richardson number on (i) average Nusselt number, (ii) average fluid temperature and (iii) temperature at the cylinder center for different diameters of the cylinder.

## 5.2 Heat Transfer

Fig. 5 (i) shows the average Nusselt number ( $Nu$ ) at the hot wall of the cavity as a function of Richardson number ( $Ri$ ) for different cylinder diameters. The average Nusselt number ( $Nu$ ) at the hot wall increases generally with increasing Richardson number ( $Ri$ ) due to the increasing effect of convection. On the other hand, for a particular values of  $Ri$  average Nusselt number ( $Nu$ ) at the heated wall is the highest for  $D = 0.6$  in the forced convection dominated region ( $0.0 \leq Ri \leq 0.75$ ) and beyond these values of  $Ri$  it is the highest for  $D = 0.4$ . The average temperature ( $\theta_{av}$ ) of the fluid in the cavity and the dimensionless temperature ( $\theta_c$ ) at the cylinder center as a function of  $Ri$  for different cylinder diameters  $D$  are also shown in Fig. 5 (ii-iii). The average temperature ( $\theta_{av}$ ) of the fluid in the cavity and the dimensionless temperature ( $\theta_c$ ) at the cylinder center are not monotonic in the cavity. From these figures it is also seen that for a particular value of  $D$ , the values of  $\theta_{av}$  and  $\theta_c$  are not monotonic with increasing  $Ri$ . On the other hand, the values of  $\theta_{av}$  is the lowest for  $D = 0.2$  at  $Ri \leq 0.5$  and for  $D = 0.0$  at  $0.5 < Ri \leq 3.5$ , but beyond these values of  $Ri$  it is the lowest for  $D = 0.6$ . The dimensionless temperature ( $\theta_c$ ) at the cylinder center is the lowest for  $D = 0.2$  at  $Ri \leq 4.0$ , beyond these values of  $Ri$  it is the lowest for  $D = 0.6$ .

## 6. Conclusion

The present study investigates numerically the characteristics of a two dimensional mixed-convection problem in a vented square cavity with a heat conducting horizontal solid circular cylinder. The weighted residuals finite element method is implemented to simulate the flow and heat transfer in the cavity. A detailed analysis for the distribution of streamlines, isotherms, average Nusselt number at the heated surface average temperature of the fluid and the dimensionless temperature at the cylinder center are carried out to investigate the effect of the diameter of the inner cylinder on the fluid flow and heat transfer in the cavity for different Richardson numbers in the range of  $0.0 \leq Ri \leq 5.0$ . Markedly dissimilar flow and thermal behaviors are observed for the various diameter of the cylinder among the three different flow regimes. The average Nusselt number at the heated surface is the highest for the largest cylinder diameter  $D = 0.6$  in the forced convection dominated region and for the second largest diameter  $D = 0.4$  in the free convection dominated region at a fixed  $Ri$ . On the other hand a gradual increase in the heat transfer rate is found with increasing  $Ri$  at constant values of cylinder diameter. It is also noted that the average temperature ( $\theta_m$ ) of the fluid in the cavity and the dimensionless temperature ( $\theta_c$ ) at the cylinder center in the cavity are not monotonic with the increasing values of the parameters.

## Acknowledgements

The authors like to express their gratitude to the Department of Mathematics, Bangladesh University of Engineering and Technology, for providing computing facility during this work.

## References

- 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, Vol. 18, pp. 213–225.
- Oh, J. Y., Ha, M. Y., and Kim, K. C. (1997): Numerical Study of Heat Transfers and Flow of Natural Convection in an Enclosure with a Heat Generating Conducting Body, Numerical Heat Transfer, Part A, Vol. 31, pp. 289-304.
- Roychowdhury, D.G, Das, S.K. and Sundararajan, T.S. (2002): Numerical Simulation of Natural Convection Heat Transfer and Fluid Flow around a Heated Body inside an Enclosure, Heat and Mass Transfer, Vol. 38, pp. 565-576. [doi:10.1007/s002310100210](https://doi.org/10.1007/s002310100210)
- Bilgen, E., and Yamane, T. (2004): Conjugate Heat Transfer in Enclosures with Openings for Ventilation, Heat and Mass Transfer, Vol. 40, pp. 401-411. [doi:10.1007/s00231-003-0418-z](https://doi.org/10.1007/s00231-003-0418-z)
- 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, Vol. 47, pp. 2233-2239. [doi:10.1016/j.ijheatmasstransfer.2003.11.018](https://doi.org/10.1016/j.ijheatmasstransfer.2003.11.018)
- Bhoite, M. T., Narasimham, G. S. V. L., and Murthy, M. V. K. (2005): Mixed Convection in a Shallow Enclosure with a Series of Heat Generating Components, Int. J. of Thermal Sciences, Vol. 44, pp. 125-135. [doi:10.1016/j.ijthermalsci.2004.07.003](https://doi.org/10.1016/j.ijthermalsci.2004.07.003)
- Braga, E. J., and de Lemos, M. J. S. (2005): Laminar Natural Convection in Cavities Filled with Circular and Square Rods, Int. Comm. in Heat and Mass Transfer, Vol. 32, pp. 1289-1297. [doi:10.1016/j.icheatmasstransfer.2005.07.014](https://doi.org/10.1016/j.icheatmasstransfer.2005.07.014)
- Tasnim, S. H., and Collins, M. R. (2005): Suppressing Natural Convection in a Differentially Heated Square Cavity with an Arc Shaped Baffle, Int. Comm. in Heat and Mass Transfer, Vol. 32, pp. 94-106. [doi:10.1016/j.icheatmasstransfer.2004.05.022](https://doi.org/10.1016/j.icheatmasstransfer.2004.05.022)
- Das, M. K., and Reddy, K. S. K. (2006): Conjugate natural convection heat transfer in an inclined square cavity containing a conducting block, Int. J. of Heat and Mass Transfer, Vol. 49, pp. 4987-5000. [doi:10.1016/j.ijheatmasstransfer.2006.05.041](https://doi.org/10.1016/j.ijheatmasstransfer.2006.05.041)
- Xu, F., Patterson, J. C., and 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. [doi:10.1007/s00348-005-0073-x](https://doi.org/10.1007/s00348-005-0073-x)
- Zhao, F. Y., Tang, G. F., and Lin, D. (2006): Conjugate Natural Convection in Enclosures with External and Internal Heat Sources, Int. J. of Engg. Science, Vol. 44, pp. 148-165. [doi:10.1016/j.ijengsci.2005.10.006](https://doi.org/10.1016/j.ijengsci.2005.10.006)
- Taylor, C. and Hood, P. (1973): A Numerical Solution of the Navier-Stokes Equations Using Finite Element Technique, Vol. 1, Computer and Fluids 1, pp.73-89. [doi:10.1016/0045-7930\(73\)90027-3](https://doi.org/10.1016/0045-7930(73)90027-3)
- Dechaumphai, P. (1999): Finite Element Method in Engineering, 2nd ed., Chulalongkorn University Press, Bangkok.