International Journal of Theoretical & Applied Sciences **5**(1): 111-118(2013)

ISSN No. (Print): 0975-1718
ISSN No. (Online): 2249-3247

Numerical Analysis Of Mixed Convection In A Square Cavity Having Adiabatic Circular Cylinder

Mohammed Nasir Uddin* and Mohammad Abdul Alim**

**Department of Mathematics, Bangladesh University of Business & Technology,* Dhaka -1216, Bangladesh
***Department of Mathematics, Bangladesh University of Business & Technology,* Dhaka -1216, Bangladesh

(Received 05 March, 2013, Accepted 15 April, 2013)

ABSTRACT: Mixed convection in a square cavity with a uniform heat source applied on the bottom horizontal wall is studied numerically. An adiabatic circular cylinder is placed within the cavity. The developed mathematical model is governed by the coupled equations of continuity, momentum and energy to determine the fluid flow and heat transfer characteristics in the cavity as a function of Reynolds number (*Re*), Prandtl number (*Pr*), Richardson number (*Ri*) and some physical parameters. In this paper, a finite element formulation for steady-state incompressible conjugate mixed convection and conduction flow is developed. The heat transfer rate at heated wall is strongly influenced with the range of parameters of *Ri* (1 - 10), *Re* (50 -200), *Pr* (0.72 – 7.0), dr (0.0- 2.0) and configurations of inlet -outlet in the cavity. Detailed result discussion is given for the effect of the mentioned parameters.

Keywords: Mixed convection; finite element method; heat transfer; square cavity; adiabatic circular cylinder.

I. INTRODUCTION

Mixed convection in enclosures and partial enclosures has gained importance in the recent past, especially in the thermal management of electronics [1,2], engineering, for example cooling of electronic devices, furnaces, lubrication technologies, chemical processing equipment, drying technologies etc. Many authors have recently studied heat transfer in enclosures with partitions, which influence the convection flow phenomenon. Omri and Nasrallah [3] studied mixed convection in an air-cooled cavity with differentially heated vertical isothermal sidewalls having inlet and exit ports by a control volume finite element method. They investigated two different placement configurations of the inlet and exit ports on the sidewalls. 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. Later on, Singh and Sharif [4] extended Finite Element Analysis of Mixed Convection in a Rectangular Cavity their works by considering six placement configurations of the inlet and exit ports of a differentially heated rectangular enclosure whereas the previous work was limited to only two different configurations of inlet and exit port. . Very recently Manca *et al.* [5] experimentally analyzed opposing flow in mixed convection in a channel with an open cavity below. Recently Rahman *et al.* [6] studied numerically the opposing mixed convection in a vented enclosure. They found that with the increase of Reynolds and Richardson numbers the convective heat transfer becomes predominant over the conduction heat transfer and the rate of heat transfer from the heated wall is significantly depended on the position of the inlet port. However, many authors have studied heat transfer in enclosures with heat-conducting body obstruction, thereby influencing the convective flow phenomenon. Shuja *et al.* [7] investigated the 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. One of the systematic numerical investigations of this problem was conducted by House *et al.* [8], 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. As the first step toward accurate flow solutions using the adaptive meshing technique, this paper develops a finite element formulation suitable for analysis of steady-state conjugate mixed convection and conduction problems. The paper starts from the Navier-Stokes equations together with the energy equations to derive the corresponding finite element model. The computational procedure used in the development of the computer program is described. The finite element equations derived [8, 10] and then the computer program developed are then evaluated by example of mixed convection in a square cavity with heat conducting horizontal circular cylinder. The purpose of the present study is to investigate the effect of a circular cylinder and numerical solution is obtained over a wide range of Richardson number, Reynolds number, Prandtl number and various physical parameters.

II. PHYSICAL MODEL AND ASSUMPTIONS

The heat transfer and the fluid flow in a two-dimensional square cavity of length L having adiabatic circular cylinder with diameter ratio (dr) are considered, as shown in the schematic diagram of figure-1. The top and bottom walls of the cavity are heated Temperature T_h while the other sidewalls are kept adiabatic. The inflow opening located on the left-top adiabatic vertical wall and the outflow opening on the opposite bottom vertical wall is arranged as shown in the schematic figures. The cavity presented in Fig. 1 is subjected to an external flow that enters via the left-top of the insulated vertical wall and leaves via the right-bottom of the opposite vertical wall. For simplicity, the heights of the two openings are set equal to the one-tenth of the enclosure height. It is assumed that the incoming fluid flow through the inlet 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 variables i.e. convective boundary conditions (CBC). All solid boundaries are assumed to be rigid no-slip walls. The fluid is assumed to be different prandtl number (Pr = 0.72, 1.0 3.0 & 7.0) and Newtonian, and the fluid flow is considered to be laminar. The properties of the fluid are assumed to be constant.

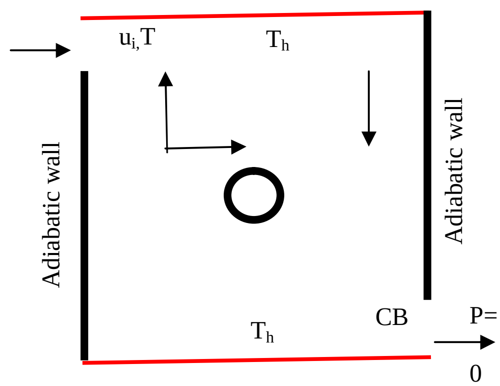


Fig. 1. Schematic diagram of the square cavity.

III. MATHEMATICAL MODEL

In the present problem, it can be assumed that the flow within the cavity is to be two-dimensional, steady, laminar, and incompressible and the fluid properties are to be constant. The Boussinesq approximation (buoyancy term) is consider and the radiation effects is neglected. The dimensionless equations describing the flow are as follows:

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

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

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

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

Non-dimensional forms of the boundary conditions for the present problem are specified 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: $\frac{\partial \theta}{\partial N} = 0$

The above equations were normalized using the following dimensionless scales:

$$X = \frac{x}{L}, Y = \frac{y}{L}, \mathrm{Pr} = \frac{\nu}{\alpha}, \ \Delta T = (T_h - T_c), P = \frac{p - p_\infty}{\rho u_i^2}$$

$$U = \frac{u}{u_i} \quad V = \frac{v}{v_i}, \ dr = \frac{D}{L} \quad \theta = \frac{T - T_i}{\Delta T},$$

$$G_r = \frac{g \beta \Delta T L^3}{\nu^2},$$

$$\mathrm{Re} = \frac{UL}{\nu} \qquad Ri = \frac{Gr}{\mathrm{Re}^2}$$

The Nusselt number (Nu) is one of the important dimensionless parameters to be computed for heat transfer analysis in natural convection flow. The average or overall Nusselt number at the heated wall is calculated by

$$Nu = \frac{1}{L}\int_0^L \frac{h(y)\,y}{k}\,dy$$

IV. NUMERICAL ANALYSIS

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 [11] and Dechaumphai [12].The finite element formulation and computational procedure are omitted herein for brevity.

V. RESULTS AND DISCUSSION

A numerical study has been performed through finite element method to analyze the laminar mixed convection heat transfer and fluid flow in a vented square cavity filled with adiabatic circular cylinder. The dimensionless governing parameters such as Reynolds number (*Re*), Richardson number (*Ri*), Prandtl number (*Pr*) and cylinder diameter (dr) have been analyzed. The presentations of the results have been started with the streamline and isotherm patterns in the cavity. Representative distributions of average Nusselt number at the heated wall have also been presented.

Obviously for high values of Reynolds number the errors encountered are appreciable and hence it is necessary to perform some grid size testing in order to establish a suitable grid size. Grid independent solution is ensured by comparing the results of different grid meshes for Re = 200, which was the highest Reynolds number. The total domain is discretized into 6176 elements that results in 40858 nodes. In order to validate the numerical code, pure natural convection with Pr = 0.71 in a square cavity was solved and the average Nusselt numbers is presented by graphically. The results were compared with those reported by Rahman *et al.* [4], obtained with an extended computational domain. In figure-2, a comparison between the average Nusselt numbers is presented. The results from the present experiment are almost same as Rahman et al.

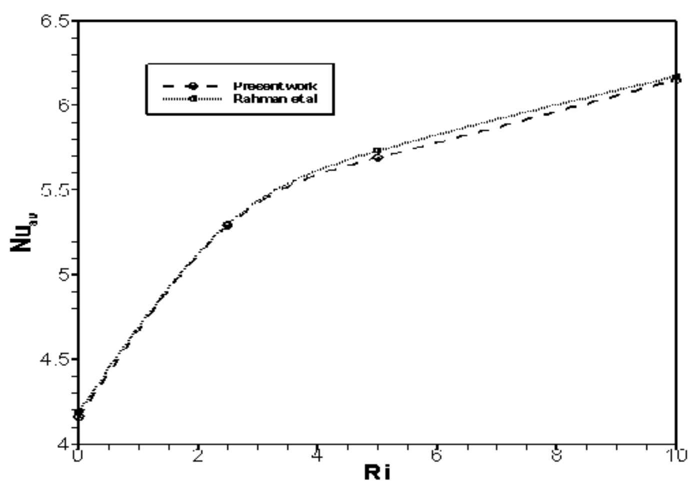


Fig. 2. Comparison of the average Nusselt number with the Richardson number while Re = 100, Pr = 0.71 and k = 10.

A. Effect of the cylinder diameter

The fluid flow and thermal fields is affected by cylinder diameter (dr) which is shown in figure 3 at AR = 1.0, Re = 100, Ri = 1.0, Pr = 0.71.We observe that in the Fig. 3, the flow pattern in the cavity reduces by the adiabatic circular cylinder and the strength of the recirculation cell induced by the heat source.

The streamlines flow create a path above the cylinder in the cavity from top to bottom for dr = 0.0 and 0.1. The slight difference have been found in the streamlines when compare cylinder diameter dr = 0.0 and dr = 0.1. It is clear that the convective flow slight influence by a small size circular cylinder diameter in the cavity. On the other hand, as the size of the cylinder increases, the space available for the buoyancy-induced re-circulating flow decreases. From the isotherms shown in Fig. 3, it has been observed that the high-temperature zone is confined to a region close to the hot surface for all cases and the lines are uniformly distributed in the cavity. A closer observation shows that the isotherm line in the cavity is slightly increased with the increase of the cylinder diameter (dr). we also observe that how heat transfer affected by the presence of the cylinder along the hot wall, average Nusselt number (Nu_{av}) has been plotted as a function of Richardson number (Ri) for different cylinder diameters (dr = 0.0, 0.1,0.15,0.2) shown in Fig. 3.

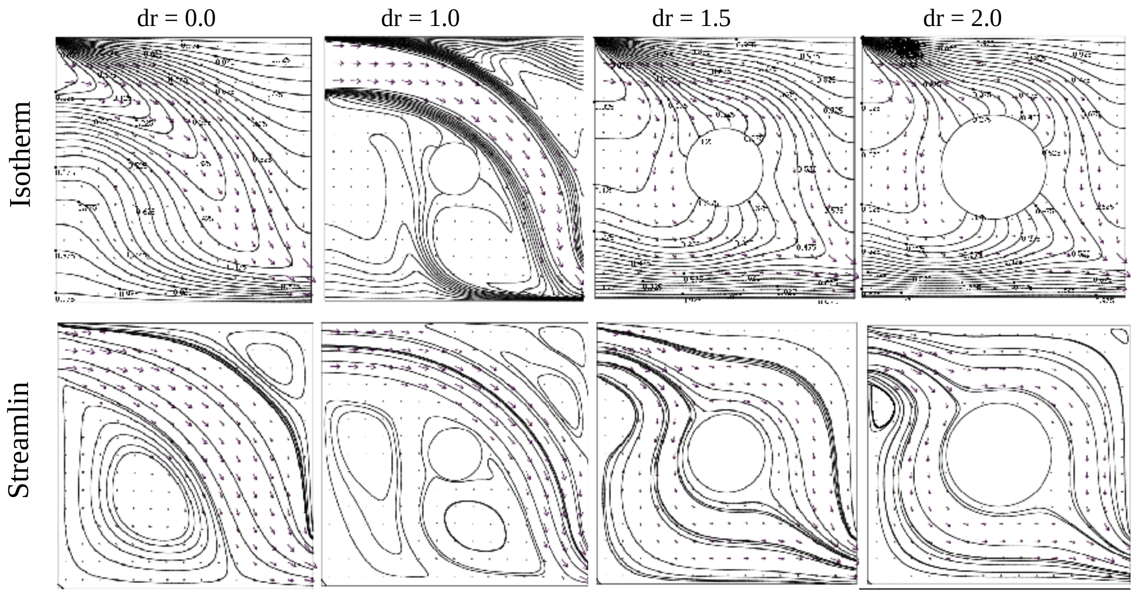


Fig. 3. Isotherms and Streamlines for the TB configuration at different diameter ratio (dr) While Re = 100, Ri = 1.0 and $L_x = L_y = 0.5$.

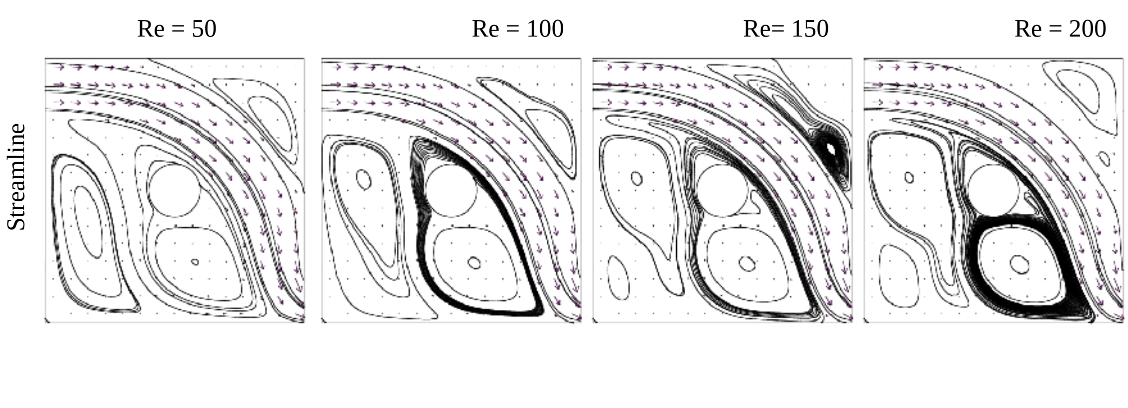
B. Effect of Reynolds number

Flow and temperature fields have simulated using streamlines and isotherms for the mentioned parameters.

In the Fig. 4 with Top Bottom (TB) configuration of cavity, the effects of the parameter *Re* on the flow and thermal fields at AR = 1.0, Ri = 1.0, dr = 0.1, Pr = 0.71 have been presented. At Re = 50, the external flow occupy almost the whole cavity and created a path above the cylinder form inlet to exit. Because, the thermal transport effect by the external cold air is little for small Re. The circulation of the flow shows two rotating vortices near the left bottom corner of the cavity and other vortex is right-top corner as shown in Fig. 4. The pattern of the streamlines become dense around the cylinder for Re = 100.As the increases of Reynolds number up to 200, the role of forced convection in the cavity become more significant, and consequently the circulation in the flow become more dense specially bottom vortices as presented in Fig. 4.

For the isotherms, the consequent temperature distributions have also been seen in Figs. 4. The external flow has created a path above the cylinder form inlet to exit. At fig-4, it has been observed that increase in Re reduces the thermal boundary layer thickness near the heated surface and it is possible, since at larger value of Re, the effect of gravitational force become negligible and the flow is governed by the forced convection. The average Nusselt numbers at the heat source in the cavity have been plotted as a function of Richardson number for a particular Reynolds numbers. From this figure it has been observed that for a fixed values of Ri, the average Nusselt number at the hot wall is the highest for large values of Re = 200. This is due to more heat has been carried away from the heat source and dissipated through the out flow opening for the large values of Re.

While Re = 100, Ri = 1.0 and $L_x = L_y = 0.5$.



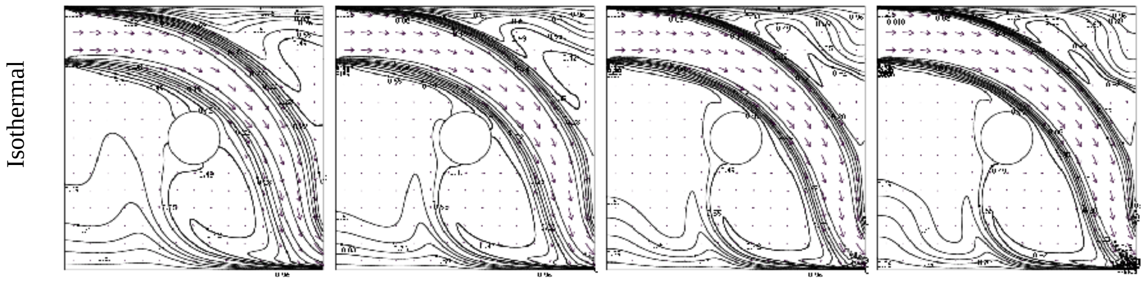


Fig. 4. Isotherms and Streamlines for the TB configuration at different Re While dr = 1.0, Ri = 1.0 and $L_x = L_y = 0.5$.

C. Effect of Richardson number

From the Figure-5, it has been indicated that the effect of streamlines and isotherms for the TB configuration at AR = 1.0, Re = 100 and dr = 0.1. The streamlines shown in Fig.-5 illustrate the interaction of forced and natural convection under various convection regimes. For Ri = 0.1-1.0, the major incoming flow is symmetric about the diagonal joining from the inlet to the exit port and a small vortex is developed near the left insulated wall starting from just below the inlet port, due to the domination of forced convection as shown in Fig. 5. At Ri = 5.0, the size of the vortex is increased dramatically and changes its pattern from a uni-cellular vortex to a bi-cellular vortices, which occupies much of the cavity as shown in Fig. 5. This is because the buoyancy force dominates the forced flow in the cavity. Further increase of Ri at 10.0, the patterns of the streamlines are about the same as those for Ri = 5.0, but, a careful observation indicates that the inner vortex become larger slightly in size and stronger in strength compared this with the before one, because the effect of free convection on heat transfer and flow increases with increasing Ri. From Figures-5 it has been seen that the isothermal lines are nearly parallel to the bottom heated wall for Ri = 0.1, this indicating a dominant heat conduction mechanism. For the larger Ri (Ri = 5.0, 10.0) the high temperature region become more strenuous near the bottom heated wall as result the thermal boundary layer become thinner.

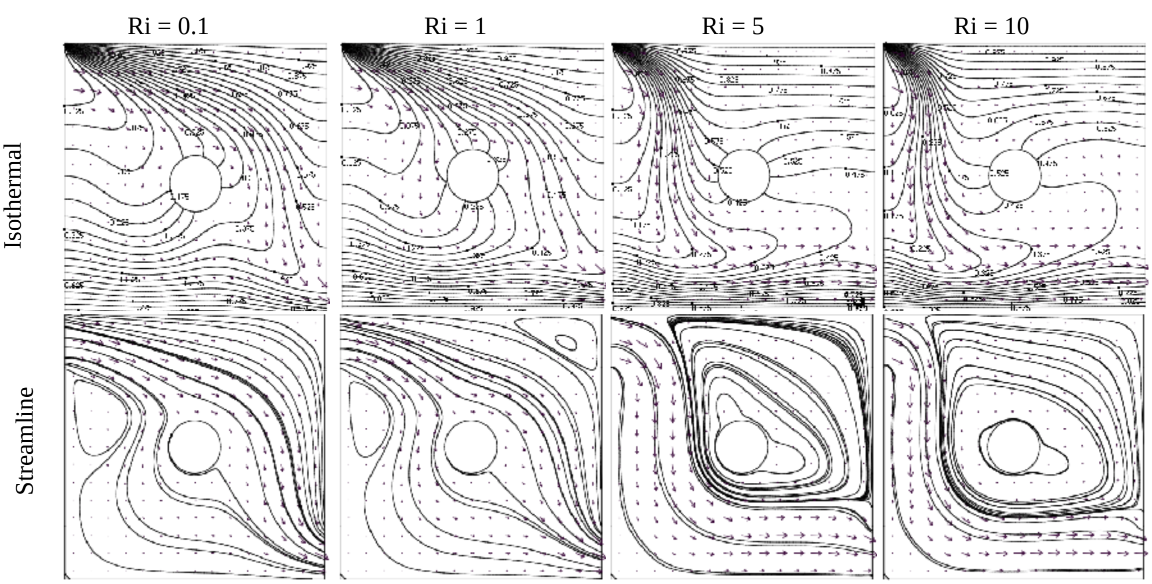


Fig. 5. Isotherms and Streamlines for the TB configuration at different diameter ratio (dr) While Re = 100, Ri = 1.0 and $L_x = L_y = 0.5$.

D. Effect of Prandtl number

In fig.6 display that the effect of Prandtl number on streamlines as well as isotherms for the TB configuration at AR = 1.0, Re = 100, Ri = 1.0 and dr = 0. At Pr = 0.71, The streamlines has been affected by the buoyancy force as a result some counter circulation flow found near the left-top and right-top corner in the cavity as shown in fig-6. This recirculation region increases with increasing Prandtl numbers as shown in Fig 6.

For the isotherms, the figures show that as the increase of Pr then the buoyancy force increases and the thermal boundary layers become thinner with heated wall. The most importance effect of Pr, the entering flow occupies more places in the cavity from inlet to outlet port as a result the isotherm lines become more packed to heated wall for different Pr. With increase of Pr, The isotherm lines moves towards the both heated wall. The region of upper and lower side of the cylinder can be considered as the thermally influenced region of the fluid by the heated wall where convective intensified.

The deviation of average Nusselt number (Nu_{av}) at the heated wall in the cavity along with Richardson number for different Prandtl numbers has been presented in Fig. 8. From this figure it is clearly seen that for a particular values of Ri, the average Nusselt number is the highest for the large Prandtl number Pr = 7. This is because, the fluid with the highest Prandtl number is capable to carried more heat away from the heat source and dissipated through the out flow opening in the cavity.

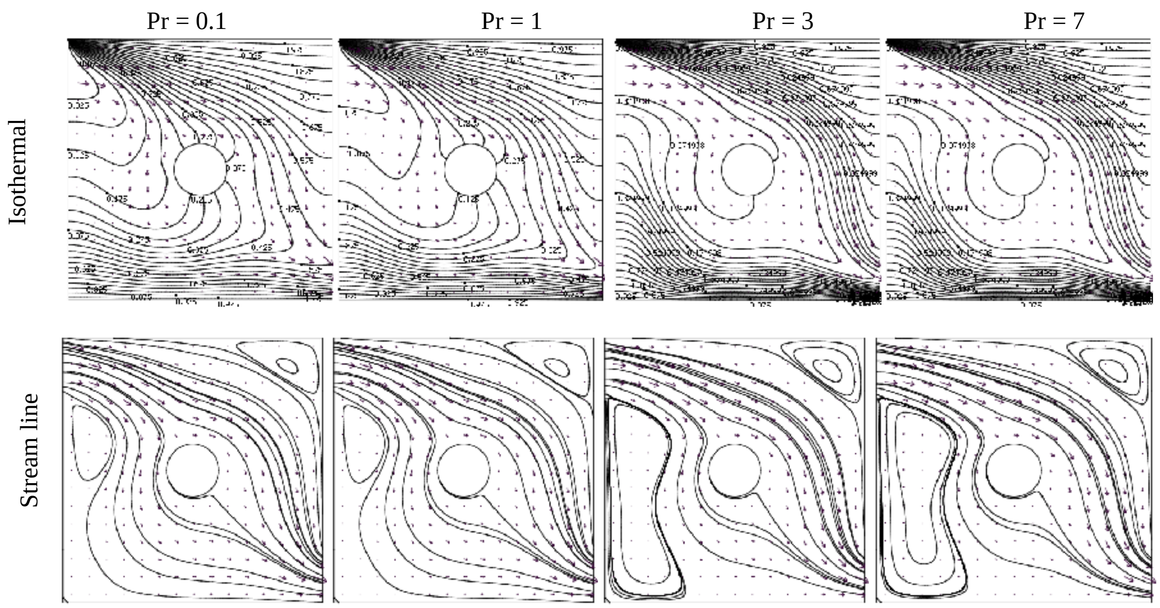


Fig. 6. Isotherms and Streamlines for the TB configuration at different Pr While dr = 1.0, Ri = 1.0 and $L_x = L_y = 0.5$.

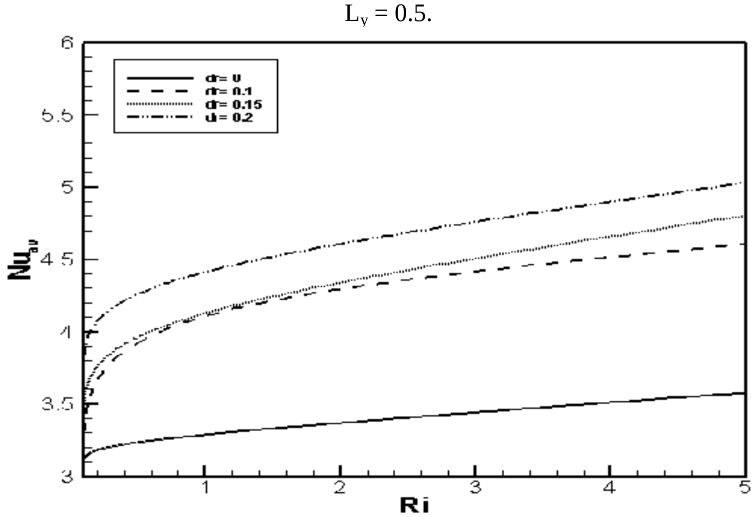


Fig. 7. Effect of cylinder diameter on Nu_{av} for the TB configuration While Re = 100, Ri = 1.0 and Pr = 0.71.

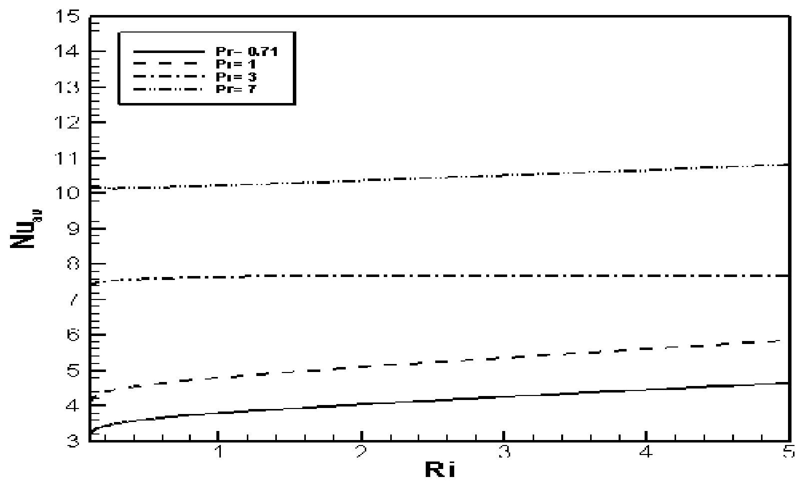


Fig. 8. Effect of Pr on Nu_{av} for the TB configuration While Re = 100, Ri = 1.0.

VI. CONCLUSION

A finite element method for steady-state incompressible mixed convection flow is presented. The finite element equations has been derived from the governing flow equations that consist of the conservation of mass, momentum, and energy equations. The derived finite element equations are nonlinear requiring an iterative technique solver. The Galerkin weighted residual method is applied to solve these nonlinear equations for solutions of the nodal velocity components, temperatures, and pressures. The above example demonstrates the capability of the finite element formulation that can provide insight to steady-state incompressible mixed convection flow behaviors.

In view of the obtained results, the following findings have been summarized:

(i) The Stream line distribution in the cavity is strongly affects by the cylinder diameter.

For that reason, buoyancy-induced circulation cell reduces with increasing cylinder diameter. The effect of diameter is comparatively small on the isotherm in the cavity. The rate of heat transfer is highest for dr = 2.0 at TB configuration of cavity.

(ii) A significant effect is observed for forced convection parameter *Re* on the fluid flow

and temperature fields in the cavity. With the increasing of *Re* the thermal boundary layer is reduce and intense where as vortex in the streamlines increased. At the heated surface the average Nusselt numbers is always the highest for the large value of *Re*.

(iii) The fluid flow and temperature fields have been affected by the mixed convection parameter *Ri* in the cavity. With the increasing *Ri* the concentrated thermal layer near the heated surface become thin and the recirculation cell become large in the streamline.

(iv) The effect of Prandtl number on isotherms and streamlines are notable for the

different values of *Pr*.The average Nusselt numbe at the hot wall is increased with the increasing value of *Pr*.

ACKNOWLEDGEMENTS

Authors would like to thank to the Department of Mathematics, Bangladesh University of Business & Technology (BUBT), for providing computer facility during this work. I also express my gratitude to anonymous reviewers for their suggestions which improved the quality of the manuscript.

REFERENCES

[1] Petersen, G.P., Ortega, A., Thermal control of electronic equipment and devices. *Adv. Heat Transfer*, Vol.**20**, pp.181 (1990).

[2] Hsu, T.H., Hsu, P.T., How, S.P., Mixed convection in a partially divided rectangular enclosure, *Numerical Heat Transfer, Part A (Applications)* Vol. **31** (6), pp 655(1997).

[3] A. Omri, S.B. Nasrallah, Control volume finite element numerical simulation of mixed convection in an air-cooled cavity, *Numerical Heat Tr. A-Appl.*,Vol. **36**, pp. 615 (1999).

[4] S. Singh, M.A.R. Sharif, Mixed convection cooling of a rectangular cavity with inlet and exit openings on differentially heated side walls, *Numerical Heat Tr. A-Appl.* Vol.**44**, PP. 233(2003).

[5] O. Manca, S. Nardini, K. Khanafer, K. Vafai, Effect of heated wall position on mixedconvection in a channel with an open cavity, *Numerical Heat Tr. A-Appl.*, Vol.**43**, pp. 259(2003).

[6] M.M. Rahman, M.A. Alim, M.A.H.Mamun, Finite Element Analysis of Mixed convectionin a Rectangular Cavity with a Heat-conduvting Horizontal Circular Cylinder, Nonlinear Analysis: *Modelling and Control*, Vol. **14**(2), pp. 217 (2009).

[7] S.Z. Shuja, B.S. Yilbas, M.O. Iqbal, Mixed convection in a square cavity due to heat generating rectangular body effect of cavity exit port locations, *Int. J. of Numerical Methods for Heat and Fluid Flow*, Vol. **10**(8), pp. 824 (2000).

[8] J.M. House, C. Beckermann, T.F. Smith, Effect of a centered conducting body on natural convection heat transfer in an enclosure, *Numerical Heat Tr. A-Appl.*, Vol.**18**, pp. 213(1990).

[9] P. E. Lewis and J. P. Ward, The Finite Element Method, Principles and Applications, (Addison-Wesley, 1991).

[10] N.S.Ottosen, Introduction to the Finite Element Method, (Prentice Hall International, UK, 1992).

[11] C. Taylor, P. Hood, A numerical solution of the Navier-Stokes equations using finite element technique, I, *Computer and Fluids*, Vol.**1**, pp. 73(1973).

[12] Dechaumphai, P., Adaptive finite element technique for heat transfer problems, *Heat & Mass transfer,* Vol.**17**, pp.87 (1995).