Effect of the Presence of a Heat Conducting Horizontal Square Block on Mixed Convection inside a Vented Square Cavity

Md. M. Rahman\textsuperscript{1}, M. A. Alim\textsuperscript{1}, S. Saha\textsuperscript{2}, M. K. Chowdhury\textsuperscript{1}

\textsuperscript{1}Department of Mathematics
Bangladesh University of Engineering and Technology (BUET)
Dhaka-1000, Bangladesh
mmustafizurrahman@math.buet.ac.bd

\textsuperscript{2}Department of Mechanical Engineering
The University of Melbourne
Victoria 3010, Australia

Received: 2009-03-16 Revised: 2009-08-15 Published online: 2009-11-11

Abstract. Finite element method is used to solve two-dimensional governing mass, momentum and energy equations for steady state, mixed convection problem inside a vented square cavity. The cavity consists of adiabatic left, top and bottom walls and heated right vertical wall; but it also contains a heat conducting horizontal square block located somewhere inside the cavity. Forced flow conditions are imposed by providing an inlet at the bottom of the left wall and an exit at the top of the right wall, through which the working fluid escape out of the cavity. The aim of the study is to describe the effect of such block on the flow and thermal fields. The investigations are conducted for various values of geometric size, location and thermal conductivity of the block under constant Reynolds and Prandtl numbers. Various results such as the streamlines, isotherms, heat transfer rates in terms of the average Nusselt number, average fluid temperature in the cavity and the temperature at the center of solid block are presented for different parameters. It is observed that the block size and location have significant effect on both the flow and thermal fields but the solid-fluid thermal conductivity ratio has insignificant effect on the flow field. The results also indicate that the average Nusselt number at the heated surface, the average temperature of the fluid inside the cavity and the temperature at the center of solid block are strongly dependent on the configurations of the system studied under different geometrical and physical conditions.

Keywords: finite element method, square block, vented cavity and mixed convection.

Nomenclature

\begin{align*}
d & \quad \text{dimensional length of the block [m]} \\
D & \quad \text{dimensionless length of the block} \\
g & \quad \text{gravitational acceleration [ms}^{-2}\text{]} \\
h & \quad \text{convective heat transfer coefficient}
\end{align*}
Efficient convection heat transfers are essential in modern technology and also very important in many industrial areas. Hence, it is necessary to study and simulate these phenomena. Several numerical and experimental methods have been developed to investigate cavities with and without obstacle because these geometries have practical engineering and industrial applications, such as in the design of solar collectors, thermal design of building, air conditioning, cooling of electronic devices, furnaces, lubrication technologies, chemical processing equipment, drying technologies etc. Analysis of above phenomena, incorporating a solid heat conducting obstruction extends its usability to practical situations. Particularly, a conductive material in an inert atmosphere inside a furnace with a constant flow of gas from outside and the cooling of electronic circuit boards constitutes some practical applications for the present study. Conjugate natural convection heat transfer inside an inclined square cavity with an internal conducting block was studied by Das and Reddy [1]. At the same time, Zhao et al. [2] numerically investigated conjugate natural convection in enclosures with external and internal heat
sources and Xu et al. [3] experimentally observed the thermal flow around a square obstruction on a vertical wall in a differentially heated cavity. Later on, Bhoite et al. [4] performed numerical investigation on 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 and Grashof numbers and block-to-fluid thermal conductivity ratios. They showed that higher Reynolds numbers created a recirculation 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. Braga and de Lemos [5] investigated steady laminar natural convection within a square cavity filled with a fixed volume of conducting solid material consisting of either circular or square obstacles. They used finite volume method with a collocated grid to solve governing equations. They found 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 [6]. 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. Bilgen and Yamane [7] examined 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. Dong and Li [8] studied conjugate effect of natural convection and conduction in a complicated enclosure. They observed the influences of material character, geometrical shape and Rayleigh number on the heat transfer in the overall concerned region. They finally concluded that the flow and heat transfer increased with the increase of thermal conductivity in the solid region and besides, both geometric shape and Rayleigh number also affected the overall flow and heat transfer greatly. Roychowdhury et al. [9] analyzed the natural convective flow and heat transfer features for a heated cylinder placed in a square enclosure with different thermal boundary conditions. House et al. [10] studied the effect of a centered, square, heat conducting body on natural convection in a vertical enclosure. They showed 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. [11], where the conducting body generated heat within the cavity. Under these situations, it was shown that the flow was driven by a temperature difference across the cavity and a temperature difference caused by the heat-generating source. Very recently Rahman et al. [12] analyzed mixed convection in a rectangular cavity with a heat conducting horizontal circular cylinder by using finite element method.

In the light of the above literature, it has been pointed out that there is no significant information about mixed convection processes when a heat conducting square block exists within a vented cavity in different locations. The purpose of the present study is to examine how the size, location and thermal conductivity of the inner heat-conducting block affect the mixed convection phenomena within the vented cavity.
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 square heat conducting block with length \( d \) and thermal conductivity \( k_s \) is located somewhere \((l_x, l_y)\) within the cavity. A Cartesian co-ordinate system is used with origin at the lower left corner of the computational domain. The top, bottom and left vertical walls of the cavity are kept adiabatic and the right vertical wall is kept at a uniform constant temperature, \( T_h \). The inflow opening located on the bottom of the left wall and the outflow opening of the same size is placed at the top of the opposite heated wall as shown in Fig. 1. For simplicity, the size of the two openings, \( w \) is set equal to the one-tenth of the cavity length \( (L) \). Cold air flows through the inlet inside the cavity at a uniform velocity, \( u_i \). It is also assumed that the incoming flow is at the ambient temperature \( T_i \) and the outgoing flow is assumed to have zero diffusion flux for all dependent variables i.e. convective boundary conditions (CBC). All solid boundaries are assumed to be rigid no-slip walls. Emphasis is placed on the effect of the various orientations and dimension of the heat-conducting block.

![Fig. 1. Schematic of the problem with the domain and boundary conditions.](image)

3 Mathematical model

In the present problem, it can be considered that the flow is steady, two-dimensional, laminar incompressible and there is no viscous dissipation. The gravity acts in the vertically downward direction, fluid properties are constant and fluid density variations are neglected except in the buoyancy term (Boussinesq approximation) and radiation effect is neglected.

Using non-dimensional variables defined below, the non-dimensional forms of the
Effect of the Presence of a Heat Conducting Horizontal Square Block

governing equations of the present problem are obtained as follows:

\[ \frac{\partial U}{\partial X} + \frac{\partial V}{\partial Y} = 0, \]

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

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

\[ U \frac{\partial \theta}{\partial X} + V \frac{\partial \theta}{\partial Y} = \frac{1}{Re Pr} \left( \frac{\partial^2 \theta}{\partial X^2} + \frac{\partial^2 \theta}{\partial Y^2} \right), \]

For heat conducting block, the energy equation is

\[ \frac{\partial^2 \theta_s}{\partial X^2} + \frac{\partial^2 \theta_s}{\partial Y^2} = 0. \]

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

\[ X = \frac{x}{L}, \quad Y = \frac{y}{L}, \quad U = \frac{u}{u_i}, \quad V = \frac{v}{u_i}, \]

\[ P = \frac{p}{\rho u_i^2}, \quad D = \frac{d}{L}, \quad L_x = \frac{l_x}{L}, \quad L_y = \frac{l_y}{L}, \]

\[ \theta = \frac{T - T_i}{T_h - T_i}, \quad \theta_s = \frac{T_s - T_i}{T_h - T_i}, \]

and the parameters \( Re, Ri, Pr \) and \( K \) are defined as

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

The appropriate dimensionless form of the boundary conditions used to solve equations (1)–(5) inside the cavity are given as:

\[ U = 1, \quad V = 0, \quad \theta = 0 \] at the inlet;

\[ P = 0 \] convective boundary condition (CBC) at the outlet;

\[ U = 0, \quad V = 0 \] at all solid boundaries;

\[ \theta = 1 \] at the heated right vertical wall;

\[ \left. \frac{\partial \theta}{\partial X} \right|_{X=0} = \left. \frac{\partial \theta}{\partial Y} \right|_{Y=1.0} \] at the left, top and bottom walls;

\[ \left. \frac{\partial \theta}{\partial X} \right|_{fluid} = K \left. \frac{\partial \theta}{\partial X} \right|_{solid} \] at the solid-fluid vertical interfaces of the block;

\[ \left. \frac{\partial \theta}{\partial Y} \right|_{fluid} = K \left. \frac{\partial \theta}{\partial Y} \right|_{solid} \] at the solid-fluid horizontal interfaces of the block.
The average Nusselt number ($N_u$) at the hot wall is defined as

$$N_u = \frac{1}{L_h} \int_0^{\frac{L_h}{L}} \frac{\partial \theta}{\partial X} \bigg|_{X=1} dY$$

and the bulk average temperature in the cavity is defined as

$$\theta_{av} = \frac{1}{\bar{V}} \int \theta d\bar{V},$$

where $L_h = L - 0.1/L$ is the length of the hot 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 [13] and Dechaumphai [14]. 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. The integration involved in each term of these equations is performed by using Gauss quadrature method. 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.

### 4.1 Grid refinement check

Five different grid sizes of 3976, 4798, 6158, 6278 and 7724 elements with 25555, 30619, 38973, 39870, and 48945 nodes respectively are chosen for the present simulation to test the independency of the results with the grid size variations. Average Nusselt number at the heated surface, average temperature of the fluid inside the cavity and the solution time are monitored at $Ri = 1.0$, $L_x = L_y = 0.5$, $D = 0.2$ and $K = 5.0$ for these grid elements as shown in Table 1.

<table>
<thead>
<tr>
<th>Elements (Nodes)</th>
<th>3976 (25555)</th>
<th>4798 (30619)</th>
<th>6158 (38973)</th>
<th>6278 (39870)</th>
<th>7724 (48945)</th>
</tr>
</thead>
<tbody>
<tr>
<td>$N_u$</td>
<td>4.8463</td>
<td>4.8478</td>
<td>4.8488</td>
<td>4.8489</td>
<td>4.8489</td>
</tr>
<tr>
<td>$\theta_{av}$</td>
<td>0.1905</td>
<td>0.1905</td>
<td>0.1905</td>
<td>0.1905</td>
<td>0.1905</td>
</tr>
<tr>
<td>Time [s]</td>
<td>385.219</td>
<td>493.235</td>
<td>682.985</td>
<td>698.703</td>
<td>927.359</td>
</tr>
</tbody>
</table>
The magnitude of average Nusselt number at the heated surface and average temperature of the fluid inside the cavity for 39870 nodes with 6278 elements shows a very little difference with the results obtained for the other denser grids. Hence for the rest of the calculation in this study, a grid size of 39870 nodes with 6278 elements is chosen for better accuracy.

4.2 Code validation
For code validation tests, see Rahman et al. [12].

5 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 a horizontal square solid block. Effect of the parameters such as Richardson number (\(Ri\)), dimensionless block length (\(D\)), solid-fluid thermal conductivity ratio (\(K\)) and the location of the solid block (\(L_x, L_y\)) on heat transfer and fluid flow of the cavity have analyzed. We have presented the results in two sections. The first section has focused on flow and temperature fields, which contains streamlines and isotherms for the different cases. The following section has discussed heat transfer including average Nusselt numbers at the hot wall, average fluid temperature in the cavity and temperature at the block center. The range of \(Ri\) for this investigation has varied from 0.0 to 5.0 by changing \(Gr\) while keeping \(Re\) fixed at 100. Air is chosen as working fluid with \(Pr = 0.71\).

5.1 Flow and temperature fields
Flow and temperature fields have simulated using streamlines and isotherms for the mentioned parameters. Effect of block size on streamlines and isotherms have presented in Figs. 2 and 3 for \(K = 5.0, L_x = L_y = 0.5\) and various \(Ri\) (0.0, 1.0 and 5.0). The flow structure in the absence of free convection effect (i.e. \(Ri = 0\)) and for the four different values of \(D\) has shown in the left column of Fig. 2. At \(Ri = 0.0\) and \(D = 0.0\), it has been seen that a comparatively large uni-cellular vortex appears at the left top corner of the cavity and a very small vortex appears at the right bottom corner of the cavity, which are owing to the effect of buoyancy force. Further increase of \(D\) sharply decreases the size of the vortex. This is due to increasing the size of the block gives rise to a decrease in the space available for the buoyancy force induced vortex. For \(Ri = 1.0\) and \(D = 0.0\), it has been also seen from Fig. 2 that the natural convection effect is present, but remains relatively weak at high values of \(D\), since the open lines characterizing the imposed flow are still dominant. Further increase of \(Ri\) to 5.0, gradually increase the size of the vortex for \(D = 0.0\). This expansion of the size of the vortex squeezes the induced forced flow path resulting almost same kinetic energy in the bulk-induced flow as that of the inlet port. It must be noticed that in this case the size of the vortex reduced dramatically at the highest value of \(D = 0.6\). The isotherms in the absence of block (\(D = 0.0\)) and for the three values of \(Ri\) are shown in the bottom row of Fig. 3.
Fig. 2. Streamlines for different block length and Richardson numbers, while $L_x = L_y = 0.5$ and $K = 5.0$. 
Fig. 3. Isotherms for different block length and Richardson numbers, while $L_x = L_y = 0.5$ and $K = 5.0$. 

539
At $Ri = 0.0$ and $D = 0.0$, the high temperature region is concentrated near the hot wall and the isothermal lines are linear and parallel to the heated surface in the cavity, indicating conduction and forced convection dominant heat transfer. On the other hand, the concentrated temperature region become thin and the isothermal lines become nonlinear for $Ri = 1.0$ and various values of $D$. Further $Ri$ increases to 5.0, the nonlinearity in the isotherms become higher and plume formation is profound, indicating the well established natural convection. As we compared the isothermal lines for $D = 0.2$ and various $Ri$ with the isothermal lines for $D = 0.0$ and various $Ri$, only small difference in isotherms is observed. Further increasing the length of the block gives the higher nonlinearity in the isothermal lines of the cavity.

The effect of the solid-fluid thermal conductivity ratio ($K$) on streamlines and isotherms for $D = 0.2$ and $L_x = L_y = 0.5$ and various $Ri$ have presented in Figs. 4 and 5 respectively. At $Ri = 0.0$ and $K = 0.2$, the bulk induced flow expands in the cavity resulting increase in potential energy. Here a small recirculation cell is formed just at the top of the inlet port of the cavity. The streamlines for the other cases (i.e. $K = 1.0, 5.0, 10.0$) at $Ri = 0.0$ are almost identical. Further as $Ri$ increases, the vortex spreads. As a consequence, the induced flow is squeezed and the vortex covers the cavity, indicating the supremacy of natural convection in the cavity. Again at $Ri = 0.0$ and $K = 0.2$, the isothermal lines are almost parallel and concentrated to the hot surface as shown in Fig. 5. Making a comparison of the isothermal lines for $Ri = 0.0$ and various $K$, no significant difference is found except that the isothermal lines at higher $K$ are shifted from the center of the block. As $Ri$ increases for a permanent $K$, nonlinearity of the isotherms becomes higher and plume formation is philosophical, indicating the well established natural convection heat transfer.

The distribution of streamlines and isothermal lines for various locations of the square block at $Ri = 0.0, 1.0$ and 5.0, while $D = 0.2$ and $K = 5.0$ have shown in Figs. 6 and 7. When the inner block is placed at the center $(0.25, 0.5)$ as shown in the bottom row of the Fig. 6, a bi-cellular vortex is seen just above the inlet port and occupies the left-top portion of the cavity and a pocket of fluid formed at the right bottom corner in the cavity for $Ri = 0.0$. Further at $Ri = 1.0$ the bi-cellular vortex spreads and the pocket of fluid the right bottom corner in the cavity is out. Also the flow changes its pattern from bi-cellular vortex to a uni-cellular vortex at $Ri = 5.0$. The corresponding isotherms for the lower values of $Ri$ are uniformly distributed around the heat source, display that the heat is mainly transport by diffusion due to weak buoyancy flow. The distribution of isotherms in the cavity at $Ri = 5.0$ is significantly different from that at the lower values of $Ri$, because the buoyancy induced convection becomes more predominant than conduction. When the inner block moves closer to the heated surface along the mid-horizontal plane, the pattern of vortex located just above the inlet port in the cavity is also uni-cellular. The distribution of the isotherms for different $Ri$ shows a similar pattern to the case, where block moves near the left wall along the mid-horizontal plane for different $Ri$. However, when the inner block moves closer to the bottom wall along the mid-vertical plane a very small uni-cellular vortex is formed just the top of the inlet at $Ri = 0.0$ and 1.0, but for $Ri = 5.0$, the flow pattern has changed drastically from a very small vortex into two large vortices and thereby squeezes the induced flow path due to the supremacy of natural
Fig. 4. Streamlines for different thermal conductivity ratios and Richardson numbers, while $L_x = L_y = 0.5$ and $D = 0.2$. 

$K = 10.0$  

$K = 5.0$  

$K = 1.0$  

$K = 0.2$  

$Ri = 0.0$  

$Ri = 1.0$  

$Ri = 5.0$
Fig. 5. Isotherms for different thermal conductivity ratios and Richardson numbers, while $L_x = L_y = 0.5$ and $D = 0.2$. 
Effect of the Presence of a Heat Conducting Horizontal Square Block

Fig. 6. Streamlines for different locations of the block and Richardson numbers, while $K = 5.0$ and $D = 0.2$. 

$K = 5.0$ and $D = 0.2$. 

543
Fig. 7. Isotherms for different locations of the block and Richardson numbers while $K = 5.0$ and $D = 0.2$. 
convection in the cavity. The isothermal lines in this case are more concentrated and vertical at the heat source for $Ri = 0.0, 1.0$, due to weak buoyancy flow. As $Ri$ increases to 5.0, the isotherms become nonlinear and plume is formed which are the cryptogram of strong natural convection heat transfer. Also when the inner block moves closer to the top wall along the mid-vertical plane a uni-cellular vortex has formed just over the inlet port and occupies the left-top portion in the cavity and a pocket of fluid formed at the right bottom corner in the cavity for $Ri = 0.0$. Further increase of $Ri$ gradually develops the size of the vortex, located at the left top corner of the cavity and diminishes the pocket of the fluid at the right bottom corner in the cavity. The isothermal lines surrounding the heat source seem to have no significant difference that of as the block moves closer to the left wall along the mid-vertical plane.

### 5.2 Heat transfer

Plots of the average Nusselt number ($Nu$) at the heated wall, average temperature ($\theta_{av}$) of the fluid in the cavity and the dimensionless temperature ($\theta_c$) at the block center as a function of $Ri$ and $D$ have shown in Fig.8. As $Ri$ increases, average Nusselt number ($Nu$) at the heated wall increases monotonically for all values of $D$, which is due to increasing $Ri$ enhances convective heat transfer. 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 ($0.0 \leq Ri \leq 1.0$) and for $D = 0.4$ in the free convection dominated region. It has been seen that the average temperature ($\theta_{av}$) of the fluid in the cavity and the temperature ($\theta_c$) at the block center increase with increasing $Ri$ for a particular values of $D$. On the other hand, the average temperature ($\theta_{av}$) of the fluid in the cavity and the temperature ($\theta_c$) at the block center increase with increasing $D$ for a particular value of $Ri$. This can be attributed to the fact that a large centered square block narrows the regions available for both the warm and cold fluid flows.

![Figure 8](image.png)

**Fig. 8.** Effect of block length on (a) average Nusselt number; (b) average fluid temperature and (c) temperature at the block center for various $Ri$, while $L_x = L_y = 0.5$ and $K = 5.0$.  

545
The effect of Richardson number on the average Nusselt number ($Nu$) at the heated surface, average temperature of the fluid in the cavity and temperature at the block center for different solid-fluid thermal conductivity ratio has shown in Fig. 9. As $Ri$ increases, average Nusselt number ($Nu$) at the hot surface sharply increases for all values of $K$. A careful observation on Fig. 9(a) shows that $Nu$ is the highest for $K = 5.0$ and $10.0$ at $Ri \leq 0.4$ and for $K = 1.0$ at $0.4 < Ri < 1.0$. Beyond these values of $Ri$ it is the highest for $K = 0.2$. As $Ri$ increases average temperature of the fluid and the temperature at the block center increases gradually for all values of $K$ in the cavity. On the other hand, for a particular value of $Ri$ the average temperature ($\theta_{av}$) of the fluid in the cavity and the temperature ($\theta_c$) at the block center is always the lowest for $K = 0.2$. This scenario occurs because the block with $K = 0.2$ acts as an insulator and prevents heat transfer between the hot and the cold fluid streams.

Fig. 9. Effect of thermal conductivity ratio on (a) average Nusselt number; (b) average fluid temperature and (c) temperature at the block center for various $Ri$, while $L_x = L_y = 0.5$ and $D = 0.6$.

Fig. 10. Effect of the locations of the block on (a) average Nusselt number; (b) average fluid temperature and (c) temperature at the block center for various $Ri$, while $K = 5.0$ and $D = 0.2$. 

546
The average Nusselt number \( (Nu) \) at the heated surface, the average temperature \( (\theta_{av}) \) of the fluid and the temperature \( (\theta_c) \) at the block center in the cavity are plotted against Richardson numbers and for the four different locations of the block have shown in Fig. 10. From these figures it has seen that the average Nusselt number smoothly increases with increasing Ri and the average temperature of the fluid and the temperature at the block center in the cavity are not monotonic with increasing Ri for different locations of the solid block.

6 Conclusion

A numerical investigation is made of laminar mixed-convective in a square cavity with a heat conducting horizontal square block. Results are obtained for wide ranges of parameters Richardson number \( (Ri) \), dimensionless block length \( (D) \), solid-fluid thermal conductivity ratio \( (K) \) and the location of the solid block \( (L_x, L_y) \).

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

- Block size affects strongly the streamline distribution in the cavity. As a result, buoyancy forced-induced circulation cell reduces with increasing block size. Comparatively small effect on the isotherms is observed for different block size. The average Nusselt number at the heated surface is the highest for the largest block length \( D = 0.6 \) in the forced convection dominated region and for the second largest length \( D = 0.4 \) in the free convection dominated region. On the other hand, a gradual increase in the heat transfer rate is found with increasing \( Ri \) at constant values of block length.

- Material properties \( (K) \) have insignificant effect on the flow field and have significant effect on the thermal fields. An unexpected result is found for the dependence of thermal transport on the solid-fluid thermal conductivity ratio. For somewhat large solid block an obvious enhancement in the heat transfer is obtained for the largest values of \( K \) in the forced convection dominated region and for the lowest value of \( K \) in the free convection dominated region. However, a steady increase in the heat transfer rate is found with increasing \( Ri \) at constant values of \( K \).

- Locations of the block have significant effect on the flow and thermal fields. The value of average Nusselt number is the highest in the forced convection dominated area when the block is located near the top wall along the mid-vertical plane and in the free convection dominated area when the block moves closure to the left vertical wall along the mid-horizontal plane. On the other hand, a gradual increase in the heat transfer rate has found with increasing \( Ri \) at constant values of block length. The values of \( \theta_{av} \) and \( \theta_c \) are not monotonic with increasing values of \( Ri \) for different locations of the block.

References


