MIXED CONVECTION HEAT TRANSFER OF AIR INSIDE A SQUARE VENTED CAVITY WITH A HEATED HORIZONTAL SQUARE CYLINDER

Ali J. Chamkha¹, Salam Hadi Hussain², and Qusay Rashid Abd-Amer²

¹Manufacturing Engineering Department, The Public Authority for Applied Education and Training, Shuweikh, Kuwait
²Mechanical Engineering Department, College of Engineering, Babylon University, Babylon, Province, Iraq

The study of two-dimensional mixed convection from a heated square solid cylinder located at the center of a vented cavity filled with air (Pr = 0.71) is performed. The horizontal walls and the right vertical wall are kept adiabatic, while the left vertical wall is maintained at a constant temperature $T_c$. The flow is assumed to be laminar, steady and of constant physical properties except for the density in the buoyancy term, which follows the Boussinesq approximation. The developed mathematical model is governed by the coupled equations of continuity, momentum, and energy and is solved by the finite-difference method. This study investigates the effects of the outlet positions, Richardson numbers, Reynolds numbers, locations, and aspect ratios of inner square cylinder on the flow and the thermal fields. The parameters studied and their ranges are as follows: the Richardson number and Reynolds number are varied in the ranges from 0–10 and 50–200, respectively, location of the inner cylinder $(0.25 \leq L_x \leq 0.75, \ 0.5 \leq L_y \leq 0.75)$, and aspect ratios of 0.1, 0.2, 0.3, and 0.4. The present results show that the average Nusselt number along the heated surface of the inner square values increases with increasing values of the Reynolds and Richardson numbers. The effect of the locations of the inner square cylinder and aspect ratio is found to play a significant role in the streamline and isotherm patterns. An empirical correlation is developed by using the Nusselt number and the Reynolds and Richardson numbers. The results are compared and verified with previously published results, and good agreement is achieved.

1. INTRODUCTION

The mixed convection region is a transitional heat transfer region between forced convection and natural convection, where both the forced component of flow and the buoyancy-induced component are important. The parameter of importance in mixed convection is the Richardson number (Ri = Gr/Re²). The heat transfer mechanism is predominantly forced convection or free convection according to $\text{Ri} \ll 1$ or $\text{Ri} \gg 1$. When $\text{Ri} \approx 1$, both mechanisms become important. The

Received 3 June 2010; accepted 3 October 2010.
Address correspondence to Ali J. Chamkha, Manufacturing Engineering Department, The Public Authority for Applied Education and Training, P.O. Box 42325, Shuweikh 70654, Kuwait. E-mail: achamkha@yahoo.com

58
phenomenon of mixed convection in a cavity is widely used in engineering applications such as heat exchangers, nuclear and chemical reactors, and food processing, but an important popular field of application is in the design of cooling systems.

Oosthuizen and Paul [1] have investigated mixed convection heat transfer in a cavity with uniformly heated, isothermal vertical walls and horizontal adiabatic walls. A forced flow, with an either aiding or opposing effect on the buoyancy, enters and leaves the enclosure across the cold wall. The heat transfer results obtained at various values of Grashof number and the aspect ratio of the cavity are compared with the pure natural convection case. Abu-Hijleh and Heilen [2] have studied numerically the Nusselt number due to laminar mixed convection from rotating isothermal cylinder. The study covers a wide range of parameters: $5 \leq \text{Re} \leq 450$ and $0.1 \leq K \leq 10$. A correlation was proposed which can be used to accurately predict the Nusselt number for the range of parameters studied. Dagtekin and Oztop [3] inserted an isothermally heated rectangular block in a lid-driven cavity at different positions to simulate the cooling of electronic equipment. They found that the dimension of the body is the most effective parameter on mixed convection flow. Misirlioglu [4] investigated numerically the heat transfer in a square cavity with a rotating circular cylinder centered within it and filled with clear fluid or porous medium. The results indicate that the rotating direction of the cylinder has a significant effect on the natural and forced convection regimes, especially in the clear fluid case. The influence of mixed convection from a heat generating element in a ventilated cavity has been studied experimentally and numerically by Radhakrishnan et al. [5].

<table>
<thead>
<tr>
<th>NOMENCLATURE</th>
</tr>
</thead>
<tbody>
<tr>
<td>$AR$</td>
</tr>
<tr>
<td>$d$</td>
</tr>
<tr>
<td>$g$</td>
</tr>
<tr>
<td>$hi$</td>
</tr>
<tr>
<td>$k$</td>
</tr>
<tr>
<td>$K$</td>
</tr>
<tr>
<td>$L$</td>
</tr>
<tr>
<td>$L_X$</td>
</tr>
<tr>
<td>$L_Y$</td>
</tr>
<tr>
<td>$Nu$</td>
</tr>
<tr>
<td>$Nu_s$</td>
</tr>
<tr>
<td>$P$</td>
</tr>
<tr>
<td>$Pr$</td>
</tr>
<tr>
<td>$p$</td>
</tr>
<tr>
<td>$Re$</td>
</tr>
<tr>
<td>$Ri$</td>
</tr>
<tr>
<td>$T$</td>
</tr>
<tr>
<td>$U$</td>
</tr>
<tr>
<td>$V$</td>
</tr>
<tr>
<td>$X, Y$</td>
</tr>
<tr>
<td>$\alpha$</td>
</tr>
<tr>
<td>$\beta$</td>
</tr>
<tr>
<td>$\varepsilon$</td>
</tr>
<tr>
<td>$\theta$</td>
</tr>
<tr>
<td>$\rho$</td>
</tr>
<tr>
<td>$\Psi$</td>
</tr>
<tr>
<td>$\Phi$</td>
</tr>
<tr>
<td>$\Omega$</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Subscripts</th>
</tr>
</thead>
<tbody>
<tr>
<td>$av$</td>
</tr>
<tr>
<td>$h$</td>
</tr>
<tr>
<td>$c$</td>
</tr>
<tr>
<td>$i$</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Abbreviation</th>
</tr>
</thead>
<tbody>
<tr>
<td>CBC</td>
</tr>
</tbody>
</table>
Experiments were carried out on heaters of two different sizes, located centrally in a parallelepiped that has an air inlet and outlet ports. A numerical investigation of fluid flow and heat transfer from the heater for exactly the same configuration was also carried out. The effect of the orientation of the heater in horizontal and inclined positions was also studied numerically. Further studies on insertion of baffles on the walls of the cavity to improve the thermal performance and placement of multiple heaters are in progress. Laskowski et al. [6] studied both experimentally and numerically heat transfer to and from a circular cylinder in a cross-flow of water at low Reynolds number. The results show that when the lower surface was unheated, the temperatures of the lower surface and water upstream of the cylinder were maintained approximately equal and the flow was laminar. Buonomo et al. [7] used the mixed convection flow in open cavities filled with a porous medium with one heated wall at uniform heat flux. The investigated configurations were a horizontal channel with a cavity at the lower wall (U-shaped cavity); and a vertical channel with a cavity at a wall (C-shaped cavity). Results were obtained for two Reynolds numbers and two Richardson numbers. Some different behaviors were observed between the two geometrical configurations in terms of flow pattern in the open cavity and wall temperature profiles.

Saha et al. [8] studied the performance of mixed convection in a rectangular enclosure. Four different placement configurations of the inlet and outlet openings were considered. Results were obtained for a range of Richardson numbers from 0 to 10 at $\Pr = 0.71$ and $\Re = 100$, with constant physical properties. The results indicate that the average Nusselt number and the dimensionless surface temperature on the heat source strongly depend on the position of the inlet and outlet. Most of the previous works have been devoted to the studies of mixed convection in the enclosure without body. There are only a few papers on numerical studies of mixed convection in the heated body inserted horizontally in the middle of the cavity. Rahman et al. [9] numerically investigated 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. 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. Rahman et al. [10, 11] also studied mixed convection in a vented square cavity with a heat conducting square or circular cylinder at different locations. They found that the flow field and temperature distributions inside the cavity are strongly dependent on the Richardson, Reynolds, and Prandtl numbers, thermal conductivity of the cylinder, and the position of the inner cylinder.

Shih et al. [12] studied the periodic laminar flow and heat transfer due to insulated or various isothermal rotating objects (circle, square, and equilateral triangle) placed in the center of the square cavity. Transient variations of the average Nusselt number of the respective systems show that high values of Reynolds number produce a quasi-periodic behavior, while for low values of Reynolds number periodicity of the system is clearly observed. Oztop et al. [13] numerically studied mixed convective flow in a lid-driven air flow within a square enclosure having a circular body. Flows are driven by the left lid, which slides in its own plane at constant velocity. This wall is isothermal and it moves up or down in $y$-direction while the
other walls remain stationary. The horizontal walls are adiabatic. The cavity is differentially heated and the left wall is maintained at a higher temperature than the right wall. Three different temperature boundary conditions were applied to the inner cylinder as adiabatic, isothermal, or conductive. They found that the flow field and temperature distribution are affected by the direction of the moving wall. Higher heat transfer is observed for the case of the downward moving wall. Multiple circulation cells are observed for the downward moving lid in some cases. Also, their model is not sensitive to thermal conductivity for low values of diameter of circular body, but the mean Nusselt number increases with the increasing of the diameter of the inserted body. However, there are still various applications of mixed convection in enclosures due to multiple discrete heat sources [14]. Wang and Jaluria [15] numerically studied three-dimensional mixed convection flow in a horizontal rectangular duct with multiple discrete heat sources flush-mounted on the bottom surface, and the main attention was focused on the characteristics of the instability and the resulting effect on the heat transfer.

Nelson and Sergio [16] studied numerically mixed convection in a two- and three-dimensional cavity. Calculations were preformed for air with Pr = 0.71, flowing with a Reynolds number in the range $0 \leq Re \leq 500$ and a Richardson number in the range $0 \leq Ri \leq 18$. They observed that the comparison between 3-D and 2-D simulations showed that a 3-D model was needed to capture the fluid mechanics for $Ri = 10$ when $10 \leq Re \leq 250$, and to calculate the global Nusselt number when $Re = 500$ for $Ri < 1$. Singh and Sharif [17] studied mixed convection in an air-cooled cavity with differentially heated vertical isothermal side walls having six placement configurations of the inlet and outlet ports. They observed that the maximum cooling effectiveness was achieved if the inlet was kept near the bottom of the cold wall, while the outlet was placed near the top of the hot wall. Laminar mixed convection in a two-dimensional enclosure heated from one side wall and submitted to an either aiding or opposing jet was numerically studied in the work of Raji and Hasnaoui [18].

Very recently, Shahi et al. [19] used a finite volume approach to solve the mixed convection flow in a square cavity ventilated and partially heated from below utilizing nanofluid. The thermal conductivity and effective viscosity of nanofluid have been calculated by Patel and Brinkman models, respectively. The results indicate that increase in solid concentration leads to an increase in the average Nusselt number at the heat source surface and a decrease in the average bulk temperature. Costa and Raimundo [20] studied the problem of mixed convection in a square enclosure with a rotating cylinder centered within. Results clearly show how the rotating cylinder affects the thermal performance of the enclosure, and how the thermophysical properties of the cylinder are important in the overall heat transfer process across the enclosure. They have also concluded that for small rotating velocities, the highest Nusselt numbers are obtained for the smallest values of the thermal conductivity and thermal capacity of the cylinder.

As shown above, most of the previous studies of the vented cavity had a common feature that the cavity was either top-vented or horizontally placed with the heat source placed at its bottom wall. Also, most of the studies that consider a cavity or enclosure were vented by being fully open or partially open at the center of the side walls or shifted from their center. But none of these studies covered the case where the enclosure was used in three different locations of the exit port,
within which a square solid cylinder with size $d$ has radius 0.01 L in each corner. Finally, the objective of the present study is based on the configuration of Rahman et al. [10] for mixed convection in a straight square cavity with an inner conductive square cylinder. The present work deals with the same problem but the horizontal walls and the right vertical wall are kept adiabatic, while the left vertical wall is maintained at a constant temperature $T_c$. In addition, a solid square cylinder inserted in the square cavity has radius 0.01 L in each corner which is kept at isothermal hot temperature instead of conductive straight corner square cylinder.

2. PHYSICAL MODEL AND COORDINATE SYSTEM

The physical model considered here is shown in Figure 1 along with the important geometric parameters. A Cartesian coordinate is used with the origin at the lower left corner of the computational domain. It consists of square cavities with sides of length $L$, within which a square solid cylinder with size $d$ has radius 0.01 L in each corner, which is kept at isothermal hot temperature $T_h$ while the left

![Figure 1](image)

*Figure 1. Three schematic configurations of a thermally driven cavity with coordinate system and boundary conditions.*
wall of the square cavity is subjected to cold temperature \(T_c\) and other walls are kept adiabatic. The inflow opening located on the left cold vertical wall is arranged and fixed at \(h_i = 0.45\,L\), whereas the outflow opening is on the opposite adiabatic vertical wall which is arranged as shown in the schematic figure and may vary in location, placed either at the top, mid wall, or bottom position. The cavity has dimensions of \(L \times L\) and thus the aspect ratio is fixed to 1. It is assumed that the incoming flow is at a uniform velocity \(u_i\) and at the ambient temperature \(\theta_i\). Since the boundary conditions at the exit of the cavities are unknown, values of \(u, v, P,\) and \(\theta\) are extrapolated at each iteration step.

The cavity presented in Figure 1a is subjected to an external flow entering into the cavity from the left cold vertical wall at \(h_i = 0.45\,L\), and leaves it from the top of the opposite vertical one. For brevity, this configuration will be referred to as CT from now on. Similarly, the other two configurations will be referred to as CB (Figure 1b) and CC (Figure 1c), respectively. For simplicity, the size of the two openings, \(w\) is set equal to the one-tenth of the cavity length \(L\). All solid boundaries are assumed to be rigid no-slip walls. The effect of various governing dimensionless parameters and the size of the square cylinder on the heat transport process are studied in the present work.

It is necessary to examine the range of the governing parameters \(Re\) and \(Ri\) chosen in the present study. In order to maintain laminar flow in the cavity, the values of \(Re\) and \(Ri\) are considered to be in the ranges 50 to 200 and 0 to 10, respectively, and Prandtl number \(Pr\) is taken as 0.71. The fluid properties are also assumed to be constant, except for the density in the buoyancy term which follows the Boussinesq approximation. The radiation effects are neglected and the gravitational acceleration acts in the negative \(y\)-direction. The fluid within the cavity is assumed Newtonian while viscous dissipation effects are considered negligible. The viscous incompressible flow and the temperature distribution inside the cavities with the horizontal square cylinder are described by the Navier–Stokes and the energy equations, respectively.

### 3. MATHEMATICAL ANALYSIS

#### 3.1. Governing Equations

Taking into account the abovementioned assumptions, the governing equations for this investigation can be written in dimensional form as follows.

**Mass conservation**

\[
\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \tag{1}
\]

**Momentum equations**

\[
u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = -\frac{1}{\rho} \frac{\partial P}{\partial x} + \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \tag{2}
\]

\[
\nu \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} = -\frac{1}{\rho} \frac{\partial P}{\partial y} + \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + g\rho\beta(T - T_c) \tag{3}
\]
Energy equation

\[ u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} = \alpha \left( \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} \right) \]  

(4)

The dimensionless variables are defined as follows [21].

\[ \theta = \frac{T - T_c}{T_h - T_c}, \quad X = \frac{x}{L}, \quad Y = \frac{y}{L}, \quad U = \frac{u}{u_i}, \quad V = \frac{v}{u_i}, \quad L_x = \frac{L_x}{L}, \quad L_y = \frac{L_y}{L}, \]

\[ AR = \frac{d}{L}, \quad \Omega = \frac{\omega}{u_i L}, \quad \Psi = \frac{\psi L}{u_i}, \quad P = \frac{p}{\rho u_i^2}, \quad Pr = \frac{v}{\alpha}, \quad Re = \frac{\rho u_i L}{\mu}, \]

\[ Gr = \frac{g \beta (T_h - T_c)^3}{\beta^2}, \quad Ri = \frac{Gr}{Re^2} \]  

(5)

where \( X \) and \( Y \) are the dimensionless coordinates measured along the horizontal and vertical axes, respectively, \( u \) and \( v \) are the dimensional velocity components along \( x \) and \( y \) axes, \( \theta \) is the dimensionless temperature, \( \beta \) is the volumetric coefficient of thermal expansion, and \( g \) is the gravitational acceleration. The dimensionless forms of the governing equations under steady state condition are expressed in the following forms.

\[ \nabla^2 \psi = -\Omega \]  

(6)

\[ \frac{\partial \Omega \partial \Psi}{\partial X \partial Y} - \frac{\partial \Psi \partial \Omega}{\partial X \partial Y} = \frac{1}{Re} \nabla^2 \Omega + Ri \frac{\partial \theta}{\partial X} \]  

(7)

\[ \frac{\partial \theta \partial \Psi}{\partial X \partial Y} - \frac{\partial \Psi \partial \theta}{\partial X \partial Y} = \frac{1}{Re Pr} \nabla^2 \theta \]  

(8)

The horizontal and vertical velocities (\( U \) and \( V \)) can be obtained from

\[ U = \frac{\partial \Psi}{\partial Y} \quad \text{and} \quad V = -\frac{\partial \Psi}{\partial X} \]  

(9)

with

\[ \Omega = \frac{\partial Y}{\partial X} - \frac{\partial U}{\partial Y} \]  

(10)

### 3.2. Boundary Conditions

The nondimensional form of the boundary conditions for the present problem is specified as follows.

- At the inlet: \( U = 1, \ V = 0, \ \theta = 0 \)
- At the outlet: convective boundary condition (CBC) [8]
3.3. Heat Transfer Calculations

The rate of heat transfer is computed at each wall and is expressed in terms of the local surface Nusselt number \( (\text{Nu}) \) and surface-averaged Nusselt number \( \overline{\text{Nu}} \) \([22]\) as

\[
\text{Nu} = \left. \frac{\partial \theta}{\partial n} \right|_s, \quad \overline{\text{Nu}} = \sum \frac{1}{S} \int_0^S \text{Nu} dn
\]

where \( n \) is the normal direction with respect to the surface, The Simpson’s 1/3 rule is used for numerical integration to obtain the average Nusselt number. The bulk average temperature is defined as

\[
\theta_{av} = \int \frac{0d\overline{V}}{\overline{V}}
\]

where \( \overline{V} \) is the cavity volume which should be minimized.

4. NUMERICAL SOLUTION PROCEDURE AND VERIFICATION

A finite difference method based on successive over-relaxation iterative method is used to solve numerically the nondimensional governing Eqs. (6)–(8). A computational program was written in Fortran 90 language to compute the values of the required variables. The region of interest was covered with vertical and horizontal nonuniformly spaced grid lines is equal to \( N \). The vorticity and temperature distributions are obtained from Eqs. (7) and (8), respectively. The stream distribution was obtained from Eq. (6) using successive over-relaxation (SOR, which is a variable between 0 and 1) and a known vorticity distribution. An iterative process is employed to find the stream function, vorticity, and temperature fields. The process is repeated until the following convergence criterion is satisfied.

\[
\frac{\Phi_{\text{New}} - \Phi_{\text{Old}}}{\Phi_{\text{New}}} \leq 10^{-6}
\]

Two dimensions body fitting grid are used for the present computation. The 2-D computational grids are clustered toward the walls. The location of the nodes is calculated using a stretching function as described by Thompson et al. [25], so that the node density is higher near the walls and the round corners of the vented square cavity and inner square cylinder. In order to obtain the grid independent solution, a grid refinement study is performed for a square cavity with the horizontal square cylinder located at the centerline of the square cavity \( (L_X = 0.5, \ L_Y = 0.5) \) for CT configuration with \( \text{Re} = 5, \ \overline{\text{Re}} = 200, \ \text{Pr} = 0.71, \ \text{and AR} = 0.2 \). In the present work, eight combinations \( (40 \times 40, \ 50 \times 50, \ 60 \times 60, \ 70 \times 70, \ 80 \times 80, \ 100 \times 100, \text{and} \ 120 \times 120) \) were considered.
120 × 120, and 150 × 150) of nonuniform grids are used to test the effect of grid size on the accuracy of the predicted results. Figure 2 shows the convergence of the average Nusselt number $(\overline{Nu})$, at the heated surface of the square cylinder with grid refinement. It is observed that grid independence is achieved with combination of (120 × 120) control volumes where there is insignificant change in the average Nusselt number $(\overline{Nu})$ with the improvement of the finer grid. The agreement is found to be excellent, which validates the present computations indirectly.

To the best of the author’s knowledge, there are no experimental results reported in the literature for this configuration. In order to make sure that the developed codes are free of error, a verification test is conducted. The present numerical approach was verified against the results published by Rahman et al. [10] for mixed convection in a square cavity of dimensions $L \times L$, with a heat conduction horizontal square cylinder. 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 ($w$ is set equal to one-tenth of the cavity length $L$) is placed at the top of the opposite heated wall. Figure 3 shows the streamlines and isotherms calculated by Rahman et al. [10] and the present study for BT configuration at $Re = 100$, $Ri = 5$, $AR = 0.2$, $L_x = 0.5$, $L_Y = 0.25$, and thermal conductivity ratio of the fluid ($Pr = 0.71$) and solid ($K = 5$) using the same boundary conditions but the numerical scheme is different. Excellent agreement is achieved between Rahman et al. [10] and the present numerical scheme for both the streamlines and temperature contours inside the square cavity within a square cylinder, as shown in Figure 3. This verification gives confidence in the present numerical model to deal with the present physical problem.
5. RESULTS AND DISCUSSION

In general, in any forced convection heat transfer situation buoyancy effects are always present and interact with the forced flow effects. If the buoyancy effect is smaller, forced convection tends to predominate and if the buoyancy is larger the heat transfer is dominated by natural convection. When there is a significant interaction between forced and free convection effects, the heat transfer mechanism is called mixed convection. In this case, both the Reynolds and the Richardson numbers become important parameters. On the other hand, the cold fluid has a tendency to flow downwards due to buoyancy while the hot fluid rises up.

5.1. Effects of Re and Ri for Different Exit Configurations

5.1.1. Considering CT configuration. Figure 4 shows the streamline and isotherm contours for four different Reynolds and Richardson numbers at AR = 0.2. For Ri = 0, the streamlines and the isotherms contours at Re = 50, 100, 150, and 200 are shown in Figure 4 (first row). For low Ri values, the buoyancy effects are weak. For smaller value of Reynolds number (Re = 50), there exist two
Figure 4. Variation of streamlines and isotherms for different Re and Ri for CT configuration at AR = 0.2.
vortex cells of the same sizes at the top and bottom of the inlet port. As the value of Re increases, the sizes of the two circulating cells gradually increase. The corresponding temperature distributions can be seen in Figure 4 (first row). It can be seen that increases in the values of the Reynolds number reduce the thermal boundary layer thickness near the opposing surface of the inner square cylinder, since at a larger value of Reynolds number the effect of the gravitation force becomes negligible. At Ri = 2.5 (Figure 4, second row), increasing the value of Ri gradually causes the development of the vortex cell located at the left top corner of the cavity for a small value of Reynolds number (Re = 50), and leads to a large change in the streamlines’ structure. Convection heat transfer causes the growth of the vortex cell resulting in faster removal of heat from the heated wall. As Re increases to 100, the vortex cell spreads and changes its pattern from a unicellular vortex to a bicellular vortices. When a further increase of the Reynolds number to 150 occurs, the patterns of the streamlines are predicted to be the same as those for Re = 100, but a careful observation of the results indicates that the two inner vortices become bigger in size because of the effect of natural convection. For the high values of the Reynolds number (Re = 200), the streamlines for this case appear to be identical to those corresponding to Re = 150, in spite of increases in the strength of the recirculation vortex. When the Richardson number increases to 5 (Figure 4, third row), the circulating vortex increases in size for a small value of Reynolds number (Re = 50) because of the strong buoyancy force. For Re = 100, the vortex grows in size and a small vortex cell develops in the bottom of the inlet port. As Re increases to 150 and 200, the vortex spreads and changes its pattern from a single vortex trend into a two vortices trend and another small vortex is formed at the right side wall of the cavity. This is because of the mixing of the fluid due to buoyancy force and convection currents in the cavity. As the Richardson number increases to 10 (Figure 4, fourth row), the buoyancy force becomes the dominant mechanism to drive the convection of the fluid and the flow is in the region of natural convection. The patterns of the streamlines and isotherms are the same as those for Ri = 5. For small Richardson numbers (Ri = 0), the vortex cell allows heat removal at the exit port of the cavity, whereas in the other cavity zone heat removal is merely diffusive because the fluid is nearly motionless in the cavity. As the Richardson number increases (Ri = 2.5 to 10), the role of convection in heat transfer becomes more significant and consequently, the thermal boundary layer on the surface of the inner cylinder becomes thinner. Also, a plume starts to appear at the top of the inner cylinder and as a result the isotherms move upward, giving rise to a stronger thermal gradient in the upper part of the cavity and a much lower thermal gradient in the lower part. Consequently, the dominant flow is in the upper half of the cavity.

5.1.2. Considering CC configuration. Figure 5 shows the streamline and isotherm contours for four different Reynolds and Richardson numbers at AR = 0.2. For Ri = 0, two secondary circulating cells of same sizes are formed at the top and bottom of the inlet port for all Reynolds number values, as seen in Figure 5 (first row). However, for Ri = 2.5 and Re = 50, one circulating cell is formed on top of the main flow. The main flow always moves below the heated cylinder towards the right side opening. The formation of the circulating cell is because of the mixing of the fluid due to buoyancy-driven and convective currents. For
Figure 5. Variation of streamlines and isotherms for different Re and Ri for CC configuration at AR = 0.2.
Re = 100, the vortex spreads and changes its pattern from the single vortex trend into the two vortices trend. This is because of the mixing of the fluid due to buoyancy force and convection currents in the cavity. For Re = 150, the vortices grow more in size especially at the bottom side near the inlet port. With further increase in the value of Re (Re = 200), the patterns of the streamlines are the same as those for Re = 150. As Ri increases from 2.5 to 5 for Re = 50, the values of the streamlines increase, i.e., the fluid flow intensity increases and the buoyancy effects accelerate the fluid further. The heated cylinder results in the recirculation near the upper corner of the cavity. When a further increase of Re from 100 to 200 takes place, all patterns of the streamlines are predicted to be the same as those for Ri = 2.5. When Ri increases (Ri = 10 and Re = 50), the buoyancy force becomes the dominant mechanism to drive the convection of the fluid and the flow is in the region of natural convection. A big vortex cell is produced near the left side of the heated cylinder and another one grows in the upper corner of the cavity moving in the direction of the exit port. At Re = 100, the convective cell in the upper corner of the cavity disappears due to a high stream of the flow. When a further increase of Re from 150 to 200 occurs, all patterns of the streamlines are predicted to be the same as those for Ri = 2.5, 5, and 200. The thermal field is governed more or less by interaction between the incoming cold fluid stream and the circulating vortex, which is created near the left side wall of the heated square cylinder. For Ri = 0, the temperature region is more concentrated between the inlet port and the left hot side wall of the inner cylinder, especially near the hot cylinder wall and the isothermal lines become more crowded and their shape changes significantly to nonuniform, almost linear and symmetrical about the x-axis as the value of Re increases as shown in Figure 5 (fifth row). The same figure also presents the temperature contours plotted for Ri = 2.5, 5 and 10, respectively. One can observe the rapid changes occurring conspicuously in the temperature contours and their shape changes significantly to nonsymmetrical about the y-axis, nonuniform horizontal, and nonlinear vertical as the value of Re increases, which indicates the effect of the increase in Richardson number on the heat transfer process.

5.1.3. Considering CB configuration. Figure 6 shows the streamline and isotherm contours for four different Reynolds and Richardson numbers at AR = 0.2. For low Ri (Ri = 0), the buoyancy effects are weak. When a smaller value of the Reynolds number (Re = 50) is used, two vortex cells of the same sizes appear at the top and bottom of the inlet port. With the increasing value of Re, the size of the two circulating cells gradually increases. For Ri = 2.5 and Re = 50, the cold fluid has a tendency to flow downwards towards the exit port due to the buoyancy force. For Re = 100, two small recirculatory vortices are observed in the upper right corner due to warm air rising along the right and in the bottom left corner due to available space for the shared force induced, through stream at the inlet. With the increasing value of Re up to 200, the intensity of recirculation continues to increase and its size continues to increase with increases in value of Re. For higher Richardson number (5 and 10), large recirculation zones are formed above the main fluid stream. The cold fluid stops down the inner cylinder in the middle of the cavity and flows along the floor towards exit port. For Re = 50 and 100, the main vortex increases in size and a small vortex cell is developed in the upper right corner due to warm air rising along the right wall in the cavity. A clear similarity of the flow patterns is found
Figure 6. Variation of streamlines and isotherms for different Re and Ri for CB configuration at AR = 0.2.
between the streamlines for $\text{Ri} = 5$ and the streamlines for $\text{Ri} = 2.5$, with the other values of Reynolds number ($\text{Re} = 150$ and $200$). On the other hand, large temperature gradients close to the hot wall of the inner square cylinder and stratified temperature distribution in the rest of the cavity are observed for $\text{Ri} = 2.5$ and $5$, while similar behavior with the CT configuration can be observed for the isotherm contours for $\text{Ri} = 0$. With increased dominance of natural convection at $\text{Ri} = 10$, the incoming cold air and the hot vortex start to mix up and carry the heat to the bulk of the cavity. The high temperature zone grows in size, especially in the upper region of the cavity.

5.2. Effect of Inner Cylinder Locations on Flow and Thermal Fields

Figure 7 shows the variation of the streamline and isotherm contours for different square cylinder locations at $\text{AR} = 0.2$, $\text{Re} = 200$, and $\text{Ri} = 1$. When the inner cylinder is located at the center, two vortex cells are observed on the upper and lower sides of the inlet port location in the cavity, as shown in Figure 7a. When the inner cylinder is located at $L_X = 0.5$, $L_Y = 0.75$ as shown in Figure 7b, the big vortex cell is formed below the main flow due to shear force and a vortex cell produced due to the buoyant force is seen in the upper left corner. When the inner cylinder is located at $L_X = 0.25$, $L_Y = 0.75$ as shown in Figure 7c, the big vortex cell grows while the other one decreases in size near the upper side of the inlet port. Finally, when the inner cylinder is located at $L_X = 0.75$, $L_Y = 0.75$ as shown in Figure 7d, the big vortex is noticed in the bottom half of the cavity and another one is observed due to warm air rising at the left top corner. The isotherms are clustered around the inner square cylinder when the cylinder is located at $L_X = 0.75$, $L_Y = 0.75$. This can be attributed to the higher forced convection currents.

5.3. Effect of Aspect Ratio on Flow and Thermal Fields

Figure 8 shows the streamline and isotherm contours for various values of the aspect ratio of the inner heated square cylinder located at the center of the cavity for $\text{Ri} = 1$ and $\text{Ri} = 200$. For an aspect ratio of $0.1$ (Figure 8a), it can be seen that two vortex cells of different sizes have developed at the upper and lower sides of the inlet port location in the cavity. For an aspect ratio of $0.2$, (Figure 8b), only small differences in the streamlines are observed when compared with Figure 8a. In Figure 8c and 8d, increasing the aspect ratio from $0.3$ to $0.4$ is seen to reduce the space available for the buoyancy-induced recirculating flow. From the isotherm contours, it is observed that the isothermal lines are clustered around the heated cylinder for all cases. Some deviation of the isothermal lines is observed on the upper side of the heated cylinder. This is due to more heat being carried away from the heated cylinder and dissipated through the out flow opening.

5.4. Effect of Outlet Positions on Heat Transfer Rate

Figure 9a shows the variation of the outlet position on the average Nusselt number along the heated surface of the inner square. When $\text{Ri}$ increases from $0$ to $2.5$, the average Nusselt number reduces very sharply for the cases of CC and CB
configurations, while it increases linearly for the CT configuration case. This behavior results from higher cooling effects of the forced currents in the CT configuration than other configurations. As $\text{Ri}$ increases from 2.5 to 10, the average Nusselt number increases gradually due to the increasing effect of convection. Figure 9b shows the variation of the outlet position on the average Nusselt number along the left cold wall of the outer square cavity. The average Nusselt number is predicted to increase rapidly with increasing values of the Richardson numbers for all outlet positions, which indicates the increasing effect of mixed convection. On the other hand, the
average temperature ($\theta_{av}$) for the cases of CC and CB configurations increases as $Ri$ increases. Conversely, the average temperature ($\theta_{av}$) decreases as $Ri$ increases when the outlet positions are at the top (CT configuration), as shown in Figure 9c.

### 5.5. Effect of Reynolds Number on Heat Transfer Rate

Figure 10a shows the average Nusselt number along the heated surface of the inner square as a function of the Richardson number for different Reynolds numbers. The heat transfer from the surface of a heated square cylinder to the
surrounding fluids can be either pure forced convection, free convection, or mixed convection depending on the Richardson number. When Ri increases from 0 to 2.5, the average Nusselt number increases moderately which indicates the onset of the buoyancy force dominated region. For increments of Ri from 2.5 to 10, the average Nusselt number increases more because the buoyancy force becomes the dominant mechanism to drive the convection of the fluid. On the other hand, the average Nusselt number increases with increasing Reynolds number. This is because more heat is carried away from the inner square cylinder and dissipated through the out flow opening for large values of Reynolds number.

Figure 10b depicts the average temperature ($\theta_{av}$) as a function of the Richardson number for various values of the Reynolds number. The average temperature ($\theta_{av}$) decreases clearly with the increase in both Ri and Re. A higher average temperature is obtained for Re = 50 indicating poor heat transfer from the inner square cylinder. But, the increase in Re (increase in the forced convection effect) enhances
the heat removal through the exit, i.e., increases the air flow rate inside the cavity. When $Re = 200$, a minimum average temperature ($\theta_{av}$) is observed among the others and optimum heat transfer performance is predicted.

### 5.6. Heat Transfer Correlation

The simulated data obtained for all governing parameters are used for the development of an empirical correlation for general convection. The complex dependence of $Nu_{ave}$ on $Re$ and $Ri$ apparently rules out obtaining a single equation that could correlate all the results. The average Nusselt numbers shown in Figure 10 are correlated in terms of Richardson number $Ri$ and Reynolds number $Re$ for CT configuration as one of the best configurations in terms of higher heat transfer rate. A least-squares method is used to correlate the data in Figure 10 for each variation of $Re$. The obtained correlation is expressed as follows.

$$Nu_{ave} = 0.121Ri^{1.149} + 0.519Re^{0.5321} - 1.159$$  \[14\]

where $50 \leq Re \leq 200$, $0 \leq Ri \leq 10$, and the maximum correlation coefficient is 0.976.

### 6. CONCLUSION

Numerical results are reported for the flow and temperature distributions fields in a heated square solid cylinder enclosed in a square air-filled cavity with various geometry configurations. The results are obtained for wide ranges of important parameters such as the Richardson number ($Ri$), Reynolds number ($Re$), location ($L_X$, $L_Y$) and the aspect ratio (AR). In view of the obtained results, the following findings have been summarized.
1. The effects of the location of the inner square cylinder and the aspect ratio are found to play significant roles in the streamline and isotherm contour patterns.

2. For low Richardson number (0 to 2.5), the results show that the average Nusselt number along the heated cylinder decreases for the CB and CC configurations. However, as the Richardson number increases from (2.5 to 10), the results show that the average Nusselt number increases for the same configurations. In addition, the present results show that the average Nusselt number for the CT configuration increases almost linearly with increases in the Richardson number.

3. The average Nusselt number along the left cold wall of the square cavity increases almost linearly with the Richardson number for all configurations.

4. The average surface temperature increases as the Richardson number increases for the CC and CB configurations, while the average surface temperature decreases with increasing values of the Richardson number for the CT configuration. This is attributed to the free convection current that pushes hot fluid to the upper part of the cross-section and cold fluid to the lower part.

5. The average Nusselt number along the heated surface of the inner square cylinder increases as the Reynolds and the Richardson numbers increase while the average surface temperature decreases with increasing values of both the Reynolds and Richardson numbers for all considered configurations.

6. The secondary flow effects created by free convection tend to decrease the heat transfer results at low Re and to increase the heat transfer results for high Re.

7. The CT configuration is found to produce more heat transfer enhancement than the CC and CB configurations.

8. The average bulk fluid temperature seems to remain constant or is slightly affected in a highly buoyancy-dominated convection regime.

A correlation among $\text{Nu}_{\text{ave}}$, $\text{Re}$, and $\text{Ri}$ is derived to evaluate the effect of $\text{Re}$ and $\text{Ri}$ on $\text{Nu}_{\text{ave}}$. It is found that the average Nusselt number from the source is only slightly affected by the Reynolds number while it varies with the Richardson number according to $\text{Ri}^{0.25}$.

REFERENCES


