Effect of Several Heated Interior Bodies on Turbulent Natural Convection in Enclosures

A. Nouri-Borujerdi*, F. Sepahi
School of Mechanical Engineering, Sharif University of Technology, Azadi Avenue, Tehran, Iran

Abstract

In this study, turbulent natural convection in a square enclosure including one or four hot and cold bodies is numerically investigated in the range of Rayleigh numbers of $10^{10} < Ra < 10^{12}$. The shape of the internal bodies is square or rectangular with the same surface areas and different aspect ratios. In all cases, the horizontal walls of the enclosure are adiabatic and the vertical ones are isothermal. It is desired to investigate the influence of different shapes and arrangements of internal bodies on the heat transfer rate inside the enclosure with wide-ranging applications such as ventilation of buildings, electronic cooling and industrial coldbox packages. Governing equations including Reynolds-averaged-Navier-Stokes equations have been solved numerically with finite volume method and $k-\varepsilon$ turbulence model in a staggered grid. The boundary condition for turbulence model is based on the standard wall function approach. Strongly implicit method is employed to solve the discretized systems of algebraic equations with a remarkable rate of convergence. The effects of several parameters such as distance between the bodies, aspect ratio and Rayleigh number on heat transfer rate have been investigated. The most change in heat transfer rate at high values of Rayleigh numbers is associated with alteration in distance between square bodies. Moreover, the horizontal installation of rectangular bodies with $h/w = 1/3$ is accompanied by a maximum reduction of heat transfer at low Rayleigh numbers. The present results have been compared with previous experimental and numerical works regarding enclosures with or without internal bodies and reasonable agreement is observed.

Keywords: Natural convection, enclosure, interior bodies; turbulent flow, numerical method

1. Introduction

Natural convection is one of the most prevalent phenomena in our daily life and industry with many engineering applications such as ventilation of buildings with radiators, double-glazed windows, solar collectors and cooling of electronic equipment. Accordingly, it has widely been considered in many experimental and numerical researches in past decades most of which involved natural convection within enclosures abundantly demanded by industrial instruments.

Several experimental works involving natural convection inside enclosures have been done in the past decades which provide useful data regarding thermal and flow fields and turbulence quantities for validation of numerical simulations. Most of these works have dealt with tall cavities at high Rayleigh numbers such as enclosures with aspect ratios of 5 in Bowler and Cheezewright [1], 3.84 in Saury et al. [2] and 28.6 in Dafa’Alla et al. [3] and Betts and Bokhari [4] investigations. A cubic water filled cavity at high Rayleigh numbers first was tested by Kirkpatrick and Bohn [5]. Later, low turbulent natural convection in an air-filled square enclosure at $Ra = 1.5 \times 10^9$ was experimented in works of Tian and Karayiannis [6,7]. Further studies for the same cavity regarding mean and fluctuation quantities performed by Ampofo and Karayiannis [8]. Salat et al. [9] also experimented an analogous cavity with the same Rayleigh number to validate their numerical results. Laminar natural convection in a set of air filled cavities with Rayleigh numbers up to $Ra = 10^6$ first was numerically studied

* Corresponding author. Tel.: +98 21 66165547; fax: +98 21 66000021
Email addresses: anouri@sharif.edu (A. Nouri-Borujerdi)
by De Vahl Davis [10] and then improved by Hortmann [11] by using much finer grids. Le Quere [12] utilized different accurate methods in cavities with Rayleigh numbers up to $Ra = 10^8$. However, at large enclosures which are of interest to present study the Rayleigh number exceeds the critical value approximately equal to $10^9$ which leads to the flow regime to be time dependent and turbulent. Hence utilizing a proper model for turbulence is of great importance and has extensively been investigated in previous researches. Phillips [13] used $k-\varepsilon$ turbulence model for simulation of an air filled cavity up to $Ra = 10^{14}$ with wall functions established by Launder and Spalding [14]. In tall cavities simulated by Inc and Launder [15] with aspect ratios of 30:1 and 5:1, it is found that the version of Jones and Launder [16] low-Reynolds-number $k-\varepsilon$ model produces accurate results in satisfactory agreement with reported experimental data. Henkes et al. [17] conducted a study on turbulent natural convection inside a cavity with different versions of low-Reynolds-number $k-\varepsilon$ model and also standard $k-\varepsilon$ model with wall functions defined for $k$ and $\varepsilon$ variables. The Nusselt numbers obtained by low-Reynolds-number $k-\varepsilon$ models is found to be in a better agreement with experimental data, however the standard $k-\varepsilon$ model predicts higher corresponding Nusselt numbers. Barakos and Mitsoulis [18] studied numerically laminar and turbulent natural convection in an air filled cavity at various Rayleigh numbers up to $Ra = 10^{10}$. The results indicate that even though standard $k-\varepsilon$ model with logarithmic wall functions employed for velocity and temperature over-predicts averaged Nusselt number of the hot wall, it performs reasonably when wall functions are defined only for $k$ and $\varepsilon$ at first computational grid points after the walls. In five different $k-\varepsilon$ models examined by Chen [19], low-Reynolds-number model shows better performance in the cavity compared to the standard and RNG $k-\varepsilon$ models, but the standard model displays an acceptable precision close to experimental data by using much coarser grid and lower computational cost. Other studies regarding turbulent natural convection inside enclosures with various methods and wall functions have been presented in works of Trias et al. [20,21], Hsieh and Lien [22], Hanjalic and Vasic [23], Dol et al. [24] and new wall functions applicable in turbulent natural convection have been developed by Craft et al. [25]. Also, a comprehensive review performed by Baiţi et al. [26] in which a broad range of experimental, analytical and numerical studies over natural convection in enclosures with various geometries, thermal boundary conditions and working fluids have been considered.

Many of the recent researchers have taken the importance of this problem into account and conducted numerical simulation of natural convection inside these enclosures with different geometries and boundary conditions of interior obstacles. Ho et al. [27,28] numerically and experimentally studied laminar natural convection of two heated cylinders confined in a circular insulated enclosure or subjected to external convection at $10^4 < Ra < 10^7$. Ha and Jung [29,30] considered the effects of Prandtl number on transient natural convection inside a cavity with a centered square body under different thermal conditions at $10^3 < Ra < 10^6$. Laminar natural convection at $10^3 < Ra < 10^6$ in square cavities including a heated plate built-in horizontally and vertically with different aspect ratios or an isothermal partition attached to the bottom wall were investigated by Oztop et al. [31,32]. Numerical studies regarding square enclosures containing two mutually orthogonal heated baffles at different boundary conditions were carried out in Kandaswamy et al [33] and Hakeem et al. [34] to evaluate the effects of baffles’ length and position on heat transfer rate. Later, many researchers studied natural convection mechanism of heat
transfer in square enclosures including one or two inner heated cylinders positioned at different vertical and horizontal locations in the range of Rayleigh numbers of $10^3 \leq Ra \leq 10^6$ [35–37] and Park et al. [38]. Garoosi et al. [39] and Garoosi and Hoseininejad [40] investigated the effects of position, size, aspect ratio and orientation on natural and mixed convection heat transfer between heated circular or square cylinders in an enclosure filled with nanofluids at $10^4 \leq Ra \leq 10^7$.

In above mentioned studies, various numerical methods were involved to model the turbulence within empty enclosures at different boundary conditions and values of Rayleigh number. Then a number of important works about natural convection inside enclosures including some kind of obstacles were reviewed and all of which were conducted at low values of Rayleigh number. Hence, the need for some numerical or experimental data about turbulent natural convection inside large enclosures with heated internal bodies at high Rayleigh numbers is impressive here and to the best of authors’ knowledge it has not ever been investigated in the literature. Accordingly, the main purpose of this work is to study turbulent buoyancy driven convection inside large enclosures containing several hot and cold bodies with applications in chemical complex industry where Coldbox packages with interior brazed heat exchangers are used. Natural convection takes place at high Rayleigh numbers inside large enclosures of these packages and the flow field is partly or fully turbulent depending on how large the Rayleigh number is. Therefore, it is desired to find the best position and orientation of heated and cooled bodies as representatives for interior heat exchangers for having access to an optimum heat transfer rate between them.

2. Physical Problem

Fig. (1a, b) shows the schematic of two square enclosures, one of the them includes a single square hot body and the other one consists of two hot and two cold bodies with their corresponding boundary conditions. The vertical and horizontal walls of the enclosure are isothermal and adiabatic respectively. The hot bodies are colored by red which stands for $\theta = 1$ and the cold bodies are colored by blue showing the dimensionless temperature of $\theta = 0$. The temperature of the isothermal walls of the enclosure is $\theta = 0$ regarding the case with one internal body and $\theta = 0$ for the enclosure with four bodies inside. The default height and width of each interior body are assumed $w/H = h/H = 0.25$ respectively. Moreover, when the effect of aspect ratio is due to be investigated the height and width of the body is changed to 0.375 and 0.125 or vice versa for aspect ratios of $h/w = 3$ and $1/3$. The surface areas of bodies are always kept constant and equal to 1. Assume that we are dealing with the viscous flow in the enclosure filled with air ($Pr = 0.71$) and with constant physical properties except for density in gravitational terms which varies linearly with temperature known by Boussinesq approximation.

3. Mathematical Formulation of the Governing Equations

Reynolds-Averaged-Navier-Stokes equations are adopted here to solve turbulent natural convection in an enclosure. In order to model the turbulence we have overlooked Low-Reynolds-Number $k-\varepsilon$ models because of two challenges including their non-uniqueness solution depending on the initial condition and also the need for an extremely dense grid near the walls according to Henkes et al. [17]. The second deficiency will appear more
challenging when enclosures containing internal bodies are to be studied and the need for an adequately concentrated grid increases the cost of computation dramatically. In addition, as it was mentioned earlier, the standard version of $k-\varepsilon$ model with logarithmic wall functions defined on velocity and temperature has an overestimation of heat transfer in the enclosure according to Barakos and Mitsoulis [18] study. Therefor we have used the standard $k-\varepsilon$ model with only wall functions on $k$ and $\varepsilon$ at first computational grid points after the wall in order to lower the cost of computation compared to LRN models and have a better prediction of heat transfer inside the enclosure.

The general form of non-dimensional conservative equations of continuity, momentum and energy as well as transport equations of turbulent kinetic energy and dissipation rate for two-dimensional unsteady incompressible flow in the enclosure is as Eq. (1) with parameters defined in Table (1).

$$\frac{\partial \rho \phi}{\partial t} + \frac{\partial \rho U_j \phi}{\partial x_j} = N \frac{\partial}{\partial x_j} \left[ \Gamma \frac{\partial \phi}{\partial x_j} \right] + S$$

When the above general equation is used as a momentum equation, $\lambda_i = 1$ if $g_i \neq 0$ or $\lambda_i = 0$ if $g_i = 0$ are defined in the table for term $S$. The other parameters of the standard $k-\varepsilon$ turbulence model in the table are as follows according to the proposal of Henkes et al [17].

$$c_{\mu} = 0.09, \quad c_{\varepsilon} = 1.44, \quad c_{2\varepsilon} = 1.92, \quad c_{3\varepsilon} = \tanh(u/v), \quad \sigma_f = 1, \quad \sigma_k = 1, \quad \sigma_\varepsilon = 1.3$$

$$P_k = \nu_1 \frac{\sqrt{Pr/Ra}}{2} \left[ \left( \frac{\partial U}{\partial X} \right)^2 + \left( \frac{\partial U}{\partial Y} + \frac{\partial V}{\partial X} \right)^2 + \left( \frac{\partial V}{\partial Y} \right)^2 \right], \quad G_k = - \frac{\nu_1 \sqrt{Pr/Ra}}{\sigma_f \sqrt{Ra} \partial Y} \frac{\partial \theta}{\partial Y}, \quad \nu_1 = C_{\mu} \frac{k^2}{\varepsilon} \sqrt{Pr/Ra}$$

The non-dimensional variables in Table (1) are defined as

$$X, Y = \frac{x, y}{H}, \quad U, V = \frac{u, v}{\sqrt{g/\beta(T_h-T_c)H}}, \quad \theta = \frac{T-T_c}{T_h-T_c}, \quad P = \frac{p}{\rho g/\beta(T_h-T_c)H}, \quad \rho = \frac{\rho}{\rho_{ref}}$$

$$\varepsilon = \frac{\varepsilon}{g/\beta(T_h-T_c)H^{3/2} / H}, \quad K = \frac{k}{g/\beta(T_h-T_c)H}, \quad \nu_1 = \frac{\nu*a}{\nu_{ref}}, \quad t = \frac{I*}{H / \sqrt{g/\beta(T_h-T_c)H}}$$

The subscript * denotes dimensional variables. The density and kinetic viscosity of laminar flow are considered as reference values of density and viscosity respectively. The values of $k = u_r^2 / \sqrt{c_\mu}$ and $\varepsilon = u_r^3 / 0.41Y$ in which $u_r = \sqrt{\rho \partial u / \partial y}$ are considered at the first computational grid points next to the wall.

The dimensionless initial and boundary conditions of the flow field ($\phi = U, V$ or $\theta$) inside the enclosure for all cases are as follows.

$$\phi(X, Y, 0) = 0$$
\[ \phi \ X,0,t = \phi \ X,1,t = \phi \ 0,Y,t = \phi \ 1,Y,t = 0 \] (5)

\[ \phi \ X_b,Y_b,t = 0 \] (6)

The subscript \( b \) denotes the body in the enclosure.

The dimensionless initial and boundary conditions of the temperature field \( (\phi = \theta) \) inside the enclosure for all cases are as follows.

\[ \phi(X,Y,0) = 0.5 \] (7)

\[ \frac{\partial \phi \ X,0,t}{\partial Y} = \frac{\partial \phi \ X,1,t}{\partial Y} = 0 \] (8)

\[ \phi \ 0,Y,t = \phi \ 1,Y,t = \begin{cases} 0, & \text{with one Interior body} \\ 0.5, & \text{with more than one interior body} \end{cases} \] (9)

The subscripts \( H \) and \( c \) denote the hot and cold surfaces.

The local and averaged Nusselt numbers of the enclosure vertical wall as well as the averaged Nusselt number between the air and the surfaces of the internal bodies are defined as follows respectively:

\[ \overline{Nu_w} = -\frac{H}{T_h - T_c} \left. \frac{\partial T}{\partial n}\right|_w = -\left. \frac{\partial \theta}{\partial n}\right|_w \] (10)

\[ \overline{Nu_w} = \frac{1}{H} \int_0^H Nu_w \ dy = \int_0^1 Nu_w \ dY \] (11)

\[ \overline{Nu_b} = \int_{X_w-\frac{w}{2}}^{X_w+\frac{w}{2}} Nu_{left} + Nu_{right} \ dX + \int_{Y_w-h/2}^{Y_w+h/2} Nu_{top} + Nu_{bottom} \ dY \] (12)

4. Numerical Procedure

In the subsequent sections, the computational algorithm for solving governing equations is presented and validated with previous studies and the grid-independence will be checked subsequently.

4.1. Computational Algorithm

To solve the set of the above non-dimensional governing partial differential equations numerically, they are discretized by finite difference method based on finite volume approach in a staggered grid. Then the SIMPLE algorithm suggested by Patankar and Spalding [41] is employed for velocity-pressure coupling. To improve the accuracy of all discretized fluxes near the wall, a finely spaced grid in the \( X \)- and \( Y \)-directions should be used, because the flow variables such as velocity and temperature vary rapidly near the wall surface. However, far away
from the surface, the grid can be coarser. Accordingly, a multi-domains method is utilized to generate a structured mesh at the entire domain of the solution. In order to use the staggered grid, the central \( u \) and \( v \) velocity components have been moved to the west and south side of the main control volume respectively. The other scalar variables are considered at the center of the main control volume. The hybrid and central differencing schemes are used to discretize the convective and diffusion terms of the equations respectively. The hybrid scheme is based on the combination of central and upwind differencing schemes. The central differencing scheme, which is accurate to second order, is employed for small Peclet numbers \( (Pe = u\Delta x / \alpha < 2) \) and the second upwind scheme which is accurate to second order but accounts for transportiveness, is employed for large Peclet numbers \( (Pe \geq 2) \). Time derivatives are discretized implicitly and the spatial derivatives are computed at new time level so that the final discretization leads to formation of systems of linear algebraic equations that should be solved simultaneously. We have adopted two different methods to solve the linear system of equations, line by line iterative and strongly implicit methods. The former method employed the tridiagonal matrix algorithm (TDMA) with alternating sweeps from left to right and bottom to top of the domain for updating variables in the solution of discretized equations. The latter one suggested by Stone [42] enabled us to reach the convergent solution at high Rayleigh numbers quickly.

For the enclosure containing interior bodies, the iterations start at \( Ra = 10^{10} \) with TDMA method. The under-relaxation-factors are set to 0.2 for turbulent kinetic energy and dissipation rate and 0.1 for the other equations. The other under-relaxation-factors required to be changed are initially set to 0.3 for continuity and momentum and 0.7 for energy equations. After a number of iterations when the residuals decrease and their oscillations stop, the solution method for the systems of the linear algebraic equations are changed to strongly implicit method. Subsequently the momentum and energy under-relaxation-factors are gradually reduced to 0.1. Although this procedure brings about the maximum reduction in residuals, it is not as enough as required to meet the convergence criteria which are defined when the residuals drop less than \( 10^{-3} \) for all equations except for energy whereas the residual of the energy equation is less than \( 10^{-6} \). At this stage, the solution is switched over unsteady mode by having access to a more physical initial guess obtained from previous steady state solution leading to a significant decrease in the computational cost. The unsteady calculation based on fully implicit scheme is adopted with time steps equal to \( \Delta t \sqrt{\frac{g\beta \Delta TH}{H}} = 0.25 \). Iterations at each time step are preceded until the residuals meet the defined convergence criteria and then the obtained fields are employed as initial guess for next time step. After careful examination of the iterations it is found that using under-relaxation-factors equal to 0.2 for momentum and 0.8 for energy equations accelerates the rate of convergence in each time step for transient mode. When the solution is converged for \( Ra = 10^{10} \), the results are supplied for calculations of \( Ra = 10^{11} \) as initial guess and this procedure is repeated for \( Ra = 10^{12} \). The evolution of the residuals versus iterations number is indicated in Fig. (2).

Although we have defined convergence criteria based on residuals, the best criterion to determine the converged solution regarding enclosed natural convection is to monitor an integrated quantity such as averaged Nusselt number of bodies. Based on this definition, iterations for each Rayleigh number are continued until the unsteadiness dies out and the solution reaches a steady state. In these circumstances, the averaged Nusselt numbers of the enclosure walls and bodies remain constant and more iteration makes no further changes in the final solution.
4.2. Validation of the Numerical Scheme

In order to validate the present numerical results, the distribution of temperature and velocity in a square enclosure with the isothermal side walls and the adiabatic horizontal walls at $Ra = 1.58 \times 10^9$ are reported in the following figures. Figs. (3a-c) illustrate temperature distribution near the hot and cold walls at level $Y = 0.5$ as well as the vertical velocity. The experimental data of Ampofo and Karayiannis [8] and Salat et al. [9] are also included in the figures for comparison. Although the dimensions and temperature of two enclosures are different in these two experiments, but the Rayleigh and Prandtl numbers in both cases are almost equal. Consequently, these two experiments are treated identically in our investigation, since all of the non-dimensional governing equations, boundary conditions, Rayleigh and Prandtl numbers are the same. These figures confirm the agreement between the present and the previous experimental results.

Figs. (4a-c) show another case of validation in which the vertical velocity at level of $Y = 0.7$ and 0.85, temperature distribution at $X = 0.5$ and the local Nusselt number along the hot wall related to the two tall enclosures with isothermal vertical and adiabatic horizontal walls are plotted at two different Rayleigh numbers of $Ra = 5 \times 10^{10}$ and $Ra = 1.2 \times 10^{11}$. The experimented data of Bowler and Cheesewright [1] and saury et al. [2] are also included for comparison.

Fig. (5) depicts the numerical results of the present work at higher Rayleigh numbers in the range of $10^8 \leq Ra \leq 10^{16}$ with the data of Henkes et al. [17] for comparison. In the figure, the Nusselt number of the hot wall which is obtained by standard $k-\varepsilon$ turbulence model and those reported by Henkes et al. via standard and Chien low Reynolds number $k-\varepsilon$ models are included. There is a close agreement between the results, but the Chien low Reynolds number $k-\varepsilon$ model predicts lower values of the Nusselt number through the wall of the enclosure.

Figs. (6a-c) present the isotherm lines, streamlines and averaged Nusselt number of the hot body at $Ra = 10^7$. To show the credibility of the heat transfer through the isothermal walls of the bodies, a laminar natural convection for water ($Pr = 5.66$) as working fluid is modeled in an adiabatic square enclosure with two heated and two cooled bodies. The data of Garoosi et al. [39] regarding the averaged Nusselt numbers are also presented for comparison.

4.3. Grid Independency

Grid-independent study means that calculation results change so little along with a denser or looser grid that the truncation error can be ignored in numerical simulation. In order to find the proper size of the grid, especially near the walls, grid independency is required to be checked. We have carried out this evaluation for a square enclosure including a single square hot body as shown in Fig. (1a) to find out the best grid size with which the most accurate results with reasonable cost of computation is attainable. Such a grid should be adequately concentrated near the walls of the enclosure and bodies, but can be coarse away from the
surfaces. We have chosen five initial values for the size of the first cell adjacent to the wall and kept the concentration factor constant leading to generation of the grids whose sizes are reported in Table (2). Fig. (7) indicates the grid independence results for the averaged Nusselt number of the left wall and local Nusselt number at its middle regarding the enclosure which includes a single heated body at three Rayleigh numbers of $10^{10}$, $10^{11}$ and $10^{12}$. It is observed that both Nusselt numbers approach their corresponding constant values when the number of grid cells is about $144^2$ for all aforementioned Rayleigh numbers.

5. Results and Discussion

The numerical method is used to simulate the turbulent natural convection inside an enclosure including four square or rectangular bodies. The preliminary investigations have been devoted to the enclosure with one interior hot body at $Ra = 10^{12}$ and further investigations are focused on the enclosure with two hot and two cold bodies in the range of Rayleigh numbers of $10^{10} \leq Ra \leq 10^{12}$.

5.1. Enclosure with One Interior Body

Figs. (8a, b) report streamlines and isotherms inside an enclosure with a single square interior hot body. In Fig. (8a), natural convection initiates in the proximity of the hot body above which a plume ascends toward the upper insulated wall establishing two vortices in upper part of the enclosure. The right hand side of which is clockwise and the other one is counter-clockwise. Subsequently, downward fluid flow forms near the wall of the enclosure. The temperature field takes shape in such a way that thermal stratification is established in the core region with a zero gradient in the horizontal direction except near the walls. At lower parts of the enclosure, no flow is induced by natural convection and the isotherm values are zero and equal to the temperature of the side walls. The results indicate the averaged Nusselt number of the body is $\bar{Nu}_b = 699$ and the corresponding value to each vertical wall is $\bar{Nu}_w = 338$ which shows that the heat transfer of the body is almost equal to the heat transfer of both side walls combined.

5.2. Enclosure with Four Interior Bodies

Figs. (9a-d) illustrate streamlines in four enclosures with different interior bodies, but with the same surface area equal to 1. In all of the enclosure except “b”, the centers of bodies are located at the intersection of the upper and lower midwidth lines with the left and right midheight lines of the enclosure so that in case “a” the distance between the edges of bodies equals to 0.25 which is bisected in the second state of “b” to investigate its effect on the heat transfer rate inside the enclosure. In cases “c” and “d” the effect of aspect ratio is due to be investigated and they are installed with $h/w = 1/3$ and 3 respectively in these two states. A pair of vortices is formed on the top and below of the hot and cold bodies respectively. The initial increase and decrease in air temperature at the
vicinity of bodies begin by conductance mechanism of heat transfer. Then due to the appearance of a temperature difference inside the domain the heat transfer mechanism is changed to natural convection which induces downward and upward flows adjacent to the cold and hot bodies leading to formation of descending and ascending plumes below and above them respectively. These plumes reach the top and bottom insulated walls and circulating there move toward the side walls of the enclosure. Since the temperature of flows at upper part of the enclosure is greater than the vertical walls, downward flows take shape in the proximity of these walls. At lower part of the enclosure leading to formation of four similar vortices beneath the cold bodies. It is also obvious that the size of vortices is proportional to the distance between the bodies and the walls of the enclosure. Therefore, when the bodies are placed closer to each other at state (b) or installed horizontally at state (c) the distance between the bodies and the walls increases and as a result two larger vortices are formed compared to the first state of (a). Similarly, the size of these vortices is reduced when the bodies are positioned vertically at state (d). In the latter state, there are only four vortices within the enclosure and the other vortices closer to the side walls disappear due to the very narrow distance existed between the bodies and insulated walls.

Figs. (10a-b) represent the vertical velocity at the midheight above and below the left hot and cold bodies respectively in order to have a more in-depth view over the induced flow and to see how the distance between the bodies affects the velocity field. The magnitude of the maximum velocity at the midheight above and below the bodies is also proportional to the distance between the bodies and insulated walls of the enclosure. These velocities will be enhanced about 45 percent when the bodies get closer to each other from \( d = 0.25 \) to \( d = 0.125 \). Furthermore, the locations of the peaks in the X- direction are exactly coincided with the position of ascending and descending plumes over and below the hot and cold bodies respectively. In a similar manner, to see how the aspect ratio affects the velocity field, the vertical velocity at the same previous position for three aspect ratios of \( h/w = 1/3, 1 \) and 3 corresponding to states (a), (c) and (d) are plotted respectively in Figs. (10c-d). While horizontal installation of bodies with \( h/w = 1/3 \) results in 45 percent increase in maximum velocities. Installation of vertical position with \( h/w = 3 \) has a greater impact on maximum velocities and reduces them by 75 percent compared to the configuration with square bodies and \( h/w = 1 \).

Figs. (11a-d) reports the detailed information of temperature field within the enclosures of Figs. (9a-d). The temperature field is completely stratified in the entire domain and the temperature gradient along the horizontal direction is almost zero except for the regions in the vicinity of walls. The boundary layer adjacent to the side walls and near the adiabatic horizontal walls is extremely thin and thickens by moving toward the downstream of the flow. Such behavior of boundary layer results in the variation of local Nusselt number of vertical walls to be the same as Figs. (12a, b). In fact, the high amount of heat transfer rate occurs at location where the boundary layer is thin and decreases when the boundary later is thicken. Accordingly, the local Nusselt number reaches exceedingly close to zero at the middle of the side walls corresponding to a maximum thickness of the boundary layer and increases afterward. It’s worth noting that, at upper half of the enclosure the fluid temperature is greater than the vertical walls temperature and the heat transfer direction is from the air to the walls of the enclosure. Conversely, at lower half of the enclosure the direction of heat transfer is from the walls to the air.

Table (3) provides the information of the averaged Nusselt numbers of bodies in the range of \( 10^{10} \leq Ra \leq 10^{12} \). In the table, “a” and “b” belong to the bodies of square shape and “c” and “d” belong to the rectangular shape.
When the gap between the bodies is reduced from $d = 0.25$ to $d = 0.125$ the averaged Nusselt number is promoted to 3.8, 7.3 and 11.9 percent for $10^{10}$, $10^{11}$ and $10^{12}$ respectively. When the aspect ratio is changed from $h/w = 1$ to $1/3$ the rate of the Nusselt number is changed to 12.9, 8.3 and 3.8 percent corresponding to the Rayleigh numbers of $10^{10}$, $10^{11}$ and $10^{12}$ respectively. When the aspect ratio is $h/w = 3$, this configuration brings about 3.1, 3.5 and 4.8 percent of Nusselt number augmentation corresponding to Rayleigh numbers of $10^{10}$, $10^{11}$ and $10^{12}$ respectively. Moreover, the averaged Nusselt number of all configurations is promoted nearly by 100 percent for increasing of the Rayleigh number from $10^{10}$ to $10^{11}$ and to $10^{12}$ subsequently.

Concluding Remarks

This study focused on numerical solution of turbulent natural convection inside a square air-filled enclosure including one or four hot and cold bodies via standard $k-\varepsilon$ turbulence model based on the standard wall functions approach at high Rayleigh numbers. The shape of the internal bodies is square or rectangular with different aspect ratios but with the same surface areas. The horizontal and vertical walls of the enclosure are considered adiabatic and isothermal respectively. The main purpose of the present work is to investigate the effects of distance between the bodies, their aspect ratios and governing the Rayleigh number on heat transfer rate. The findings of this work indicate that the critical Rayleigh number in which the transition from laminar to turbulent flow takes place is about $10^{9}$. In all considered configurations, temperature field is partly or entirely stratified especially in the region over the cold and under the hot bodies for the enclosure containing two pairs of heated and cooled bodies. This stratification is intensified with the reduction of mentioned distance when the bodies are mounted in a halved distance from each other or oriented vertically with $h/w = 3$. Moreover, the most reduction in the Nusselt number at high Rayleigh numbers is achieved by doubling the distance between square bodies through which the Nusselt number decreases by 3.8, 7.3 and 11.9 percent corresponding to $Ra = 10^{10}$, $10^{11}$ and $10^{12}$ respectively. The maximum reduction in the Nusselt number at low values of the Rayleigh number pertains to horizontal installation of bodies with $h/w = 1/3$ which results in 12.9, 8.3 and 3.8 percent of reduction in the Nusselt number at the Rayleigh numbers of $10^{10}$, $10^{11}$ and $10^{12}$ respectively. The vertical positioning of bodies with $h/w = 3$ is accompanied by 3.1, 3.5 and 4.8 percent of promotion in the Nusselt number at mentioned Rayleigh numbers. Moreover, increasing the Rayleigh number from $10^{10}$ to $10^{11}$ and to $10^{12}$ the Nusselt number is approximately augmented by 100 percent for all configurations.

Nomenclature

\begin{align*}
  c & \quad \text{constant coefficients} \\
  d & \quad \text{dimensionless distance between bodies} \\
  g & \quad \text{gravitational acceleration, } m\ s^{-2} \\
  G_k & \quad \text{production of turbulent kinetic energy} \\
  h & \quad \text{body height, } m \\
  H & \quad \text{enclosure height, } m \\
  k & \quad \text{turbulent kinetic energy, } m^2/\ s^2 \\
  X, Y & \quad \text{dimensionless Cartesian coordinates, } x/H, y/H
\end{align*}

Greek Symbols

\begin{align*}
  \beta & \quad \text{thermal expansion coefficient, } 1/K \\
  \varepsilon & \quad \text{dimensionless kinetic energy dissipation rate} \\
  \phi & \quad \text{general variable} \\
  \lambda & \quad \text{parameter}
\end{align*}
\( K \)  
\[ \text{dimensionless turbulent kinetic energy} \]

\( N \)  
\[ \text{dimensionless numbers} \]

\( \text{Nu} \)  
\[ \text{local Nusselt number} \]

\( p \)  
\[ \text{pressure, Pa} \]

\( P \)  
\[ \text{dimensionless pressure} \]

\( P_k \)  
\[ \text{shear production of turbulent kinetic energy} \]

\( Pr \)  
\[ \text{Prandtl number, } \nu/\alpha \]

\( \dot{q}'' \)  
\[ \text{heat flux, W/m}^2 \]

\( R \)  
\[ \text{residual} \]

\( Ra \)  
\[ \text{Rayleigh number, } g\beta (T_h - T_c) H^3/\alpha \]

\( S \)  
\[ \text{source term} \]

\( t \)  
\[ \text{dimensionless time} \]

\( T \)  
\[ \text{temperature, } K \]

\( \Delta T \)  
\[ \text{temperature difference, } T_h - T_c \]

\( u, v \)  
\[ \text{dimensional velocity components, m/s} \]

\( U, V \)  
\[ \text{dimensionless velocity component} \]

\( u_t \)  
\[ \text{shear velocity, } \sqrt{\rho \partial U_i/\partial y_n} \]

\( w \)  
\[ \text{body width, } m \]

\( x, y \)  
\[ \text{Cartesian coordinates, } m \]

\( \Gamma \)  
\[ \text{transport property} \]

\( \nu \)  
\[ \text{dimensionless kinematic viscosity} \]

\( \theta \)  
\[ \text{temperature, } (T - T_c)/(T_h - T_c) \]

\( \rho \)  
\[ \text{dimensionless density} \]

\( \sigma \)  
\[ \text{turbulent Prandtl number} \]

**Subscripts**

- \( b \)  
  \[ \text{interior body} \]

- \( c \)  
  \[ \text{cold, center of body} \]

- \( h \)  
  \[ \text{hot} \]

- \( n \)  
  \[ \text{closest grid to wall} \]

- \( ref \)  
  \[ \text{reference} \]

- \( t \)  
  \[ \text{turbulent} \]

- \( w \)  
  \[ \text{enclosure wall} \]

- \( \varepsilon \)  
  \[ \text{dissipation energy rate} \]

- \( k \)  
  \[ \text{turbulent kinetic energy} \]

**Superscripts**

- \( * \)  
  \[ \text{average} \]

- \( * \)  
  \[ \text{dimensional variable} \]

**References**


steady natural convection in a horizontal enclosure with a square body

Natural convection in differentially heated and partially divided cavities with internal heat generation


Biographies

A. Nouri-Borujerdi is Professor of Mechanical Engineering at Sharif University of Technology, Tehran, Iran, where he teaches undergraduate and graduate level courses in the Thermal/Fluids Sciences in school of Mechanical Engineering. His teaching focuses on heat transfer, computational fluid dynamics and two-phase flows, including boiling and condensation. His current research programs include experimental, theoretical and numerical works in heat and mass transfer, two-phase flow, and porous media. Professor Nouri has published more than 150 articles in international journals and conferences.

F. Sepahi received his B.S. degree from Shiraz University and M.S. degree from Sharif University of Technology in school of Mechanical Engineering in 2013 and 2016, respectively. His research interests include: mathematical and numerical modeling, applications of computational fluid dynamics in heat transfer and turbulence modeling with RANS, LES and DNS approaches.
Caption of Tables

Table 1. Definition of corresponding dimensionless parameters in Eq. (1)
Table 2. Size of the first cell adjacent to the walls and number of grid cells
Table 3. Averaged Nusselt numbers of bodies in the enclosure for four cases, (a-d)

Caption of figures

Fig. 1 Schematic of enclosures with boundary conditions, (a) including a single hot body, (b) including two hot and two cold bodies
Fig. 2 Evolution of residuals versus iteration numbers for all discretized equations
Fig. 3 (a) temperature near the hot wall, (b) temperature near the cold wall, (c) vertical velocity, all at $Y = 0.5$,
Fig. 4 (a) vertical velocity at $Y = 0.7$ and 0.85 near the hot wall, (b) temperature distribution at $X = 0.5$, (c) local Nusselt number along the hot wall.
Fig. 5 Averaged Nusselt number for hot wall in the square enclosure
Fig. 6 Enclosure filled with water ($Pr = 5.66$), (a) isotherm lines, (b) streamlines at $Ra = 10^7$, (c) averaged Nusselt number of hot body
Fig. 7 Averaged and local Nusselt numbers of the enclosure’s left wall and at $Y = 0.5$
Fig. 8 Air filled square enclosure. (a) streamlines, (b) isotherms at $Ra = 10^{12}$
Fig. 9 Streamlines in a square enclosure at $Ra = 10^{12}$, horizontal walls are adiabatic, both vertical walls are at $\theta_w = 0.5$, hot bodies are at $\theta_h = 1$, cold bodies are at $\theta_c = 0$.
Fig. 10 Vertical velocity at the midheight above the left hot bodies (a, c) and under the left cold bodies (b, d), (a, b) the effect of the distance between bodies, (c, d) the effect of aspect ratio
Fig. 11 Isotherms in a square enclosure at $Ra = 10^{12}$, horizontal walls are adiabatic, both vertical walls are at $\theta_w = 0.5$, hot bodies are at $\theta_h = 1$, cold bodies are at $\theta_c = 0$.
Fig. 12 Local Nusselt number along the left wall of the enclosure including two hot and two cold bodies
<table>
<thead>
<tr>
<th>Equation</th>
<th>φ</th>
<th>Γ</th>
<th>N</th>
<th>S</th>
</tr>
</thead>
<tbody>
<tr>
<td>Continuity</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Momentum</td>
<td>(U_i)</td>
<td>(\nu + \nu_t)</td>
<td>(\sqrt{Pr/Ra})</td>
<td>(-(\partial P/\partial x_i)/\rho - \chi_i)</td>
</tr>
<tr>
<td>Energy</td>
<td>(\theta)</td>
<td>(\nu/Pr + \nu_t/\sigma_t)</td>
<td>(1/\sqrt{PrRa})</td>
<td>(0)</td>
</tr>
<tr>
<td>Turbulence</td>
<td>(K)</td>
<td>(\nu + \nu_t/\sigma_k)</td>
<td>(\sqrt{Pr/Ra})</td>
<td>(P_k + G_k - \varepsilon)</td>
</tr>
<tr>
<td>Turbulence</td>
<td>(\varepsilon)</td>
<td>(\nu + \nu_t/\sigma_\varepsilon)</td>
<td>(\sqrt{Pr/Ra})</td>
<td>(</td>
</tr>
</tbody>
</table>

**Table 5**

<table>
<thead>
<tr>
<th>No. of grid</th>
<th>(96^2)</th>
<th>(112^2)</th>
<th>(130^2)</th>
<th>(144^2)</th>
<th>(162^2)</th>
</tr>
</thead>
<tbody>
<tr>
<td>First cell size</td>
<td>(2.5 \times 10^{-4})</td>
<td>(1.25 \times 10^{-4})</td>
<td>(5 \times 10^{-5})</td>
<td>(2.5 \times 10^{-5})</td>
<td>(10^{-5})</td>
</tr>
</tbody>
</table>

**Table 6**

<table>
<thead>
<tr>
<th>Ra</th>
<th>position of body</th>
<th>(\bar{Nu}_b)</th>
</tr>
</thead>
<tbody>
<tr>
<td>a</td>
<td>b</td>
<td>c</td>
</tr>
</tbody>
</table>

16
<table>
<thead>
<tr>
<th>10^{10}</th>
<th>( \sum_{i=1}^{n} (Nu_b)_i )</th>
<th>30.01</th>
<th>31.17</th>
<th>26.14</th>
<th>30.96</th>
</tr>
</thead>
<tbody>
<tr>
<td>compared with</td>
<td>(a)</td>
<td>1</td>
<td>3.8%</td>
<td>-12.9%</td>
<td>3.1%</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>10^{11}</th>
<th>( \sum_{i=1}^{n} (Nu_b)_i )</th>
<th>57.29</th>
<th>61.49</th>
<th>52.5</th>
<th>59.30</th>
</tr>
</thead>
<tbody>
<tr>
<td>compared to</td>
<td>(a)</td>
<td>1</td>
<td>7.3%</td>
<td>-8.3%</td>
<td>3.5%</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>10^{12}</th>
<th>( \sum_{i=1}^{n} (Nu_b)_i )</th>
<th>113.26</th>
<th>126.71</th>
<th>108.94</th>
<th>118.63</th>
</tr>
</thead>
<tbody>
<tr>
<td>compared with</td>
<td>(a)</td>
<td>1</td>
<td>11.9%</td>
<td>-3.8%</td>
<td>4.8%</td>
</tr>
</tbody>
</table>
Fig. 3

(a) $\theta = \frac{T - T_c}{T_h - T_c}$

(b) $\theta = \frac{T - T_c}{T_h - T_c}$

(c) $Y = \sqrt{\frac{H \Delta T}{a}}$

- Present Study
- Salat et al. [9]
- Ampofo et al. [8]

Ra = $1.58 \times 10^9$
Y = 0.5
Fig. 6

Fig. 7
Fig. 8

Fig. 9
Fig. 10