Steady and unsteady mixed convection flow in a cubical open cavity with the bottom wall heated

Gorg Abdelmassih, Anton Vernet, Jordi Pallares *

Department d’Enginyeria Mecànica, Universitat Rovira i Virgili, Av. Països Catalans 26, 43007 Tarragona, Spain

ARTICLE INFO

Article history:
Received 28 January 2016
Received in revised form 4 May 2016
Accepted 17 May 2016

Keywords:
Mixed convection
Open cavity
PIV
DNS
Turbulent

ABSTRACT

In this study we analyze experiments and numerical simulations of steady and unsteady mixed convection flow in a cubical cavity located at the bottom of a square channel. The Reynolds numbers based on the mean flow velocity and the channel width are in the range 100 \( \leq Re \leq 1500 \) and the Richardson numbers vary within 0.1 \( \leq Ri \leq 10 \). Particle Image Velocimetry has been used for the measurements in a water channel. Three-dimensional direct numerical simulations have been carried out with a second order finite volume code considering the Boussinesq approximation since, for the experimental conditions considered, the variation of the physical properties with temperature has no significant influence on the overall flow topology. For 100 \( \leq Re \leq 1500 \) and 0.1 \( \leq Ri \leq 10 \) the flow is steady and it consists in a single roll that exhibits larger velocities as the Richardson number is increased. An unsteady periodic flow is found at Re = 100 and Ri = 10. Alternate flow ejections from the cavity to the channel occur near the lateral walls while the flow enters the cavity from the channel through the central part of the cavity. A conditional sampling technique has been used to elucidate the evolution of the mean unsteady turbulent flow at Ri = 10. Computed Nusselt numbers are in general agreement with a previously reported correlation, valid for two dimensional cavities of different aspect ratios.

© 2016 Elsevier Ltd. All rights reserved.

1. Introduction

During several decades efforts have been directed to analyze fluid flow and/or heat transfer processes in flows over heated open cavities due to its importance in several engineering applications. This type of geometry can be found, for example, in the landing gear wells [1–6]. Cooling of electronic components [5–7] is one of the most used applications at small scales for the flow over an open cavity. It is simple in design and has a cheap maintenance cost. The electronic component considered as the source of heat and the natural or mixed convection effecting on the flow structure is the crucial point. It can be classified under three main categories regarding to the location of the heated wall, e.g. the electronic board, which can be located on the leading vertical wall (natural convection assisting flow), on the trailing vertical wall (natural convection opposing flow) or on floor of the cavity (heated from below), see [7–11].

In renewable energy research it has been reported that the installation of wind barriers along the perimeter of solar cells improves their absorption efficiency. The geometry of the flat solar collectors with the wind barriers has the same geometry as an open cavity at medium scales.

Early studies of flow past cavities were achieved in the 1960s [1–3]. More studies followed to reveal the flow structure and the heat transfer process occurring for both natural and mixed convection for geometries with different aspect ratios [4–6,9]. Manca et al. [7], investigated experimentally the opposing mixed convection flow in an open cavity with a heated wall bounded by a horizontal unheated plate for (100 \( \leq Re \leq 2000 \)) and (4.3 \( \leq Ri \leq 6400 \)). The cavity aspect ratio was (0.5 \( \leq AR \leq 2.0 \)). They reported that for low Reynolds numbers, the forced motion penetrates inside the cavity, and the vortex structure is adjacent to the unheated vertical plate. At higher Reynolds numbers, the vortex structure has a larger extension while AR is held constant. The effects of the position of a heated wall on mixed convection in a channel with an open cavity have been studied numerically in [8]. A two-dimensional numerical approach was considered with different aspect ratios. The authors found that the maximum temperature values decrease as the Reynolds number and Richardson number increase for all the studied configurations.

Three dimensional numerical studies of the flow and heat transfer characteristics for assisting and opposing incompressible laminar flow past an open cavity can be found in Stiriba [10] and Stiriba...
et al. [11]. They reported that the flow exhibits a three-dimensional structure and it is steady for \( Re = 100 \) with \((0.001 \leq Ri \leq 10)\) and \( Re = 1000 \) with \((0.001 \leq Ri \leq 1)\). The forced flow dominates the transport mechanism and a large recirculating zone occurs inside the cavity which results in heat transfer mainly by conduction. Abdelmassih et al. [12], performed numerical simulation of incompressible laminar flow in a three-dimensional channel with a cubic open cavity with a bottom wall heated. Air-flow has been used in the inflow. They noted that the buoyancy is weak and not affect the steadiness of the flow for small Richardson numbers \( Ri \leq 0.1 \) on a range of \( Re \leq 1000 \).

O’Hern et al. [13], carried out an experimental study using Particle Image Velocimetry (PIV) techniques for an isothermal configuration of an open cavity. Water inflow in isothermal configuration was used with steady state in the range of Reynolds numbers from 100 to 900. These authors reported the difficulties found using PIV because the maximum velocity inside the cavity is many times smaller than the flow velocity over the cavity. Stiriba et al. [14], compared the velocities of the air-flow structure within a rectangular cavity heated from below. They reported that increasing the Richardson number generates remarkable velocities differences while the inflow Reynolds is held constant. Zamzari et al. [15], performed a two dimensional study of the entropy generation and mixed convection in a horizontal channel with an open cavity. Laminar air-flow in the ranges 200 \( \leq Re \leq 500 \) and 0.25 \( \leq Ri \leq 1 \) is considered. Their results show that the cavity flow, heat transfer rates and entropy generation are strongly affected by variations of the Reynolds number, Richardson number and the aspect ratio. Most of the works found in the literature are numerical studies for both two and three dimensional configurations with a varying \( Re, Ri \) and \( AR \). However a few experimental studies have been reported specially using PIV techniques because its measurement challenges [13]. In addition, the flow structure and its time evolution when the flow becomes unsteady have not been analyzed in detail [12][14][16]. The main objective of this work is to study the mixed convective flow and three dimensional instabilities, focusing on the effect of the buoyancy forces on the flow topology for an incompressible laminar water inflow in an open cavity with a bottom wall heated. The ranges of Reynolds and Richardson numbers considered \((100 \leq Re \leq 1500\) and \(0.1 \leq Ri \leq 10)\) correspond to the usual operating conditions of some processes associated with the cooling of electronic components. The effect of \( Ri \) on the flow stability and the analysis of the flow topology and its time evolution when the flow is unsteady are the main parts of this work. The experimental measurements allowed the confirmation of the numerical results.

2. Numerical approach

2.1. Physical problem and computational domain

The geometry of the channel including the cubical open cavity of size \( L \) and the computational domain used in this study are shown in Fig. 1. The inflow located at \( x = 0 \) has uniform velocity \( U_0 \) and temperature \( T_{in} \). Convective Euler boundary conditions are imposed at the outflow located at \( x = 4L \) and the non-slip boundary condition is applied for the rest of the boundaries. The cavity is heated from below at a constant temperature \( T_{inh} \) and the remaining walls of the cavity and of the channel are adiabatic. A cooling incompressible water-flow has been used. The Prandtl number of water is \( Pr = 7 \).

2.2. Governing equations and numerical method

The 3DINAMICS finite volume parallel code has been used in this work [10–12,14]. The code solves numerically the three-dimensional incompressible Navier–Stokes Eqs. (1–3) for mixed convection [17], on non-uniform staggered Cartesian meshes. The SMAC-method is used to join continuity and momentum equation, in which, the Poisson equation for the pressure is computed with the biconjugate gradient method (BiCGtab). The convective and

\[ \text{Greek letters} \]

- \( \Delta \) increment
- \( \chi \) thermal diffusivity
- \( \beta \) thermal expansion coefficient
- \( \lambda_2 \) second largest eigenvalue of the velocity gradient tensor
- \( \nu \) kinematic viscosity

\[ \text{Superscripts and subscripts} \]

- * non-dimensional quantity
- \( \infty \) reference value
- \( H \) hot
- \( l \) local
- \( S \) surface averaged
- \( w \) wall

---

**Nomenclature**

- AR aspect ratio
- \( f \) frequency
- \( g \) gravity acceleration
- \( Gr \) Grashof number
- \( L \) cavity dimensions
- \( Nu \) Nusselt number
- \( p \) pressure
- \( Pr \) Prandtl number
- \( Re \) Reynolds number
- \( Ri \) Richardson number
- \( t \) time
- \( T \) temperature
- \( u, v, w \) velocity components
- \( U_0 \) inflow velocity
- \( V \) velocity vector
- \( x, y, z \) cartesian coordinates

---

**Fig. 1.** Sketch of the computational domain.
the diffusive terms are approximated using the SMART scheme [18] and central differences respectively.

\[
\nabla \cdot \mathbf{v} = 0
\]

\[
\frac{\partial \mathbf{v}}{\partial t} + (\mathbf{v} \cdot \nabla) \mathbf{v} = -\nabla p + \frac{1}{Re} \nabla^2 \mathbf{v} - RT \frac{\theta}{\theta t}
\]

\[
\frac{\partial T}{\partial t} + (\mathbf{v} \cdot \nabla) T = \frac{1}{Pr} \nabla^2 T
\]

The dimensionless variables are defined as:

\[
\tilde{x} = \frac{x}{L}, \quad \tilde{t} = \frac{U_T t}{L}, \quad \tilde{v} = \frac{v}{U_T}, \quad \tilde{p} = \frac{p}{\rho \omega U_T^2}, \quad \tilde{T} = \frac{T - T_w}{T_h - T_w}, \quad Ri = \frac{Gr}{Re^2}
\]

where \(U_0\) is the inflow velocity, \(\rho_\infty\) is the reference density, \(T_h\) is the temperature at the heated wall surface, \(Re = (U_0 L)/\nu\) is the Reynolds number, \(Ri\) is the Richardson number, \(Gr\) is the Grashof number, \(Re = (\beta \rho_\infty \Delta T L^3)/\nu^2\), and \(Pr\) is the Prandtl number, \(Pr = \nu/\alpha\). Here \(\beta\), \(\nu\) and \(\alpha\) are the coefficients of volumetric expansion, the kinematic viscosity, and the thermal diffusivity, respectively.

\[
Nu_l = \frac{\partial T}{\partial y}|_{y=0}, \quad Nu_s = \int_0^1 \int_0^2 Nu_{l} dx' dz'
\]

In Eq. (4), \(Nu_l\) is the local Nusselt number and \(Nu_s\) is the surface average Nusselt number. The validation of the numerical code can be found in [10–12,14].

2.3. The computational grid

Non-uniform grids have been used in this work. The grid is finer close to the walls to capture the steep changes in temperature and velocity, see (Fig. 2a). The grid is uniform along the \(z\) direction. To check the grid independence of the numerical results, three different grids were tested: \(60x60x60\), \(100x100x100\) and \(150x150x150\) grid points inside the cavity, respectively. The grid independence was checked at \(Re = 1500\) and \(Ri = 10\). Fig. 2b, shows the comparison of the temperature profile at \(x^* = 1.5\) and \(z^* = 0.5\) obtained with different grids. The results obtained with the grid of \(200 \times 100 \times 50\) nodes show a difference of less than 2% in the time-averaged temperature and the time and surface averaged Nusselt numbers at the heated wall in comparison with the other two meshes. Consequently, the grid of \(200 \times 100 \times 50\) (\(50^3\) grid points inside the cavity) is used for the simulations.

3. Experimental approach

3.1. Experimental technique

Particle Image Velocimetry (PIV) is used for experimentally measuring the two in-plane velocity components in the cavity [19]. The concept of PIV is that the velocity of the flow can be measured by tracing the motion of small tracer particles seeded in the fluid. This process can only be done accurately if the particles are small enough to completely follow the fluid flow, but at the same time they need to be large enough to scatter the light. The positions of the illuminated particles in a thin sheet are recorded twice with a digital camera. Velocity field of the fluid can be estimated from the displacement of the particles according to the time between two consecutive images. In this case the cross correlation between the two images allows the calculation of the displacement of the particles. See for example references [20] and [21] for extensive descriptions of the technique.

3.2. Experimental setup

The experimental model consists in a cubical cavity with a dimension of 100 mm. The total length of the channel is 500 mm while the length from the inflow tank to the cavity leading edge is 100 mm. The cavity and the channel walls are made from transparent poly-methyl-methacrylate.

The heated wall is made from copper metal. The copper is characterized by a high thermal conductivity which allows a homogeneous temperature distribution on the heated wall surface. The Cooper plate has been thermally insulated to avoid effects of the surrounding temperature. The bottom wall of the channel and the cavity were painted black in order to reduce the reflections of the laser light sheet and improve the quality of the images at the edges.

Six electric resistances (2500 Ohms) are used to heat the copper flat plate. Each resistance has a calibrated thermocouple (type “k”) to monitor the temperature. The thermocouples have a uncertainty of 0.1 K. Thermal paste was used to increase the effective thermal conductivity of the interface between the resistances and the cooper block. The cooper block was heated in a range of temperature from 293 to 305 K. Water inflows at 293 K while a temperature control was used to maintain a constant water temperature at the inflow.

Fig. 3, shows the flow cycle. The flow is supplied by a main tank, with a capacity of about 1 m³. A centrifugal pump with flow rates up to 2.5 m³/h transports the water to the inflow tank trough a PVC.
pipe (25 mm ID). After the pump the fluid flows through a valve and a flow-meter to control the flow rate. Then the fluid is conducted to the inflow tank from the top and the bottom in order to minimize the creation of eddies.

The water in the inflow tank passes through seven plastic mesh screens (1 mm x 1 mm) separated 15 mm. The screens are divided into two groups separated by a honeycomb (10 mm x 10 mm with 30 mm thickness). A nozzle is placed at the end of the inflow tank to conduct the water to the channel. The outflow is collected in a drain tank located at a higher height than the central channel and the inflow tank to eliminate the air bubbles from the flow system. Finally the water is conducted from the drain tank to the main tank.

A Nd-Yag green laser, Monocrom MP532-3W is used to illuminate planes of the flow. This laser is linearly polarized with 532 nm wavelength and it was used in continuous mode. The optical system composed of 3 lenses is mounted on the laser exit to generate a light sheet with a thickness of 0.7 mm. MotionPro digital cameras, model HS-3, are used to record images of the illuminated particles. The sensor of the camera has a resolution of 1280 x 1024 pixels with square pixels of 12 μm. A zoom lens Sigma, 28–300 mm F3.5–6.3 DG Macro has been used. Polyamide particles with a mean diameter of 50 μm have been used to seed the flow.

Different points have been considered to obtain a good accuracy during the measurements. The image frame was focused on the whole cavity plus the 5% of the flow in the channel. The flow cycle was running for more than 30 min before starting the image capture to avoid fluctuations generated by the pump as well as to eliminate air bubbles in the flow cycle.
Fig. 6. Contours of $v'$ and temperature distribution in several planes at $Re = 100$. (a–d) Isothermal configuration and (e–h) heated bottom wall configuration at $Ri = 1$.

Fig. 7. Periodic pattern of $Re = 100$ at $Ri = 10$. (a) $u'$ velocity component and (b) Power spectrum.

Fig. 8. Vertical $v'$ velocity profile at cavity border cross-section ($y' = 1$). (a) $v'$ velocity at $t'/0$, (b) $v'$ velocity at $t'/0 + 2$, (c) $v'$ velocity at $t'/0 + 3.5$ and (d) $v'$ velocity at $t'/0 + 5.5$. 
4. Results and discussion

4.1. Effect of temperature dependent physical properties

The ranges of Reynolds and Richardson number used in this work are $100 \leq Re \leq 1500$ and $0.1 \leq Ri \leq 10$, respectively. The Prandtl number used for water is kept constant at 7. Before considering constant physical properties in the numerical approach in this range of $Re$ and $Ri$, the effect of temperature dependent physical properties has been studied numerically. The comparison between the velocity components at $Re = 1500$ with $Ri = 10$ shows an average velocity difference of 4% considering the variation of the thermal expansion coefficient and viscosity with temperature. The corresponding difference in the surface averaged Nusselt number is less than 5%. Consequently, it can be assumed that the temperature dependent fluid properties are not affecting the structure of the water-flow in the range of $Re$ and $Ri$ considered.

4.2. Experimental validation of the numerical results

Several experiments were carried out to verify the numerical results. Fig. 4, corresponds to $Re = 1500$ and $Ri = 10$. In this case the difference of the temperature between the heated wall surface and the inflow water was 11°C. Fig. 4 shows the velocity field at the vertical symmetry plane of the cavity. The velocity at the bottom part of the channel can be seen at the top of the figures. The numerical and experimental velocity vectors are shown in (Fig. 4a and b), respectively. The difference between the numerical and experimental results is a shift of about a 5% in the location of the center of the main recirculation. Comparisons between the numerical and experimental velocity components along the vertical and horizontal bisectors are shown in Fig. 5. On average the velocity difference is less than 6% and it can be attributed to the different location of the center the main vortex.

4.3. Numerical results

The isothermal configuration was studied in the same range of $Re$ in order to analyze the effect of heating the bottom wall of the cavity on the flow stability. The list of the studied cases and the observed flow regimes are shown in Table 1.

![Fig. 9. Instantaneous flow structures at $Re = 100$ at $Ri = 10$ in terms of an isosurface of $\lambda_2$. (a) corresponds to the time of Fig. 8b and (b) to the time of Fig. 8d.](image)

![Fig. 10. The time-averaged vortex core at $Re = 1500$ and $Ri = 10$.](image)

![Fig. 11. (a) The instantaneous $w$ velocity in a specific node with time and (b) Power spectrum of the $w$.](image)
The flow in the isothermal configuration is steady in all cases. In the heated bottom wall configuration the flow remains steady for all cases at $Ri = 0.1$. The flow becomes unsteady for $Ri = 1$ and 10 except for the case of $Re = 100$ and $Ri = 1$. At $Re = 100$ and $Ri = 10$ the flow is periodic while in the other cases the flow is turbulent.

4.3.1. Steady flow pattern

The steady flow pattern at $Re = 100$ is observed for the isothermal case and at small Richardson numbers ($Ri \leq 1$). Fig. 6, shows the vertical velocity component, $v'$, at $Re = 100$ for the isothermal configuration and the heated bottom wall configuration at $Ri = 1$.

Fig. 6b and f, show the $v'$ vertical velocity contours at $y' = 0.5$ for both configurations. The flow descends near the right half and ascends near the left half of the cavity. The main difference between both cases is found in the intensity of the velocity, which is larger in the mixed convection case because of the assisting effect of the buoyancy term in the vertical momentum equation. Comparison of Fig. 6a and e shows that the flow exchange between the cavity and the channel is similar in both cases. This also can be observed comparing Fig. 6c and d with Fig. 6g and h, which show a very similar distribution of velocities and streamlines within the channel and differences located inside the cavity.

4.3.2. Periodic flow pattern

The periodic flow pattern is observed at $Re = 100$ and $Ri = 10$. The evolution of the $x$ component of the velocity ($u'$) at $x' = 1.2$,
\[ y^* = 0.5, z^* = 0.5, \] which is located in the symmetry plane close to the leading vertical wall of the cavity, is shown in Fig. 7a. The power spectrum (Fig. 7b) shows a non-dimensional main frequency of \( f^* = 2/7 \) and a harmonic at \( f^* = 1/7 \). These frequencies correspond to non-dimensional periods of 3.5 and 7, respectively.

The contours of the instantaneous vertical velocity component at \( y^* = 1 \) are shown in Fig. 8. The points indicated in Fig. 7a correspond to the plots indicated as a, b, c and d shown in Fig. 8. It can be seen that the sequence of the flow alternates flow ejections near each one of the two lateral walls of the cavity (see Fig. 8b and d), with an intermediate state that corresponds to a simultaneous symmetric ejections near both lateral walls (see Fig. 8a and c).

The three dimensional instantaneous flow structure for \( Re = 100 \) at \( Ri = 10 \) in the cavity and in the channel is shown in Fig. 9 in terms of isosurfaces of \( k^2 \). This quantity proposed by Jeong and Hussein [21] identifies the location of the vortex cores. Fig. 9a and b correspond to the instantaneous contours shown in Fig. 8b and d, respectively. Both figures depict a pair of counterrotating streamwise vortices connected with the main vortex inside the cavity. It can be seen a secondary vortex near the bottom of the cavity parallel to the edge located at \( x^* = 2 \) and \( y^* = 0 \). The pair of large scale counterrotating streamwise vortices extracts fluid near the lateral walls of the cavity and introduces fluid from the channel to the cavity between them. The instantaneous flow structures are not symmetric with respect to \( z^* = 0.5 \). It can be seen that the different intensity of the vortices is associated with the location of the flow ejection (Fig. 8b and d). The larger vortex induces an important flow ejection near the corresponding lateral wall.

4.3.3. Turbulent flow

The time averaged flow topology at \( Re = 1500 \) and \( Ri = 10 \) is shown in Fig. 10 in terms of an isosurface of \( \lambda_2 \). On average the flow is organized similarly to the periodic flow. A main vortex inside the cavity, a small secondary vortex at the bottom of the cavity and two counter-rotating vortices distributed along the streamwise direction in the channel.

As an example of the flow unsteadiness Fig. 11a shows the time evolution of the spanwise velocity component \( (w^*) \) at the point \( x^* = 2.2, y^* = 1.2 \) and \( z^* = 0.5 \), which is located in the channel near the top trailing edge of the cavity. The corresponding power spectrum (Fig. 11b) shows a more uniform distribution of energy among a wide range of frequencies than that obtained for the periodic flow at \( Re = 100 \) and \( Ri = 10 \) (Fig. 7b). It can be seen a marked peak at \( f^* = 1/15 \) indicating the existence of periodicity in the flow pattern.

To elucidate the flow structures responsible for this periodicity a conditional sampling technique has been used. The time evolution of 300 individual samples of the vertical velocity distribution at \( y^* = 1 \) during a period of 150 non-dimensional time units is used to detect the strong flow ejections near lateral walls of the cavity. These events are classified into four different patterns depending on the location of the flow ejection according to the values of the correlation between the instantaneous distributions and each of the four templates. The templates are built with patterns of positive lobes distributed in different locations: one symmetric template with two lobes near the lateral walls (A), one template with a single lobe near the wall located at \( z^* = 1 \) (B), one symmetric template with a single lobe centered at \( z^* = 0.5 \) (C), and one template with a single lobe near the wall located at \( z^* = 0 \) (D). A similar technique has been applied successfully in previous studies to obtain flow patterns in turbulent forced [22] and natural [23] convection flows.

Results are shown in Fig. 12. The four ensemble averages of the vertical velocity distribution at \( y^* = 1 \) obtained correspond to the 63% of the flow history analyzed, indicating that these are not the only structures present in the flow. It should be noted that even the important periodicity of the flow (Fig. 11b) strong fluctuations are present (Fig. 11a). The individuals used to obtain the ensemble
averages in each case are indicated in Fig. 12 at the top of each graph. The most probable structure is depicted in Fig. 12a, while the other three structures have similar probability. The inspection of the time evolution of the velocity distribution shows the existence of transitions among the four patterns. The numbers of transitions observed are included near the arrows in Fig. 12. It has been found that the main frequency of the flow corresponds to transitions between structures A and B or A and D similarly to the periodic flow case.

To illustrate the effect of the detected flow structures on the channel, Fig. 13 shows the conditionally averaged temperature and cross-stream velocity field in a vertical plane near the trailing edge of the cavity ($x' = 2$). It can be seen that the symmetrical distributed flow ejections (Fig. 12a) produces two hot ascending currents near the lateral walls of the channel. Patterns B and D show an individual stronger ejection of hot fluid near the corresponding lateral wall. Pattern C has more uniformly distributed hot fluid along the spanwise direction.

4.3.4. Heat transfer rate

The average Nusselt number at the bottom wall of the cavity is shown in Fig. 14. It can be seen that the heat transfer rate increases with the Richardson and Reynolds numbers. The present curves for water ($Pr = 7$) are similar to that found previously for air-flow ($Pr = 0.7$) [12]. The large increase of the Nusselt number at $Re = 1500$ between $Ri = 1$ ($Gr = 2.25 \times 10^6$) and $Ri = 10$ ($Gr = 2.25 \times 10^7$) is produced by the dominant effect of the natural convection inside the cavity that improves the flow exchange between the cavity and the channel.

Leong et al. [5] reported a correlation (Eq. (5)), based on numerical simulations, to predict Nusselt number in two dimensional cavities for several aspect ratios ($AR = 1, 2$ and 4).

$$ Nu_w \left( \frac{Re_w}{Gr_w} \right)^{1/4} = 0.0195 + 0.219\left( Gr_w \left( \frac{Re_w}{Gr_w} \right) \right)^{1/4} \tag{5} $$

where $Re_w$ and $Gr_w$ are Reynolds and Grashof numbers based on the cavity width.

Fig. 15 compares the present data with those obtained by the Leong et al. [5]. The gray zone indicates the scatter of the data used to compute the correlation (two dimensional results for several aspect ratios), the squares correspond to the three dimensional air flow simulations [12], and the triangles to the present three dimensional water flow results. The agreement between Eq. (5) and the present results indicates the validity of the correlation to predict heat transfer rates in three dimensional cavities.

5. Conclusion

The flow structure in a cubical open cavity located in the bottom of horizontal channel has been studied in ranges of Reynolds and Richardson numbers $100 \leq Re \leq 1500$ and $0 \leq Ri \leq 10$, respectively. Numerical simulations have been validated with experiments in a water channel. The dependence of the physical properties with temperature was numerically analyzed and for the conditions considered the assumption of constant physical properties is acceptable. Three flow regimes have been found in these ranges of $Re$ and $Ri$, which are steady, periodic and turbulent flow. Steady flow is found at small $Ri$ ($Ri = 0.1$), where the buoyancy is weak for the whole range of Reynolds numbers. At $Ri = 1$ the flow is unsteady for $Re \geq 500$, while the flow became unsteady for the whole range of Reynolds at $Ri = 10$. A periodic flow pattern with a non-dimensional specific period ($t' = 7$) appears at $Re = 100$ and $Ri = 10$. In this case, alternate flow ejections from the cavity to the channel near the lateral walls while the flow enters from the channel to the cavity through the central part.

The turbulent regime is observed for $Re \geq 500$ and $Ri = 1$ and 10. In these conditions the natural convection dominates the heat transfer mechanism within the cavity. A conditional sampling technique was used to analyze the mean flow history of the unsteady flows for $Re = 1500$. The results show four mean flow structures, which are periodically repeatable. The most probable structure consist in two symmetrically distributed flow ejections, from the cavity to the channel, near the lateral walls and a flow inrush at the center of the cavity (structure A). The other three structures have a similar probability of occurrence. One consists in a central flow ejection from the cavity and two symmetrically distributed flow inrushes (structure C). The other two structures (structure B and D) have a single mean lateral flow ejection near a lateral wall. It has been found that the most probable transitions occur between structure A and structure B or D.

The computed Nusselt numbers are in general agreement with a previously reported correlation, valid for two dimensional cavities of different aspect ratios.

Acknowledgments

This study was financially supported by the Spanish Ministerio de Economía y Competitividad under project CTQ2013-46799-C2-1-P.

References