Numerical simulation of incompressible laminar flow in a three-dimensional channel with a cubical open cavity with a bottom wall heated

G Abdelmassih, A Vernet, J Pallares
Rovira i Virgili University, Mechanical Engineering Dept., Av. Paisos Catalans 26, Tarragona 43007, Spain
E-mail: gorg.abdelmassih@urv.cat

Abstract. A three-dimensional numerical simulation study for both laminar steady and unsteady regimes has been carried out for the mixed convective flow over a three-dimensional cubical open cavity. The cavity is heated from below at constant temperature while the other walls are adiabatic. The numerical simulation has been done using a three-dimensional incompressible finite volume flow solver. The effects over the velocity and temperature distribution of the buoyancy forces acting perpendicular to the mainstream flow are studied for Reynolds numbers (Re) between 100 and 1500; Prandtl number (Pr) is set to 0.7 and Richardson number (Ri) between $10^{-3}$ to $10^1$. The phenomenological description of the mixed convection inside and outside the cavity and the combined effects of the natural and forced convection have been obtained. For both high Re and Ri the flow becomes unsteady. The mixed convection effects dominate the flow transport mechanism and push the recirculation zone and the flow further upstream.

1. Introduction
During the last decades, considerable efforts have been devoted to study the heat transfer which occurs in flow over open cavities due to their importance in various engineering applications. This type of geometry can be found in many applications such as, cooling of electronic devices, landing gear bays and solar collectors (see for example [1 and 2]). The early studies of flow past cavities were carried out in the 1960s. These studies were done by Burggraf [3], by Weiss and Florsheim [4], and by Pan and Acrivos [5]. More studies have been carried out to reveal the flow structure and the heat transfer process occurring for both natural and mixed convections for some geometries with different aspect ratios see for instance [6-9]. Manca et al [10], 2003, studied the effect of heat wall position on mixed convection in a channel with an open cavity, these authors found that the maximum temperature values decrease as the Reynolds number and Richardson number increase for all the configurations studied. Yao et al [11], 2005, used finite difference scheme to study numerically incompressible laminar flow over three-dimensional rectangular cavities and they indicated that as Reynolds number increases the flow inside the enclosure becomes unsteady and complex. Stiriba [12] and Stiriba et al [13], studied numerically the flow and heat transfer characteristics for assisting and opposing incompressible laminar flow past an open cavity. The results show that the flow exhibits a three-dimensional structure and is steady for Re=100 with Ri ranging from 0.01 to 10 and Re=1000 with Ri ranging from 0.001 to 1. The forced flow dominates the flow transport mechanism and a large recirculating zone from inside the enclosure which results in heat transfer by conduction. In general
even the two-dimensional version of this configuration has received considerable attention; very few results have been obtained for the three-dimensional case. In this work numerical simulations have been done using a three-dimensional incompressible finite volume flow solver. The effects over the velocity results have been obtained for the three-dimensional case. In this work numerical simulation has been even the two-dimensional version of this configuration has received considerable attention; very few

channel are adiabatic. A cooling incompressible airflow at constant temperature $T_\infty$ heated from below at a constant temperature $T_w$ at $x=0$, convective Euler boundary condition is assumed for 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 $T_w$ and the remaining walls of the cavity and of the channel are adiabatic. A cooling incompressible airflow at constant temperature $T_\infty$ is used, having a Prandtl number (Pr=0.7). Viscous dissipation is negligible. The other fluid properties are assumed constant except for the change in fluid density with temperature, according to the Boussinesq approximation.

2. Physical problem and geometry
The geometry of the channel including the cubical open cavity and the computational domain used in this study are shown in figure 1, where $L$ is the length of the cavity. We considered a cubic cavity (AR=1), $2L$ is the length of the channel behind the cavity in the stream wise direction, and the height of the channel is $L$. The inflow direction was from left to right with a uniform velocity $U_0$ and temperature $T_\infty$, and the non-slip boundary condition is applied for the rest of the boundaries. The cavity is approximated. constant except for the change in fluid density with temperature, according to the Boussinesq approximation.

![Figure 1. Sketch of the computational domain](image)

3. Governing equations and numerical method
The 3DINAMICS finite volume parallel code has been used in this work [12]. The code solves numerically the three-dimensional incompressible Navier-stokes equations (1) to (3) 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 (BiCGstab). Convective and the diffusive terms are approximated using SMART scheme [12] and central differences respectively.

$$
\frac{\partial u_i^*}{\partial x_i} = 0
$$

$$
\frac{\partial u_i^*}{\partial t^*} + \frac{\partial (u_i^* u_j^*)}{\partial x_j^*} = -\frac{\partial p^*}{\partial x_i^*} + \frac{1}{Re} \frac{\partial^2 u_i^*}{\partial x_i^* \partial x_j^*} + \frac{Ri}{Re} T^* \delta_{ij}
$$

$$
\frac{\partial T^*}{\partial t^*} + \frac{\partial (u_i^* T^*)}{\partial x_i^*} = \frac{1}{Re Pr} \frac{\partial^2 T^*}{\partial x_i^* \partial x_j^*}
$$

$$
Nu_l = \frac{\partial T^*}{\partial y^*} |_{y^*=0} , \quad Nu_s = \int_0^1 \int_0^1 Nu_l d_x^* d_y^*
$$

The dimensionless variables are defined as:

$$
x_i^* = \frac{x_i}{L} , \quad t^* = \frac{t}{U_0 L} , \quad u_i^* = \frac{u_i}{U_0} , \quad p^* = \frac{p - \rho_\infty}{\rho_\infty U_0^2} , \quad T^* = \frac{T - T_\infty}{T_H - T_\infty} , \quad Ri = \frac{Gr}{Re^2}
$$

where $U_0$ refers to the inflow velocity components, $p_\infty$ is the reference pressure, $\rho_\infty$ is the 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, $Gr=(\rho_0 \Delta T L^3)/\nu^2$, and Pr is the Prandtl number, $Pr=\nu/\alpha$. Here $\beta, \nu$
and \( \alpha \) are the coefficient of volumetric expansion, kinematic viscosity, and thermal diffusivity, respectively. \( \text{Nu}_l \) is the local Nusselt number and surface average Nusselt number is \( \text{Nu}_S \), are applied to represent heat transfer at the heated wall. The validation of the code can be found in [12 and 13].

Non-uniform grid sizes were used in this work especially close to the walls to capture the rapid changes in the dependent variables. In order to check the grid independence of the solution, three different grids were tested 160×80×40, 200×100×50 and 220×110×60. Finally the grid of 200×100×50 is used in this work because the results with the two other meshes are very similar. The grid has 50 grid points inside the cavity, \( \Delta x_{\min} = \Delta y_{\min} = 0.01L \), \( \Delta x_{\max} = \Delta y_{\max} = 0.0204L \) and \( \Delta z = 0.0204L \).

4. Results and Discussion
The results were obtained to study the effect of the Reynolds number (Re) on the stability of the flow and the flow structure under the effect of Richardson number \( (\text{Ri}=\text{Gr}/\text{Re}^2) \). The Reynolds numbers considered in this study are 100, 500, 1000 and 1500. For each value of the Reynolds number we considered five values of Richardson number \( (\text{Ri}= 0.001, 0.01, 0.1, 1 \text{ and } 10) \).

Stream line and temperature distribution on the vertical symmetry plane of the computational domain are presented to analyze the flow structure and the effect of the mixed convection. The heated wall has a dimensionless temperature equal to 1, and the inlet airflow temperature is equal to 0, while the increment between contours in the figures is equal 0.025. In all cases in this work two phenomena were found. The first is a recirculation cell structure inside the cavity which is similar to the flow structures found in a lid driven cavity see \[6-13\]. The second is the finding that there is no significant change in the cases of \( \text{Ri}=0.001 \) and 0.01 for all Reynolds numbers. The interpretation is that for small \( \text{Ri} \) in the range \( 0.001 \leq \text{Ri} \leq 0.01 \), the effect of the buoyancy is not important.

![Figure 2](image_url)

**Figure 2.** Streamlines and temperature distribution for \( \text{Re}=100 \), with \( \text{Ri}=0.01(a), 0.1(b), 1.0(c) \) and 10.0(d)

Figure 2, draws the results for \( \text{Re}=100 \), for different values of \( \text{Ri} \) in range of 0.01 to 10.0, in which the flow is steady. At \( \text{Ri}=0.01 \) and 0.1, Figures 2a and 2b, the induced flow does not penetrate in to the cavity and the recirculation cell is encapsulated by the external flow. The recirculation center is located near the top of the cavity, as well as, the effect of the buoyancy force is small as shown in the temperature profile. The temperature contours are parallel and clustered closed to the heated wall, indicating that forced convection is dominant. Increasing \( \text{Ri} \) to one, figure 2c, the recirculation center moves down slightly and is located closed to the center of the cavity. The buoyancy force becomes more effective on mixed convection and pushes the flow upstream. At \( \text{Ri}=10 \), figure 2d, the induced
flow penetrates into the cavity and the recirculation center move to the lower half of the cavity. The natural convection is dominant.

At Re≥500, the temperature distribution exceeds the top of the cavity and reaches the outflow region for all values of Ri, as shown in figures 3 to 5. At Re=500 and Ri≤0.1, figures 3a and 3b, the flow is steady and the effect of buoyancy is small. The increase of Richardson number produces an increase of the buoyancy force. As shown in figures 3c and 3d, that buoyancy pushes the flow upstream, moreover there is a small eddy in low-right-corner in the bottom of the cavity in both, and the simulation predicts a fully 3D flow.
At $Re=1000$, the flow is steady only for low values of $Ri$ ($Ri \leq 0.01$). At $Ri \geq 0.01$ the flow becomes unsteady. Consequently the flow in the cavity for $Re \geq 500$ with $Ri \geq 1$ becomes unsteady and fully 3D. Figures 4a and 4b, depicts the cases for $Re=1000$, at low $Ri$ ($Ri=0.01$ and 0.1), the temperature distribution is clustered towards the heated wall and the left vertical wall of the cavity.

![Figure 5. Streamlines and temperature distribution for $Re=1500$, with $Ri=0.01(a)$, 0.1(b), 1.0(c) and 10.0(d)](image)

![Figure 5. Streamlines and temperature distribution for $Re=1500$, with $Ri=0.01(a)$, 0.1(b), 1.0(c) and 10.0(d)](image)

At $Ri \geq 1$ and $Re=1000$, the flow become unsteady. Figures 4c and 4d show the time averaged flow structure in streamlines and the time averaged temperature distribution. As in the case of figure 4c, the time average flow structure depicts that the flow in the cavity is encapsulated, and the
temperature distribution reach to the lower part of the outflow opening. On the contrary the case shown in figure 4d while the induced flow penetrates the cavity and the temperature distribution affects the outflow.

At Re=1500 and Ri=0.01, figure 5a, the time averaged flow structure is similar to the one presented in figure 4a, although the center of the recirculation cell is slightly lower due to the increase of Re. For Re=1500 and Ri≥0.1, figures 5b,5c and 5d, the flow becomes unsteady and a small eddy appears in the low right corner. The flow penetrates in to the cavity as for Ri=10.

Figures 6a and 6b, displays the instantaneous flow structure and the time average flow structure for Re=1500 and Ri=10, respectively. Figure 6c, depicts the instability of calculated velocity u in x direction with time, while figure 6d draws the RMS between the instantaneous and average velocity for velocity u in x direction at the cavity centerline, while maximum RMS is equal 0.26. Figure 6e depicting the instantaneous temperature distribution in z-y plane at x=L/3 and x=2L/3.

The average Nusselt number at the heated wall is calculated. Figure 7, depicts the average Nu with Ri for cases Re=100, 500, 1000 and 1500. Nusselt number increase with Ri for each value of Re. When Ri is low the buoyancy is weak, thus the effect of the natural convection is low in the transport mechanism and the convective effect is dominate. Nu increase strongly at higher values of Ri, and this depicts that natural convection is dominate.

5. Conclusions
Numerical simulation has been carried out for the mixed convective flow over 3D cubical open cavity heated from below. The results shown that the effect of the buoyancy becomes fixed for small Ri in the range 0.001≤Ri≤0.01 for all Re≤1500. The flow becomes unstable and complicated for Re≥500 with Ri≥1. Time average flow structure approach is used to study the unstable flows and calculate Rms. Nusselt number increase with Ri for each value of Re, when Ri is low the buoyancy is weak, convective effect is dominate. At high Ri the Nu become high as the buoyancy force becomes stronger and natural convection dominates which have an effect to push the recirculating cell upstream, which makes the recirculating cell’s center moves slightly to the right part of the cavity.

Acknowledgments
This study was financially supported the Spanish Ministry of Science and Technology and FEDER under project DPI2010-17212.
6. References

[1] Papanicolaou E and Jaluria Y 1992 *J. Fluid Mech.* **239** 489-509
[2] Zdanski P Ortega M and Fico J 2005 *J Heat Trans-T Asme.* **127** 699-712
[3] Burggraf O 1966 *J. Fluid Mech.* **24** 113–51
[4] Weiss R and Florsheim B 1965 *J. Phys. Fluids* **8** 1631–35
[5] Pan F and Acrivos A 1967 *J. Fluid Mech.* **28** pp. 643–55
[6] Papanicolaou E and Jaluria Y 1990 *Numer. Heat Tr. A-Appl.* **18** 427–61
[7] Leong J Brown N and Lai F 2005 *Int. Commun. Heat. Mass.* **32** 583–92
[8] Manza O Nardini S and Vafai K 2006 *Exp. Heat Transfer* **19** 53–68
[9] Papanicolaou E and Jaluria Y 1993 *Numer. Heat Tr. A-Appl.* **23** 463–484
[10] Manca O Nardini S Khanafer K and Vafai K 2003 *Numer. Heat Tr. A-Appl.* **43** 259-282
[11] Yao H Cooper R and Raghunathan S 2004 *J. Fluid Eng.* **126** 919-27
[12] Stiriba Y 2008 *Int. Commun. Heat Mass* **35** 901–07
[13] Stiriba Y Grau F Ferré J and Vernet A 2010 *Int. J. Heat Mass Tran.* **53** 4797–808.