ISSN 1392−1207. MECHANIKA. 2017 Volume 23(4): 545−551

Two-Dimensional numerical simulation of natural convection in a square cavity

R. Benchabi*, A. Lanani**
**LEAP Laboratory, Mechanical Engineering Department, Freres Mentouri University, Constantine, 25000, Algeria, E-mail: rbenchabi@yahoo.fr*
***MAM Laboratory, Mathematics Department, Freres Mentouri University, Constantine, 2500, Algeria, E-mail: alanani1@yahoo.fr*

crossref http://dx.doi.org/10.5755/j01.mech.23.4.14482

1. Introduction

The natural heat transfer convection is a mode in which the movement of the fluid is simply the result of the difference in density between the hot and cold regions of the fluid in the presence of a gravitational field. Natural convection in confined spaces remains a very interesting topic, both for its practical applications in industry, as for the fundamental questions it raises. The natural convection in enclosures has many applications in thermal engineering, such as double glazed windows, solar panels, cooling devices for electronic instruments, cavities filled with gas around the cores of nuclear reactors; …etc. Natural convection in enclosures, also known as internal convection is well developed by Faghri et al. [1]. The need to search in this field motivated us to undertake this investigation. In the present work, we are interested in numerical simulation of laminar natural convection in a square cavity. The heat transfer by natural convection in enclosures heated by heat sources has been studied by many researchers.

The natural convection in a square cavity was numerically analyzed using a control volume approach. The calculations were performed for the laminar and turbulent flow regimes and the $k-\varepsilon$ model was used for modeling turbulence for a range of Rayleigh (*Ra*) numbers up to 10^{10}. Accurate results were obtained according to the independence of the mesh. The solution captures very well the whole flow and heat transfer phenomena, especially near the walls [2].

A numerical study concerning the phenomenon of natural convection to laminar flow in a differentially heated square cavity filled with air is given in [3]. It has established a set of temperature reference solutions, velocities, current lines and the average Nusselt number for values of the Rayleigh number ranging from 10^3 to 10^{16}.

Marcatos and Pericleous have numerically studied the turbulent convection for Rayleigh numbers up 10^{16}. The authors have shown correlations between the Nusselt and Rayleigh numbers [4].

The characteristics of the velocity field and the heat transfer rate for stationary laminar flow have been investigated in a numerical study [5]. The study is performed for a wide range of Rayleigh number in the range 10^3 to 10^{10}. The authors analyzed the flow structures and have found that, when the Rayleigh number increases the flow is confined near adiabatic walls and have proposed correlation between the Nusselt and the Rayleigh number: $Nu = 0.163\times Ra^{0.282}$, if $10^3 \le Ra \le 10^{10}$.

The study of Prasopchingchana et al. [6] is devoted to numerically study the natural air convection in a differentially heated square cavity. The finite volume method was used in order to discretize the partial differential equations of the air flow into the cavity. The inclination angles of the square cavity giving the average Nusselt numbers are 110° for $Ra = 10^3$ and 130° for $\left[\left(Ra = 10\right)\right]^3$ and $Ra = 1\times 10^4$.

Previous work was conducted by [7] to experimentally study a natural convection in a differentially heated square cavity for a number of $Ra = 1.89\times 10^5$. The experience permitted to visualize and describe the flow inside the cavity from thermal and dynamic fields.

A numerical study of natural convection in an enclosed square cavity has been carried out by Mitra. The constrained interpolated profile (CIP) method was used to simulate the considered natural convection [8].

Two-Dimensional natural convection in differentially heated enclosure has been studied [9]. The purpose of this study is to investigate the effect of aspect ratio and inclination angle on flow and heat transfer.

The previous study of the authors [10] concerns the application of a commercial code, CFX-10 to simulate a natural convection of air in a square cavity.

A numerical study of natural convection in partially heated square cavity has been carried out by [11]. The authors used a numerical code (CAVITY) based on the finite difference method.

The objective of our study is to characterize the laminar natural convection in a differentially heated square cavity filled with air. The horizontal walls are subject to a zero heat flow. Also, this study consists in studying the effect of Rayleigh number that is either due to the change of the dimensions of the cavity or to the temperature difference.

2. Physical and mathematical model formulation

2.1. Physical model

The considered geometry is a differentially heated square cavity. This cavity is composed of two active vertical walls, maintained at temperatures T_h and T_c with $T_h > T_c$. For the other walls, they are assumed to be adiabatic. The studied area has a length in the directions (x) and (y). Also, the flow in this chamber is caused by the buoyancy force resulting from the hot wall.

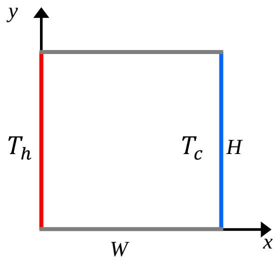


Fig. 1 The studied configuration

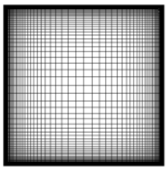


Fig. 2 Generating the used mesh

2.2. Development equation

The equations governing the incompressible two-dimensional steady flow inside the square cavity using the Boussinesq approximations are given by [12-13]:

- Continuity equation:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0. \qquad (1)$$

- x - Momentum equation:

$$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=-\frac{1}{\rho}\frac{\partial p}{\partial x}+\vartheta\left[\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right]. \qquad (2)$$

- y - Momentum equation:

$$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=-\frac{1}{\rho}\frac{\partial p}{\partial y}+\vartheta\left[\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right]-g\beta\left(T-T_0\right). \qquad (3)$$

- Energy equation:

$$u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}=\alpha\left[\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}\right]. \qquad (3)$$

3. Numerical method

The equations governing the flow in the square cavity are discretized by the finite volume method and the SIMPLEC algorithm is adopted to solve the velocity - pressure coupling problem of the air flow. Also, the TDMA (Tri-Diagonal Matrix Algorithm) is used to obtain the approximated solution and the upwind second order scheme is used to discretize the convective terms of the transport equation. In addition, a relaxation technique is applied to control the convergence iterative process which is achieved for a criterion of the maximum absolute values of normalized residuals on all computation domain $\varepsilon<10^{-4}$. In order to obtain stable and precise solution, the effect of the mesh on the solution was performed for different numbers of nodes. The used mesh is given in Fig. 2.

4. Results and discussion

The numerical results are obtained using the FLUENT computational code for an air flow in a differentially heated square cavity. They are presented as curves for the thermal and dynamic fields.

4.1. Effect of the mesh

The influence of the mesh on the numerical results was considered to obtain satisfactory solutions. The grid independence test is performed using successively sized grids 40×40; 60×60; 80×80 and 100×100 in a laminar flow for $Ra=10^5$. Also, the distribution of the horizontal u – velocity in the vertical mid plane $x=W/2$ and vertical v is velocity in the horizontal mid plane $y=H/2$ and temperature are presented on Figs. 3 - 5. It is observed that the curves overlap with each other for all grids. So a grid number of 80×80 is chosen for all computation for Rayleigh number in the range 10^3–10^5; but for $Ra=10^6$, an increased number of nodes was necessary, so a grid number of 100×100 is considered.

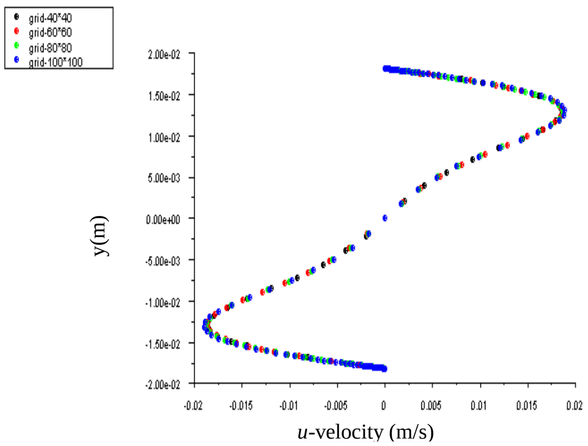


Fig. 3 Horizontal velocity profiles (u) in the vertical mid plane $x=W/2$; $Ra=10^5$

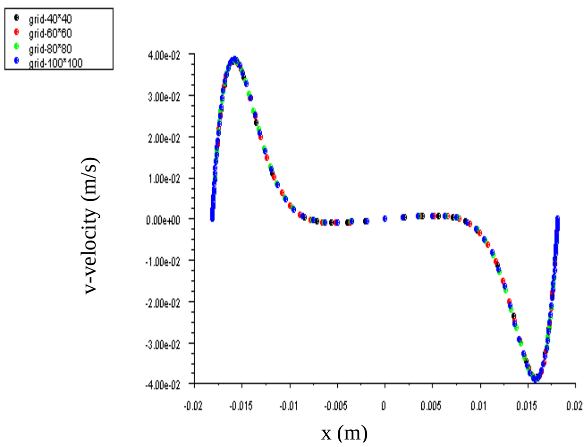


Fig. 4 Vertical velocity profiles (v) in the horizontal mid plane $y=H/2$; $Ra=10^5$

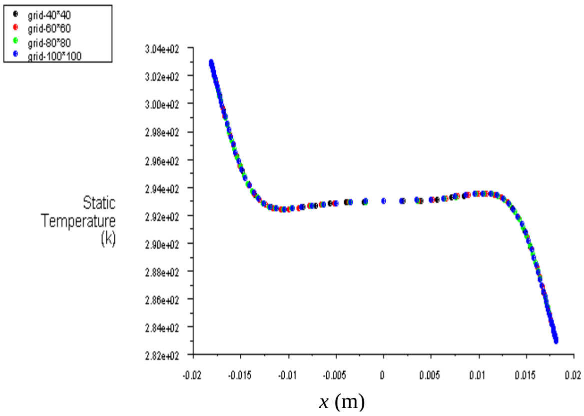


Fig. 5 Temperature profiles in the horizontal mid plane, $x = H/2;\ Ra = 10^5$

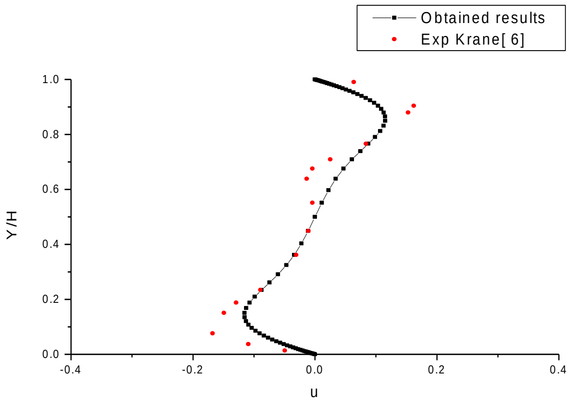


Fig. 7 Horizontal velocity profiles (u) in the vertical mid plane $x = W/2;\ Ra = 1.89 \times 10^5$

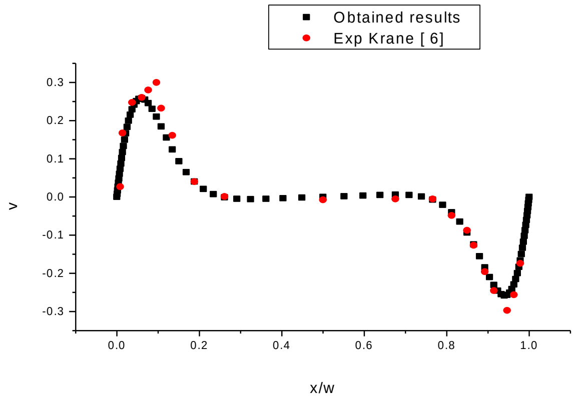


Fig. 8 Vertical velocity profiles (v) in the horizontal mid plane $y = H/2;\ Ra = 1.89 \times 10^5$

4.2. Code validation

The validation of numerical simulation is needed to verify the accuracy of numerical results obtained by the FLUENT CFD code. A comparison of our results with the experimental study of Krane et al. [7] was established. The authors have considered a differentially heated square cavity filled with air for $Ra = 1.89 \times 10^5$.

Figs. 6 and 8 show a comparison for the temperature profile in the horizontal plane $x = H/2$ and the vertical velocity profile in the horizontal plane $x = H/2$ between our results and those of Krane et al. [7]; there is a good agreement, but for the horizontal velocity profile in the vertical plane $x = W/2$, there is slightly higher values for the maxima at the boundary layers.

After this comparison, we notice that the maximal difference between all these results remains less than 3%.

4.3. Results and discussion

The Studied configuration is a square cavity filled with air. The horizontal walls are kept insulated and the left vertical wall of the cavity is kept at a constant temperature T_h, while its right vertical wall is maintained at temperature T_c with $T_h > T_c$. This study was done for different

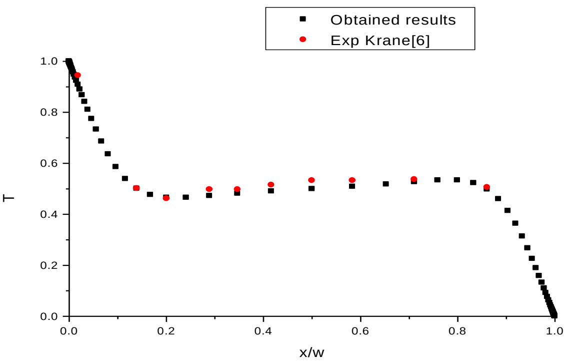


Fig. 6 Temperature profiles in the horizontal mid plane $y = H/2;\ Ra = 1.89 \times 10^5$

4.3.1. Influence of Rayleigh number due to the variation of dimensions

Rayleigh number, to be obtained by changing either the dimensions of the cavity or the temperature difference (ΔT).

Initially, we will study the effect of the Rayleigh number due to the variation of the dimensions on the thermal field, the Nusselt number and the dynamic field.

a- *Thermal field.*

The temperature field inside the cavity for Rayleigh numbers in the range 10^3 – 10^6 is shown in Fig. 9. It is noted that for $Ra = 10^3$, the isotherms of the cold wall are almost straight and parallel to the wall and near the left wall, the isotherms are therefore weakly distorted so the heat is transferred by conduction between hot and cold walls. At horizontal walls, all isotherms are vertical.

The deformation of the isotherms at the top of the cavity is observed from $Ra = 10^4$; so the heat transfer mechanism changes from conduction to convection. It is also noted that the increase in Rayleigh number results in an increase of the isothermal deformation which is more pronounced and that these isotherms become parallel to the horizontal walls. Thermal stratification on the left side of the cavity is observed and thermal boundary layers are becoming thinner than the Rayleigh increases. Natural convection is more significant and the heat convection flow is transferred of the hot wall to the cold wall through the top wall.

Fig. 9 shows that there is a good agreement between the obtained results and those of Barakos et al. [2].

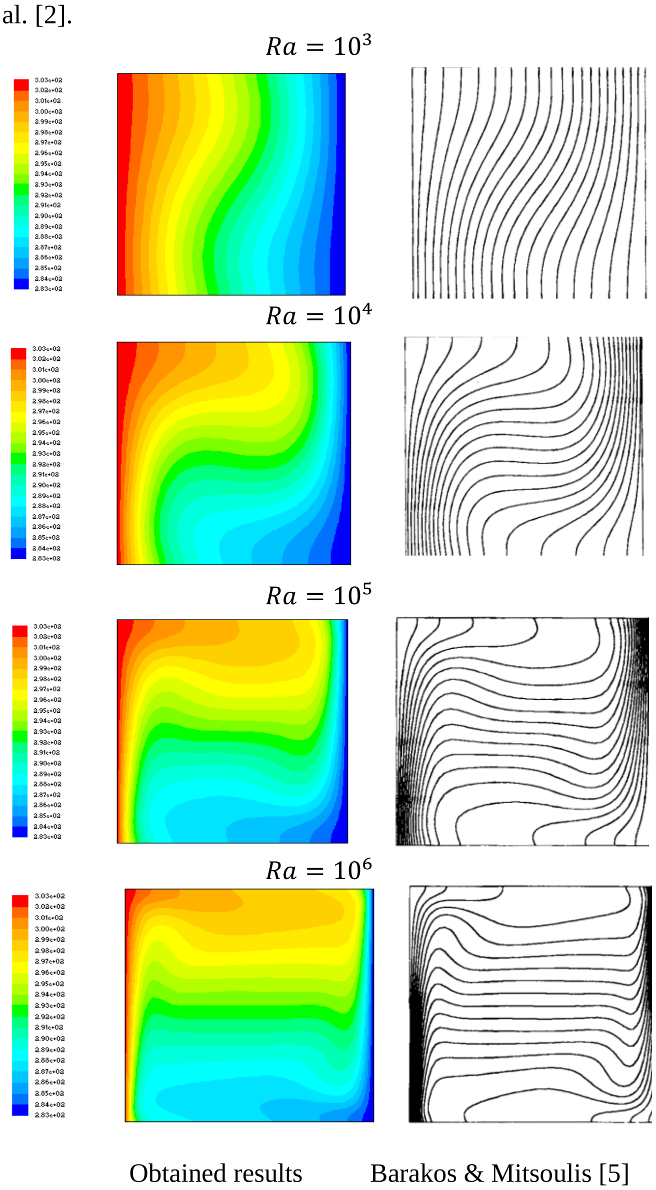


Fig. 9 Isotherms for different Rayleigh values

b- *Dynamic field*.

Fig.10 presents the profiles of the vertical component in the horizontal mid plane $y = H/2$ for different Rayleigh. A significant change in the vertical component is observed with the increase in Rayleigh number; this is due to the importance of the temperature of horizontal gradients which are causing gravitational forces and which leads to a convective motion inside the cavity. Also, there is an increase in velocity from zero to a maximum and decreases until it becomes zero in the middle of the cavity. This is the area where the fluid is in upward and finally becomes zero on the axis, this area is which where the fluid is in downward movement.

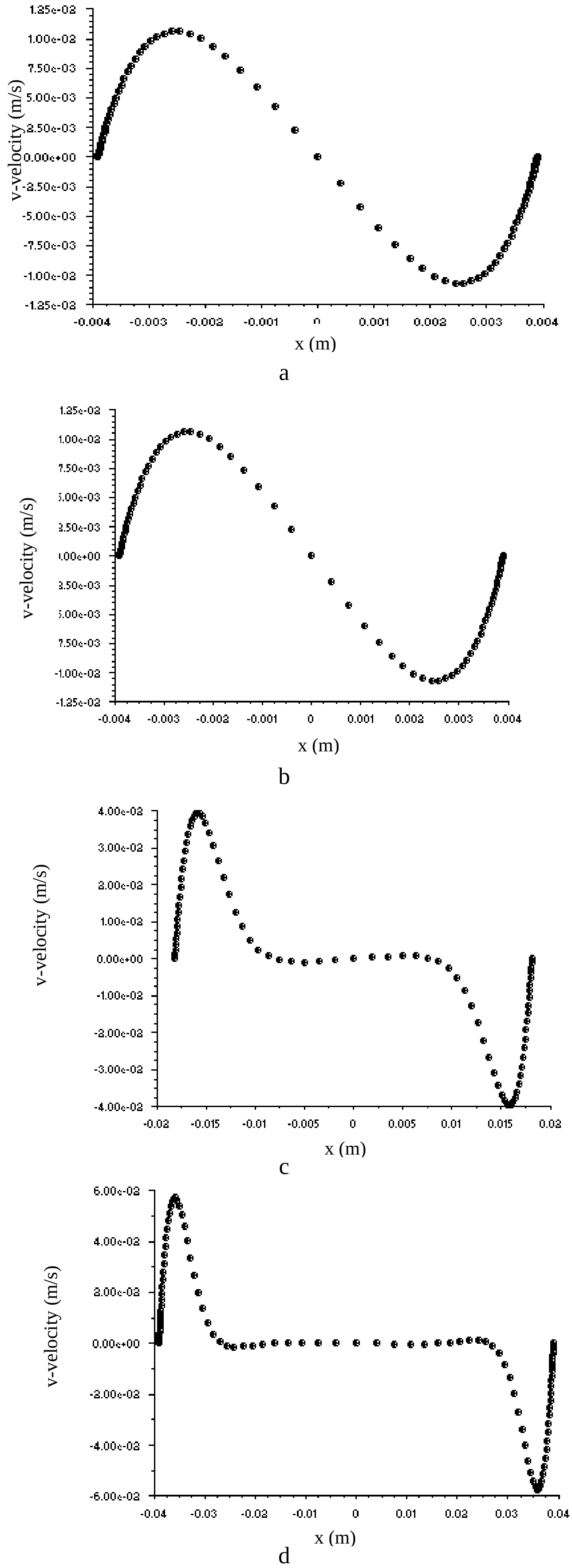


Fig. 10 Vertical velocity profiles (v) in the horizontal mid plane $y = H/2$: a - $Ra = 10^3$; b - $Ra = 10^4$; c - $Ra = 10^5$; d - $Ra = 10^6$

Fig. 11 shows the profiles of the horizontal component in the vertical mid plane $x = W/2$ for different Rayleigh number. A symmetrical profile with respect to the vertical central axis is observed. Also, we see a gradually decreasing velocity near the centre.

The average Nusselt values obtained in this study and compared with the results of the authors [3-5] are presented in Table 1.

Table 1 shows the variation of the average Nusselt according to the Rayleigh number obtained by this study and the comparison between the results of the present simulation and those of other authors [3-5], a good agreement is observed. We also found that increasing the Rayleigh number causes an increase in the Nusselt number.

Table1

Comparison of mean Nusselt numbers

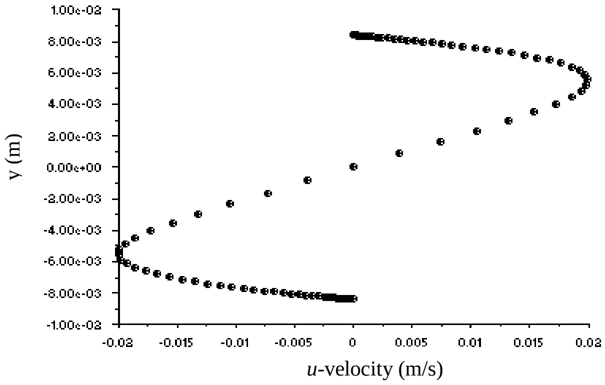
	Obtained results	Markatos and Pericleous	De Vahl Davis	Fusegi et al
Mean Nu numbers	$Ra = 10^3$			
	1.1073	1.108	1.118	1.105
	$Ra = 10^4$			
	2.3057	2.201	2.243	2.302
	$Ra = 10^5$			
	4.7175	4.430	4.519	4.646
	$Ra = 10^6$			
	9.0339	8.754	8.799	9.012

4.3.2. Influence of Rayleigh number due to the temperature difference

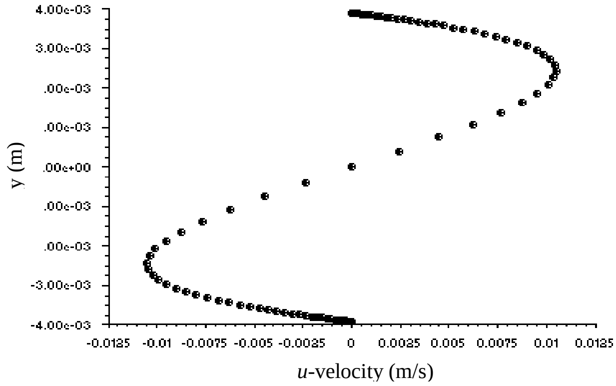
The convective flow is governed by the number of Grashof $(Gr), Ra = Gr.Pr,$ that compares the buoyancy forces to viscous forces and is proportional to the temperature gradient.

The thermal field is represented by the Fig. 12 for different temperature gradients in the range ranging from 5°K - 20°K with a step of 5°K. It is observed that the isotherms changed slightly and increases with increase of the Rayleigh number, $(0.614 \times 10^3 \le Ra \le 2.461 \times 10^3)$. Comparing the vertical velocity profiles in the horizontal mid plane $y = H/2$ for different ΔT is shown in Fig. 13. It is noted that an increase of ΔT (increased *Ra*) provides increased the maximum velocity, this shows that natural convection is more significant by the thermal buoyancy.

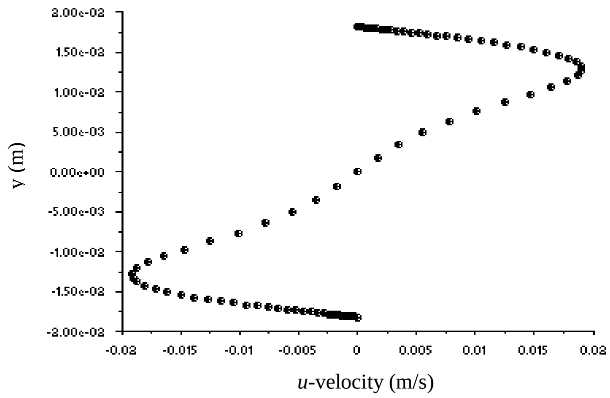
Fig. 14 shows the increase in horizontal velocity profiles in the vertical mid plane $x = W/2$. It is observed that the effect of the convection becomes significant with increasing imposed difference temperature.



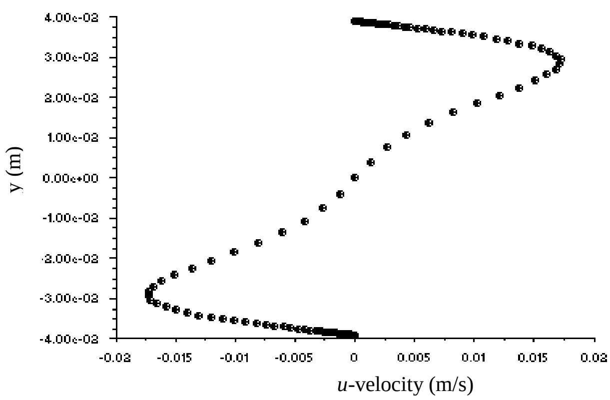
a



b

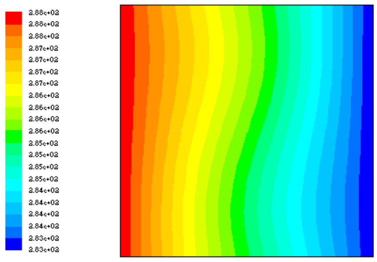


c

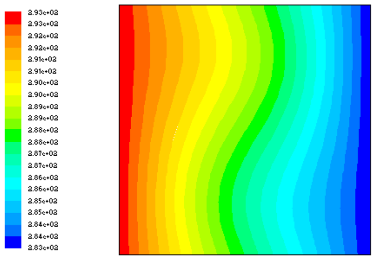


d

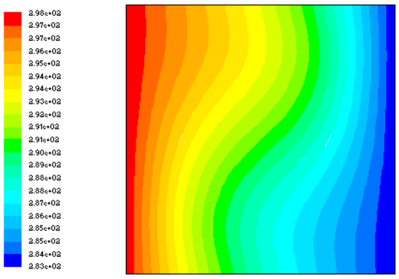
Fig. 11 Horizontal velocity profiles (u) in the vertical mid plane $x = W/2$: a - $Ra = 10^3$; b - $Ra = 10^4$; c - $Ra = 10^5$; d - $Ra = 10^6$



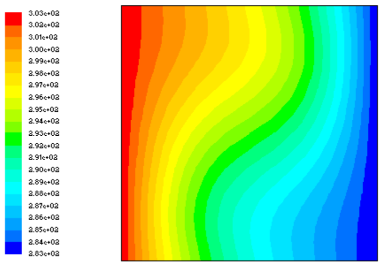
a



b



c



d

Fig. 12 Temperature contours: a - $\Delta T = 5^{o}\mathrm{K}$; ; b - $\Delta T = 10^{o}\mathrm{K}$; c - $\Delta T = 15^{o}\mathrm{K}$; d - $\Delta T = 20^{o}\mathrm{K}$

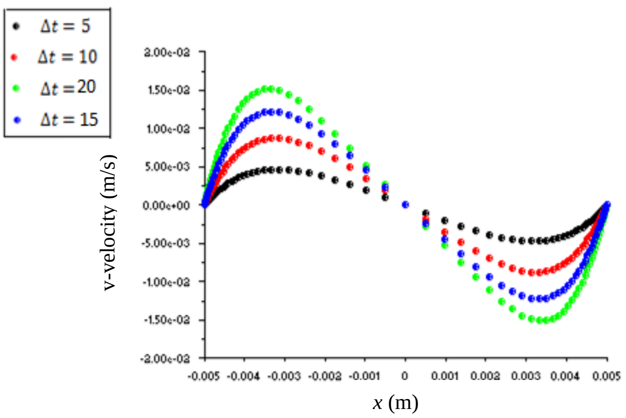


Fig. 13 Vertical velocity profiles (v) in the horizontal mid plane $y = H/2$ for different ΔT

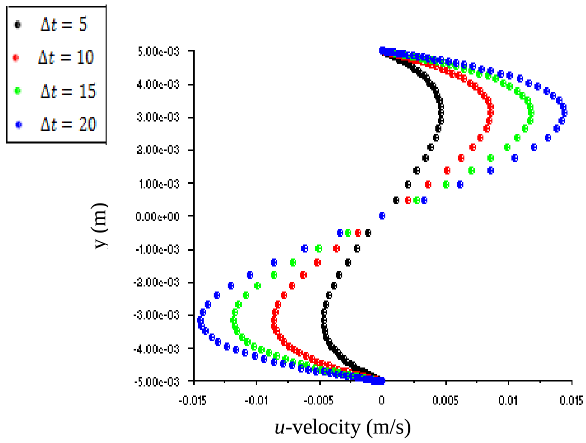


Fig. 14 Horizontal velocity profiles (u) in the vertical mid plane $x = W/2$ for different ΔT

5. Conclusions

In the present work, the numerical study of the laminar natural convection inside a differentially heated square cavity filled with air to horizontal walls kept insulated was developed by the FLUENT CFD code. Numerical calculations were performed on a fine square mesh near the walls. The finite volume method was used for the discretization of the equations governing the flow in natural convection. The numerical procedure was validated by comparing the results obtained by the code with the existing results in the literature. Good agreement was observed between them.

This study was conducted according to the Rayleigh number either due to the change of the dimensions of the cavity or to the temperature difference. We could see through the thermal, dynamic fields and the Nusselt number, the influence of the Rayleigh number on the observed structures, the convective regime and the heat transfer rates. The obtained results of the simulation also show the intensification of the natural convection in the cavity which increases with the increase of the Rayleigh number.

Acknowledgments

The authors thank the reviewers and the executive editor for their careful reading of the manuscript and their many valuable comments and suggestions for the improvement of this article.

References

1. **Faghri, A.; Zhang, Y.; Howell, J.R.** 2010. Advanced Heat and Mass Transfer. Global digital Press, Columbia, USA.
2. **Barakos, G.; Mitsoulis, E.; Assimacopoulos, D.** 1994. Natural convection flow in a square cavity revisited: laminar and turbulent models with wall functions, International Journal of Numerical Methods in fluids 18: 695-719. http://dx.doi.org/10-1002/fld.1650180705.
3. **Davis, De Vahl, G.** 1983. Natural convection of air in a square cavity: A benchmark numerical solution, International Journal of Numerical Methods in fluids 3: 249-264. http://dx.doi.org/10-1002/fld.1650030305.

4. **Markatos, N. C.; Pericleous, K. A.** 1984. Laminar and Turbulent Natural Convection in an enclosed cavity, International Journal of Heat and Mass Transfer 27 (5): 755-771.
http://dx.doi.org/10.1016/0017-9310(84)90145-5.
5. **Fusegi, T.; Hyun, J. M.;Kuwahara, K.** 1991. Transient Three-dimensional Natural Convection in a Differentially Heated Cubical Enclosure, International Journal of Heat and Mass Transfer 34 (6): 1543-1557.
http://dx.doi.org/10-1002/fld.1650130704.
6. **Prasopchingchana, U.; Pirompugd, W.;Laipradit, P.;Boonlong, K.** 2013. *Numerical* Study of Natural Convection of Air in an Inclined Square Enclosure, International Journal of Materials, Mechanics and Manufacturing 1 (2): 131-135.
http://dx.doi.org/10.7763/IJMMM.2013.V1.29.
7. **Krane, R.J.; Lessee, J.** 1983. Some detailed field measurements for a natural convection flow in a vertical square enclosure, Proceedings, 1st ASME-JSME Thermal Engineering Joint Conference, Honolulu Hi., USA 1: 323-329.
8. **Mitra, A.** 2013. Numerical study of natural convection in an enclosed square cavity using constrained interpolated profile (CIP) method. International journal of research in engineering and technology 2(9):1-25.
http://dx.doi.org/10.15623/ijret.2013.0209003.
9. **Chang, B.H.** 2014, Numerical study of flow and heat transfer in differentially heated enclosures, Thermal science 18(2): 451-463.
http://dx.doi.org.102298/TSCI110626007C.
10. **Yoon, C.; Park, J.H.; Hwang, M.; Yang, J.E.** 2006. Simulation of natural convection of air in a square cavity using CFX-10, Transactions of the Korean nuclear society spring meeting Chuncheon, Korea, May 25-26, 2006.
11. **Tabet, S.**; Dellil, A.Z.; Azzi, A. 2016. Natural convection in partially heated square cavity, Mechanika 22 (2): 119-124.
http://dx.doi.org/10.5755/j01.mech.22.2.12215.
12. **Kays, W.; Crawford, M.; Weigand, B.** 2005. Convective Heat and Mass Transfer, 4th ed., Mc Graw - Hill Companies, inc., New York, NY, USA.
13. **Bergman, T.L; Lavine, A.S.; Incropera, F.P.; Dewitt, D.P.** 2011. Fundamentals of Heat and Mass Transfer, 7th edition John Wiley.

R. Benchabi, A. Lanani

TWO-DIMENSIONAL NUMERICAL SIMULATION OF NATURAL CONVECTION IN A SQUARE CAVITY

S u m m a r y

The present work is based on a numerical simulation of natural convection in a steady two-dimensional cavity filled with incompressible air. This convection is subjected to a horizontal temperature gradient and the flow is modeled by differential equations of conservation. Numerical calculations were performed on a mesh using the procedure of finite volume and SIMPLEC algorithm was used to solve the pressure-velocity coupling in the equations governing the flow. The thermal and dynamic fields of the flow in a square cavity were obtained and compared with the experimental results and they are in good concordance. Also, the obtained results show the intensity of the flow induced by the buoyancy force and allowed to see the influence of Rayleigh number due either to the change in size or to the difference temperature on the thermal and dynamic behavior.

Keywords: Heat transfer enhancement, Numerical simulation, Natural convection, Square cavity, Finite volume method.

Received March 29, 2016
Accepted August 04, 2017