International Journal of Mechanics and Energy (IJME)
Vol. 2, Issue 3, 2014, ISSN: 2286-5845

# Effect of the heat flow on the turbulent macrostructure with boundary layer in a cavity differentially heated

*Djedid Taloub[1,]*, Abdelhadi Beghidja[1]*

[1] Laboratory of Renewable Energy and Sustainable Development (LRESD) Constantine1University, Constantine 25000, Algeria
B.P. 25, Ain el hadjel 28310, M'sila, 28000, ALGERIA

**Abstract:** The heat transfer by convection is, so far, a basic principle of many industrial applications. This study led to the analysis of turbulent convection. $Ra > 10^9$ in a rectangular three-dimensional cavity filled with air, the two opposite vertical walls are differentially heated a constant temperature at cold wall and the heat flux at the hot wall, the other walls are hot wall except ceiling wall is cold according to fig. 2. The finite volume method has been used to discretize the equations of flow in turbulent convection. The model of turbulence used is $\kappa$-$\varepsilon$. The results obtained are suitable, because they show that for a number of $Pr = 0.71$ and while varying the heat flow, generating consequently a great influence on the transfer of heat inside the field of study, the numerical results of the heat flux at the hot wall are over predicted. The strong influence of the undulation of the cavity and its orientation is well shown, and the release of the instability due to the interactions of the swirling structures with the boundary layer.

**Key words:** Natural convection, finite volume, parallelepiped, k-epsilon.

## 1. Introduction

Natural convection in parallelepiped cavities with vertical walls and differentially heated constitutes a basis configuration of various industrial systems, and particularly a reference case very simple for the development and validation of numerical simulation of flows natural convection.

The study of the natural convection of fluids in the cavities has been a very large number of both theoretical as experimental work. The interest of this study reside in its involvement in many natural and industrial processes such as cooling of electronic circuits and nuclear reactors, building insulation, metallurgy, crystal growth for the semiconductor industry drivers, etc..

The fluid flow, whether laminar or turbulent, are described by the system of partial differential equations.

Thus, all physical phenomena are governed by the system formed by the equations of continuity, moving amount and energy that must be resolved to know the characteristics of the temperature field and flow field. The main purpose of the numerical simulation is to determine the physical behavior of the system submitted to heat transfer and can be important and instationary. Concerning the system composed of an isothermal turbulent fluid flow and solid structure, problems related to thermal fatigue involve important features, other than a medium of temperature fields, such as frequency spectral and amplitude of temperature fluctuations. These temperature fluctuations depend mainly of investigated physical configuration (which can be very complex), the regime of flow (i.e., Reynolds number), the Prandtl number

---

*** Corresponding author:** Djedid TALOUB
E-mail: djtaloub@yahoo.fr

and the nature of the thermal coupling between the fluid and the solid. This work is based on many previous studies both experimental and numerical, including Mergui [1-2], Salat [3-4] Lankhorst [5], Tian [6] Ampofo [7] examined numerically flows turbulent natural convection in a parallelepiped cavity. Many numerical investigations have also been conducted with cavities of modest size or $Ra_h < 10^9$ [8-9]. Beyond ($Ra_H > 10^{10}$).

We consider here a configuration of natural convection in a parallelepiped cavity fluid, where a heat flow is imposed according to fig. 2.

There are examples of applications such a configuration in the solar systems, or the description of the air flow within a room.

## 2. Description of the problem

The physical model considered is shown schematically in Fig. 1. is a three-dimensional parallelepiped cavity of large size (H = 2.46 m high, L = 0.385 m width, P = 0.72 m deep).

The fluid within the cavity is considered incompressible and Newtonian. Because studying heat transfer appears only by natural convection, radiation is not taken into consideration in the numerical model. The dissipation of heat by viscous friction is neglected. The Boussinesq approximation is considered. The fluid within the cavity is air.

For the boundary conditions of the walls, the two vertical walls are considered imposed. Q is the heat flux of the hot wall and the $T_f$ of the cold wall. Other surfaces delimiting the cavity are considered isothermal according to fig. 2.

The gravitational acceleration g is taken into consideration. The study is performed for different heat flux between the hot and cold walls.

We study this configuration. The flow in the cavity is turbulent Ra > $10^9$.

$T_c$: hot wall, $T_f$: cold wall, $L_D$: right side wall,

$L_G$: left side wall, $P_d$: ceiling wall and $P_c$: floor wall.

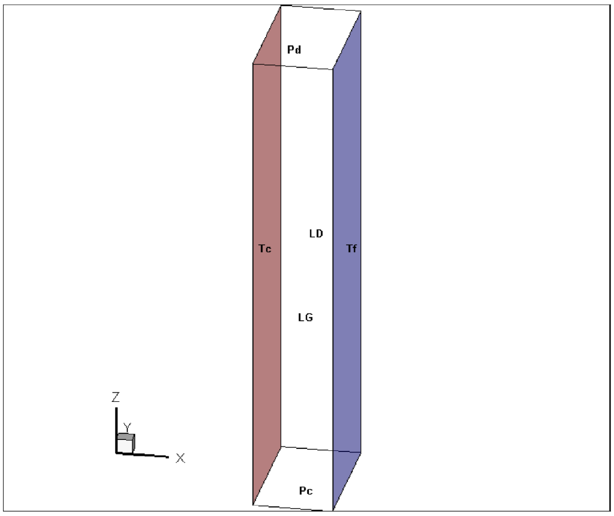

**Fig. 1. Schematic of the cavity.**.

| | | | |
|---|---|---|---|
| | | $P_d$ | |
| $T_f$ | $L_G$ | $T_c$ | $L_D$ |
| | | $P_c$ | |

| Configuration | | | |
|---|---|---|---|
| | | F | |
| F | C | Q | C |
| | | C | |

**Fig. 2. Configuration studied and their mode of representation.**

## 3. Mathematical formulation

*Model RSM*

The transport equations of the constraints of Reynolds in Fluent are written

$$\underbrace{\frac{\partial}{\partial x_k}\left(\rho U_K \overline{u_i u_j}\right)}_{1} = -\frac{\partial}{\partial x_k}\underbrace{\left[\rho\overline{u_i u_j u_k} + \overline{p\left(\delta_{kj} u_i + \delta_{ik} u_j\right)}\right]}_{2}$$

$$+\underbrace{\frac{\partial}{\partial x_k}\left[\mu\frac{\partial}{\partial x_k}\left(\overline{u_i u_j}\right)\right]}_{3} - \left[\underbrace{\overline{u_i u_k}\frac{\partial U_j}{\partial x_k} + \overline{u_j u_k}\frac{\partial U_i}{\partial x_k}}_{4}\right] \qquad (1)$$

$$+ \quad \underbrace{\overline{p\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)}}_{5} - 2\mu\underbrace{\overline{\frac{\partial u_i}{\partial x_k}\frac{\partial u_j}{\partial x_k}}}_{6}$$

1: $C_{ij} \equiv$ convection; 2; is $D_{T,ij} \equiv$ diffusion turbulente

3: $D_{L,ij} \equiv$ diffusion molecular

4: $P_{ij} \equiv$ term of production of constrained

5: $\varphi_{ij}\equiv$ pressure rate of deformation;

6: $\varepsilon_{ij}\equiv$ dissipation

$$D_{T,ij}=C_S\frac{\partial}{\partial x_k}\left(\rho k\frac{\overline{u_k u_1}}{\varepsilon}\frac{\overline{u_i u_j}}{\partial x_1}\right) \tag{2}$$

$$D_{T,ij}=\frac{\partial}{\partial x_k}\left(\frac{\mu_t}{\sigma_k}\frac{\partial\overline{u_i u_j}}{\partial xk}\right) \tag{3}$$

The equation of pressure - of deformation is written:

$$\phi_{ij}=\phi_{ij,1}+\phi_{ij,2}+\phi_{ij,\omega} \tag{4}$$

$$\phi_{ij,1}=-C_1\rho\frac{\varepsilon}{k}\left[\overline{u_i u_j}-\frac{2}{3}\delta_{ij}k\right]\quad\text{Slow term} \tag{5}$$

$$\phi_{ij,2}=-C_2\left[\left(P_{ij}+C_{ij}\right)-\frac{2}{3}\delta_{ij}\left(P-C\right)\right]\quad\text{Fast term} \tag{6}$$

$$\begin{aligned}\phi_{ij,\omega}=&C_1''\frac{\varepsilon}{k}\left(\overline{u_k u_m}n_k n_m\delta_{ij}-\frac{3}{2}\overline{u_i u_k}n_j n_k-\frac{3}{2}\overline{u_j u_k}n_i n_k\right)\frac{k^{3/2}}{C_1\varepsilon d}\\&+C_2''\left(\phi_{km,2}n_k n_m\delta_{ij}-\frac{3}{2}\phi_{ik,2}n_j n_k-\frac{3}{2}\phi_{jk,2}n_i n_k\right)\frac{k^{3/2}}{C_1\varepsilon d}\end{aligned} \tag{7}$$

$$C_1=\frac{C_\mu^{3/4}}{k} \tag{8}$$

The expression (7) represents a term to deaden the fluctuation and it is included by defect in model RSM.

**Table 1. Values of the constants of model RSM.**

| $C_1$ | $C_2$ | P | C | $C_1'$ | $C_2'$ | $C_\mu$ | k |
|---|---|---|---|---|---|---|---|
| 1.8 | 0.6 | 1.68 | 0.7179 | 0.5 | 0.3 | 0.09 | 0.4187 |

## 4. Procedure of resolution

The geometry is presented on fig 1, to see the effect of the heat flow on the tourbillonner structure in a cavity, and one based in this study on the model of turbulence k-ε. The inside running out fluid is air, with a density of 1.244575 kg/m$^3$ and a dynamic viscosity of 1.7714e$^{-5}$ kg/ms. 872176 The number of nodes.

### 4.1 Boundary conditions

$$\begin{aligned}&x=y=z=0\Rightarrow u=v=w=0\\&x=L;y=D;z=H\Rightarrow u=v=w=0\end{aligned} \tag{9}$$

The walls $T_f$, $L_D$, $L_G$, $P_d$, and $P_c$ have a temperature imposed constant, on the other hand the $T_c$ wall had a constant heat flow according to fig. 2.

### 4.2 Choice of the diagram of discretization

**Table 2. Diagram of discretization.**

| | |
|---|---|
| Pressure | Body force Weighted |
| Coupling velocity- pressure | Simple |
| Momentum | Second order Upwind |
| Turbulent kinetic energy | Second order Upwind |
| Rate of dissipation | Second order Upwind |
| Constraints of Reynolds | Second order Upwind |

### 4.3 Validation

In order to verify the accuracy of the numerical results obtained in the present work, a validation of the numerical code was made taking into account some numerical studies available in the literature. Ampofo results [7] obtained in the case of a square cavity containing air, were used to test our simulation by Fluent. The comparison was made by considering the Rayleigh number 1.58 x 10$^9$. Comparison of velocity profiles along the plane V medium shows excellent agreement.

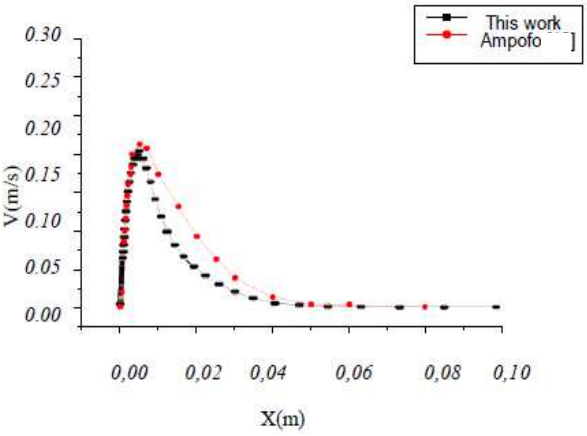

Fig. 3. Comparison of velocity profile along the y = 0.375m.

## 5. Results and discussion

We will treat from now on the effect of the heat flow applied to the hot wall. In Fig. 5 one represents the thermal field. The recovered heat of the hot wall is transported by convection towards the cold wall and upwards, it is what explains the relatively high temperature near to the ceiling. We notice an increase in the average temperature of the cavity with the increase in the value of the heat flow. The wall is not isothermal. Fig. 6 represents the field of vector velocity (plane U-W). The flow is a priori mono cellular with the ascending fluid along the hot wall and to descend along the cold wall. However one notes a disturbance as well as the appearance of several zones of recirculation indicating a greater complexity and a higher degree of turbulence. It appears a flow characteristic for a zone of dominant recirculation. In Fig. 7 represent the field vertical velocity; we clearly observe the rise of the fluid near to the hot wall and descend it near to the cold wall. In the profile vertical velocity one can see the dynamic boundary layer near to the wall hot and cold, the thickness of this layer is equal approximately 0,1m. And in Fig. 8 the fields velocity horizontal, we observe near to the floor, the fluid flow of the cold wall towards the hot wall, on the other hand near to the floor it is the reverse. The flow and mono cellular, as it appears in Fig. 9. In addition it comprises first dominant tourbillonner structure, located at a height varying from configuration to another. One can notice a clear increase in the maximum value the speed related to the increase of the heat flow. For Fig. 10 we represent the profile of temperature for different from value of the heat flow, One can notice the thermal boundary layer near to the hot and cold wall with thermal thickness of boundary layer approximately 0,03m. One can clearly see the stratification of the mass of air with profiles temperature along the high and the cavity we notice that an increase in the temperature of the hot wall to the increase in the heat flow. In end Fig. 10 indicates the profiles of temperature for different position height we notice that the temperature of the hot wall increases with the direction. Profile temperature show also the progressive diffusion of heat and the increase the thermal thickness of boundary layer with the height. One realizes clearly that starting from certain a height, the profile of temperature is almost invariant with the height with the immediate vicinity of the wall.

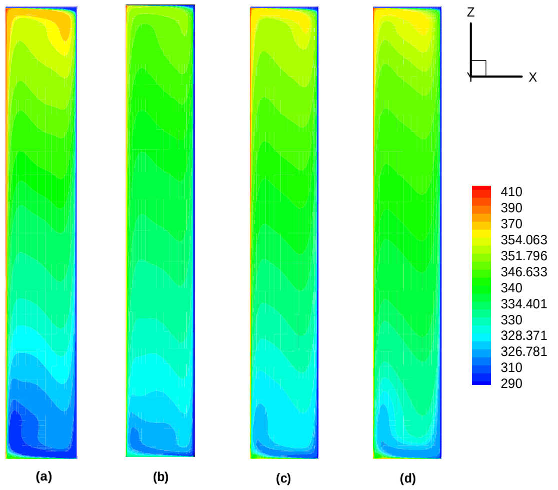

**Fig. 4. Thermal fields for the configuration in the plane y=D/2=0.36m where: (a) Q = 50w/m$^2$, (b) Q = 100w/m$^2$, (c) Q = 150w/m$^2$, (d) Q = 2000w/m$^2$.**

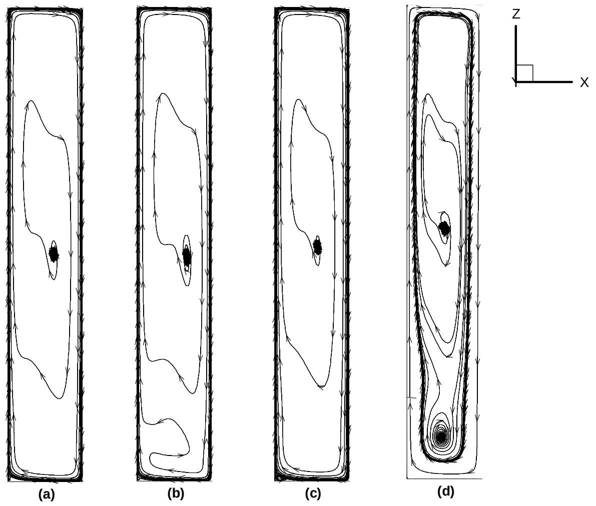

**Fig. 5. Velocity vectors (U, W) for configuration in the plane y = D/2 = 0.36 m where: (a) Q = 50w/m$^2$, (b) Q = 100w/m$^2$, (c) Q = 150w/m$^2$, (d) Q = 2000w/m$^2$.**

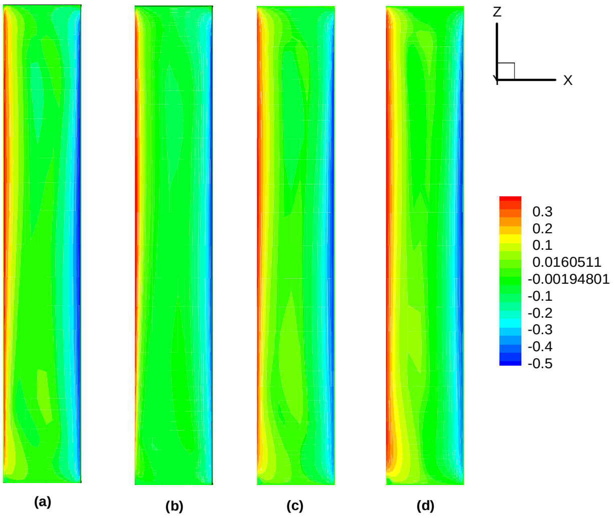

**Fig. 6. Fields velocity vertical for configuration in the plane. y=D/2=0.36m where: (a) Q = 50w/m$^2$, (b) Q = 100w/m$^2$, (c) Q = 150w/m$^2$, (d) Q = 2000w/m$^2$.**

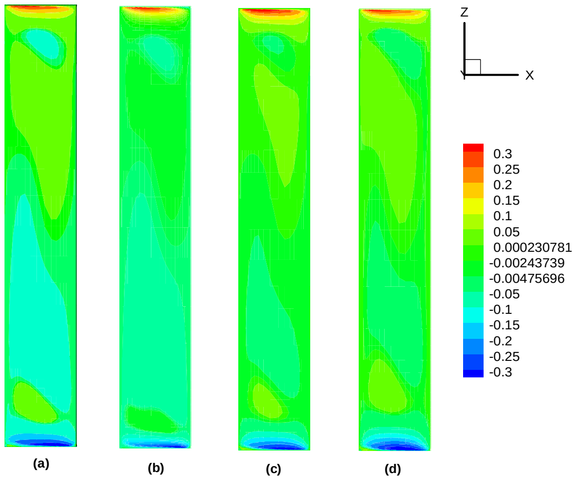

**Fig. 7. Fields velocity horizontal for the configuration in the plan. y=D/2=0.36m where: (a) Q = 50w/m$^2$, (b) Q = 100w/m$^2$, (c) Q = 150w/m$^2$, (d) Q = 2000w/m$^2$.**

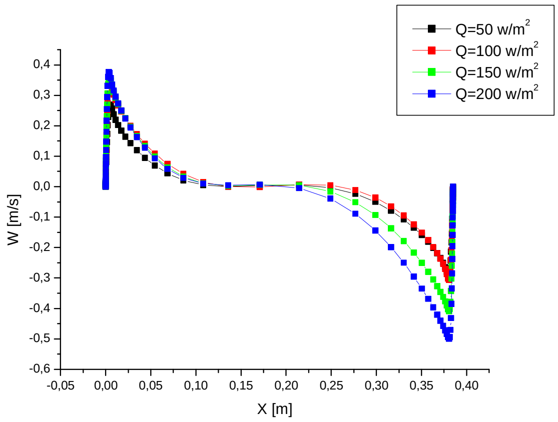

**Fig. 8. Profile vertical velocity for different heat flow in the configuration with the plane y=D/2=0.36m and z=H/2=1.23m.**

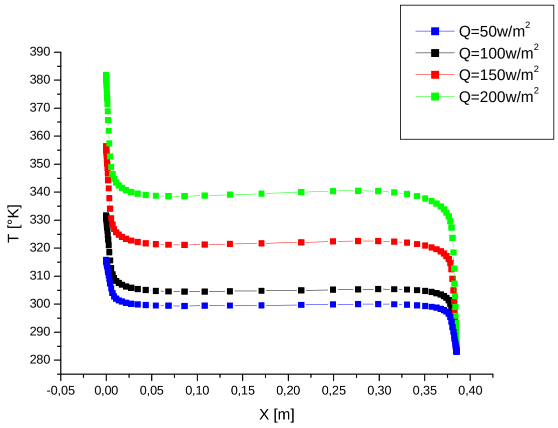

**Fig. 9. Profile of temperature for different heat flow in the with the plane y=D/2=0.36m and z=H/2=1.23m.**

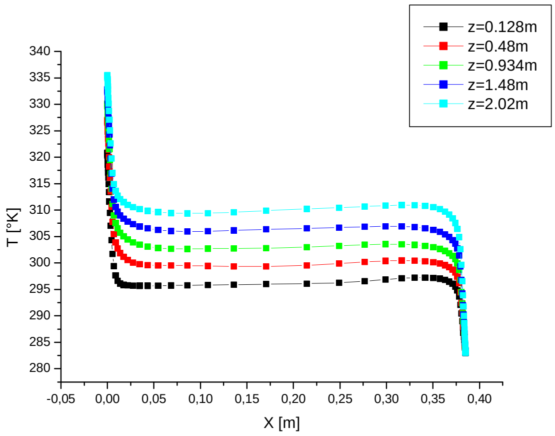

**Fig. 10. Profile of temperature for configuration and Q = 100 W/m$^2$ in the plan y=D/2=0.36m and z varied.**

## 6. Conclusion

In conclusion, we can confirm through this study the direct influence of side walls on the nature of the air flow in cavity. This influence results in the stratification of the mass of air which changes according to the configurations i.e. boundary conditions (heat flow imposed on the side wall). Of other by, we highlighted the effect of heat flow (since the characteristics of the fluid remain unchanged), on the phenomena of convection and consequently on the zone of release of turbulent instability.

Taking into account the coherence of our results with those established numerically and in experiments, by others we succeeded in validating code Fluent the computer for the configuration studied with standard model *k*-*ε*, and finished volumes. Consequently, we can consider consequently, to deal this problem in the future with the introduction of calculation code conceived on the method of simulation (Large Eddy Simulation: LES), While basing itself on the results obtained in this work.

## References

[1] S. Mergui, F. Penot, Analyzes speeds and temperature of the air in natural convection in a cavity differentially heated with Ra=1.69 10$^9$, Int. J. Heat Mass Transfer 40 (14) (1991) 3427-3441.

[2] S. Mergui, Experimental characterization of the flows of air of natural and mixed convection in a closed cavity, Thesis Doctorate, University of Poitiers1993.

[3] J. Salat, F. Penot, Experimental approach of the natural convection of turbulent transition in a cubic cavity differentially heated VI Inter-University conference Frensh-Quebec Thermics of the Systems, (26-28 mai 2003, Quebec).

[4] J. Salat, Contribution to the study of the three-dimensional natural convection in cavity differentially heated. Thesis of Doctorate of the Poitiers University , 2004.

[5] A.M. Lankhorst, D. Angirasa, C.J. Hoogendoorn, LDV measurements of buoyancy induced flows in an enclosure at high Rayleigh numbers, Exp. Thermal and Fluid Science 6 (1993) 74-79.

[6] Y.S. Tian, T.G. Karayiannis, Low turbulence natural convection in an air filled square cavity: Part 1. The Thermal and Fluid Flow Fields, IJHMT 43 (2000) 849-866.

[7] F. Ampofo, T.G. Karayiannis, Experimental benchmark data for turbulent natural convection in an air filled square cavity, IJHMT 46 (19) (2003) 3551-3572.

[8] G. De Vahl Davis, Laminar natural convection in an enclosed rectangular cavity, In. J. Heat and Mass Transfer 11 (1968) 675-1693.

[9] S. Xin, P. Le Quéré, Direct numerical simulation of two dimensional chaotic natural convection in a differentially heated cavity of aspect ratio 4, J. Fluid Mech. 304 (1995) 87-118.

[10] A.Dafa'Alla, P. Betts, Experimental stady of turbulent natural convection in a tall air cavity, Exp. Heat Transfer 9 (1996) 165-194.

[11] A.Beghidja, Dj. Taloub, Effect of Rayleigh number on the turbulent structures with boundary layer in a differentially heated, In. J. Mechanics and Energy (IJME) Vol. 1, Issue 2, 2013, ISSN: 2286-5845

[12] A.Beghidja, M. Bouafia, J.J. Vullierme, F. Yguel, A. Zammit, Modeling of the exchanges convective on the front verticals of cavities to great numbers of Rayleigh, General review of thermics Tom 34 (405) (1995) 515-526.