MIXED CONVECTION IN A THREE-DIMENSIONAL VENTILATED CAVITY DIFFERENTIALLY HEATED FROM THE SIDES/T08

H. DOGHMI[1], B. ABOURIDA[1], L. BELARCHE[1], M. SANNAD[1], M. OUZAOUIT[1]

[1]LMPEE, Ecole Nationale des Sciences Appliquées d'Agadir, BP 1136 Agadir, Maroc.
E-mails: hicham.gte@gmail.com; b.abourida@uiz.ac.ma; mohamedsannad@gmail.com; belarchelahoucine@gmail.com

Abstract

Mixed convection in a three-dimensional ventilated cavity with differentially heated side walls has been analyzed numerically by the finite volume method. The results are presented in terms of streamlines, temperature distribution and average Nusselt number; for two configurations (CH and HC) and different combinations of thermal controlling parameters namely, Reynolds and Richardson numbers. The obtained results show that injecting air through the cold wall is more effective in heat removal (CH configuration), compared to the case of where the inlet opening is placed on the heated wall (HC configuration).

Keywords: *Three-dimensional, ventilated cavity, mixed convection*

1. Introduction

In the last few decades, mixed convection in ventilated cavities has received a sustained attention due to its wide applicability in several engineering problems, such as solar energy collectors, cooling of electronic devices, thermal design of buildings, furnace design, air conditioning, air-cooling, nuclear reactors others… Several studies have been carried out in the area of mixed convection Hence, Omri and Nasrallah [1] studied numerically the mixed convection in an air-cooled cavity for two different positions of the inlet and outlet openings on the sidewalls of the cavity. The results presented show that the clearance of the heat is more effective when the air is injected from the bottom of the hot wall. Moraga and López[2] compared numerically the mixed convection in a two- and three-dimensional air-cooled cavity. They found that a 3-D model must be used to capture the fluid mechanics for Ri =10 when 10 < Re < 250, and to calculate the global Nusselt number when Re= 500 for Ri < 1. However, a literature review showed the existence of a limited number of three-dimensional studies and explains the choice of our present physical model. Hence, the purpose of our study is to compare numerically the effects of conjugated Richardson numbers and Reynolds numbers on the thermal transport and fluid flow phenomena in the considered tri-dimensional enclosures.

2. Physical problem and governing equations

The studied geometries are presented in Fig. 1a-1b. They consist of three-dimensional ventilated cavities with inlet and outlet openings on their side walls. The inlet opening has a rectangular cross section of relative height B=h/L and is located on the top of the left vertical wall allowing the air flow to get in at a uniform velocity Ui and ambient temperature T_c. Underneath the opening, the remaining part of the wall is heated with a uniform temperature T_h. An outlet opening, with a relative height B=h/H, is placed at the bottom of the right vertical wall, which is maintained at cold temperature T_c. This first configuration will be named HC configuration, while the CH configuration has the right and left vertical walls maintained at cold temperature T_c and heated temperature T_h respectively.

A channel with the same cross section is placed at the opening to extend the cavity and makes possible the use of developed boundary conditions for the fluid flow and heat transfer at the outlet. Many tests has been conducted in order to check the influence of the channel length on the outlet boundary conditions. Finally, a length of L has been adopted as it gives a compromise between the results accuracy and the computational time. The other four walls of the cavity are maintained adiabatic.

a)

b)

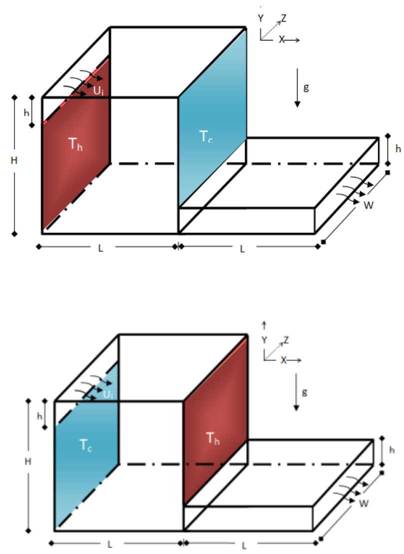

Figure 1: Studied configurations and coordinates
a) HC configuration b) CH configuration

The cooling flow is air with a Prandtl number Pr of 0.71 and is assumed to be laminar and incompressible. Viscous dissipation is negligibly small and all other fluid properties are assumed constant except the fluid density in the buoyancy terms according to the Boussinesq approximation. Under the above assumptions, the continuity, momentum and energy equations for a three-dimensional laminar incompressible fluid can be expressed in dimensionless form as follows:

$$\frac{\partial U}{\partial X}+\frac{\partial V}{\partial Y}+\frac{\partial W}{\partial Z}=0 \qquad (1)$$

$$U.\frac{\partial U}{\partial X}+V.\frac{\partial U}{\partial Y}+W.\frac{\partial U}{\partial Z}=-\frac{\partial P}{\partial X}+\frac{1}{Re}\left(\frac{\partial^2 U}{\partial X^2}+\frac{\partial^2 U}{\partial Y^2}+\frac{\partial^2 U}{\partial Z^2}\right) \qquad (2)$$

$$U.\frac{\partial V}{\partial X}+V.\frac{\partial V}{\partial Y}+W.\frac{\partial V}{\partial Z}=-\frac{\partial P}{\partial Y}+\frac{1}{Re}\left(\frac{\partial^2 V}{\partial X^2}+\frac{\partial^2 V}{\partial Y^2}+\frac{\partial^2 V}{\partial Z^2}\right)+\frac{Gr}{Re^2}\theta \qquad (3)$$

$$U.\frac{\partial W}{\partial X}+V.\frac{\partial W}{\partial Y}+W.\frac{\partial W}{\partial Z}=-\frac{\partial P}{\partial Z}+\frac{1}{Re}\left(\frac{\partial^2 W}{\partial X^2}+\frac{\partial^2 W}{\partial Y^2}+\frac{\partial^2 W}{\partial Z^2}\right) \qquad (4)$$

$$U.\frac{\partial \theta}{\partial X}+V.\frac{\partial \theta}{\partial Y}+W.\frac{\partial \theta}{\partial Z}=\frac{1}{RePr}\left(\frac{\partial^2 \theta}{\partial X^2}+\frac{\partial^2 \theta}{\partial Y^2}+\frac{\partial^2 \theta}{\partial Z^2}\right) \qquad (5)$$

Where p_0 and ρ_0 are respectively the reference pressure and density. T_h is the temperature at the heated surface, T_c the temperature at the cooled surface, p is the pressure and (U,V,W) are the velocity components.

In the above equations, the parameters Pr, Gr and Re denote the Prandtl number, Grashof number and the Reynold number, respectively. These parameters are defined as:

$$Re=\frac{L.u_i}{\nu}, Gr=\frac{g\beta L^3(T_h-T_c)}{\nu^2}, Ri=\frac{Gr}{Re^2}, Pr=\frac{\nu}{\alpha} \qquad (6)$$

Where β, ν and α are the coefficient of thermal expansion, kinematic viscosity and thermal diffusivity, respectively.

The boundary conditions, associated to the problem are:

$U=V=W=0$ on the rigid walls of the enclosure;

$U=1, V=W=0, \theta_c=0$ at the inlet;

$\theta_h=1$ on heat wall; $\theta_c=0$ on the cold wall;

$\frac{\partial \theta}{\partial n}=0$ on other vertical and horizontal walls (n is the normal direction to the considered wall);

$\frac{\partial U}{\partial X}=0, \frac{\partial \theta}{\partial X}=0, V=W=0$ at the outlet.

The average Nusselt characterizing the heat transfer at the heated walls excluding the width of the openings is defined by:

$$Nu_h=\iint Nu(Y,Z)dYdZ \qquad (7)$$

3. Numerical method

The incompressible Navier-Stokes and energy equations are discretized by the finite volume method developed by Patankar [3] adopting the power law scheme for the convective terms. The SIMPLEC algorithm is used to couple momentum and continuity equations. The discretized equations are iteratively solved using an Alternating Direction Implicit (ADI) scheme. The system of algebraic equations is then solved iteratively by means of the Thomas algorithm. Convergence of the numerical code is established according to the criterion:

$$\sum_{i,j,k=1}^{imax,jmax,kmax}\frac{\left|\phi_{i,j,k}^{n+1}-\phi_{i,j,k}^{n}\right|}{\left|\phi_{i,j,k}^{n}\right|}\leq 10^{-4} \qquad (8)$$

Where ϕ represents a dependent variable U, V, W, T, and P while i, j, and k indicate the grid positions, n represents the iteration number

4. Results and discussion

The aim of the study is to compare the convective heat transfer between two different configurations. The computed thermal and flow fields are analyzed in terms of temperature distribution, flow structures and average Nusselt at the heated wall for different Ri and Re values.

A 3D view of the temperature distribution within the cavity is presented in Fig 2 for the two considered configurations. The isotherms are also affected by the choice of Reynolds and Richardson numbers for both configurations.

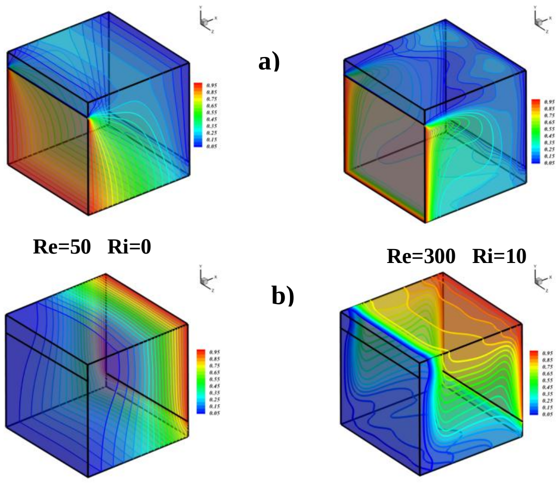

Figure 2:. 3-D isotherm lines at different values of Re and Ri for a) HC configuration b) CH configuration

4.1 Streamlines and isotherms

Fig 3 shows streamlines and isotherms plots at Z=0.5 illustrating the effect of Re and Ri on the dynamical and thermal fields for HC configuration. For the lower Re (Re=50) and Ri (Ri<1), the inflow is not very powerful, and the open lines pattern conserves the same shape until the outlet. The higher temperature regions are concentrated near the heated wall and the isotherms are parallel showing a dominant conduction heat transfer. By increasing Re (Re=300), a big cell clockwise rotating appears inside the cavity under the open lines while a small clockwise is positioned in the upper corner of the cavity, due to the dominance of inertia forces. By increasing Ri (Ri>1), the thermal buoyancy effects become more significant and consequently the convective cell increases. This intensification of the flow improved the heat exchange between the active walls of the cavity and the incoming flow characterized by the distortion of the isotherms.

Fig. 4 shows the dynamic and thermal fields for the second configuration CH for different Ri and Re values. We can notice the same behavior of convective heat transfer as discussed previously for the HC configuration, in the case of low Ri and Re number. By increasing Re from 50 to 300 at Ri=0, the incoming accelerated cold air, which is at the same temperature as the cold wall, is moved toward the heated wall of the cavity due to the vigorous actions of inertia forces.

As the Richardson number increases to Ri = 1, for Re=50 a change in the flow structure and the isotherms is then observed. The streamlines show the existence of a clockwise rotating cell above the open lines and the isotherms are linear from the heated wall to the cold wall, due to the balance between inertia and buoyancy forces.

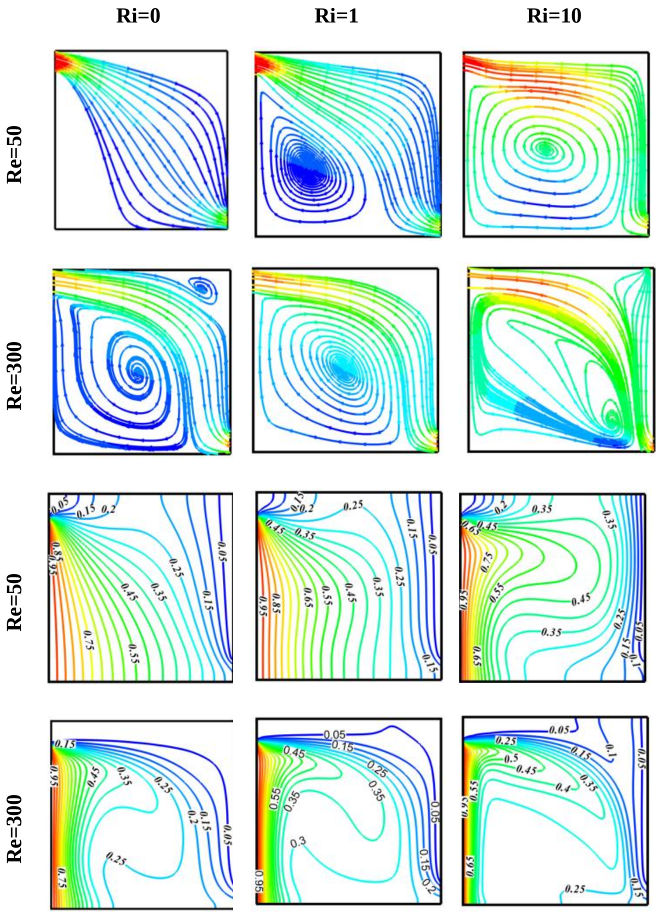

Figure 3: streamlines (top) and Isotherms (bottom) at Z=0.5 for HC configuration

By increasing Ri to 10, the resulting flow is divided into two streams: one flowing toward the lower region parallel to the cold wall then in the direction of the outlet and the other simply going around in the central region of the cavity, above the forced flow and closer to the hot wall. A distortion of the isotherms is observed from the heated wall towards the cold wall. The streamlines corresponding to Re=300 and Ri =1, show the presence of two closed convective cells separated by the open lines of the forced flow. However for Ri=10, a big convective cell occupying almost the whole domain above the open lines is observed and results from the intensification of the flow. The isotherms are non-linear because of the existence of stronger buoyancy forces as shown for Ri=10.

4.2 Nusselt number

In order to analyze the thermal performance for the two configurations, we compare in Table 1, the average Nusselt number at the hot wall for different value of Re and Ri. The average Nusselt number increases with increasing Re and Ri numbers for each of the considered configurations.

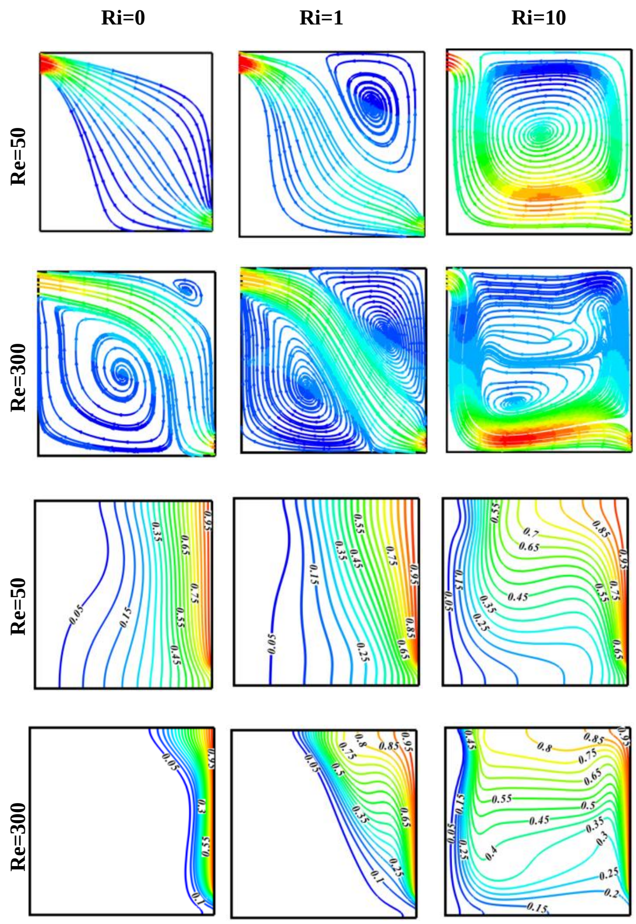

Figure 4: streamlines (top) Isotherm lines (bottom) at Z=0.5 for CH configuration

As we can see that the average Nusselt number is significantly higher for CH configuration at Re=300. This can be explained by the enhancement of thermal interaction between the natural convection flow and the incoming forced flow.

	Richardsons					
	Ri=0		**Ri=1**		**Ri=10**	
Reynolds	**HC**	**CH**	**HC**	**CH**	**HC**	**CH**
Re=50	3,238	4,886	3,278	4,553	4,092	6,048
Re=300	4,621	17,375	5,908	12,679	8,834	18,935

Table 1. Global Nusselt number at the hot wall for HC and CH configurations at different values of Re and Ri

5. CONCLUSION

A numerical study has been carried out in the case of laminar three-dimensional mixed convection flow in a two configurations with different openings positions. The results show that injecting air through the cold wall is more effective in heat removal (CH configuration), due to the impinging effect of the cold air jet on the opposite hot wall.

Références

[1] A.Omri & S. B. Nasrallah, *Control volume finite element numerical simulation of mixed convection in an air-cooled cavity*, Numerical Heat Transfer Part A: Applications (1999), 36(6), 615-637.

[2] M oraga, N. O., & Lopez, S. E. (2004). Numerical simulation of three-dimensional mixed convection in an air-cooled cavity. Numerical Heat Transfer, Part A: Applications, 45(8), 811-824

[3] S. PATANKAR, *Numerical heat transfer and fluid flow*, CRC press, 1980.