5eme Conférence International des Énergies Renouvelables CIER 2017
Proceeding of Engineering and Technology –PET
Vol.27 pp.81-85

CFD modeling of heat transfer in a double-skin with secondary ventilation flow on unheated wall

Mohamed BOURAOUI[1], A.ABIDI-Saad[2],Souad HADJADJ[2], Sarra FATNASSI[3,4],M. Salah. ROUABAH[1], Catalin. POPA[3], Guillaume. POLIDORI[3]

[1]*Laboratoire de Génie Climatique, Département de Génie Climatique, Université Frères Mentouri – Constantine 1 Constantine, Algeria.*

[2] *Faculté des Hydrocarbures et Énergies renouvelables et Sciences de la terre et de l'univers, Université Kasdi Merbah, Ouargla 30000, Algéria.*

[3] *GRESPI/Thermomécanique, UFR sciences Exactes ET Naturelle, Campus du Moulin de la Housse-BP 1039 51687 Reims, France.*

[4]*Laboratory of Energizing and Thermal and Mass Transfers, University of Tunis El Manar, Tunis, Tunisia*

Corresponding author: dr-mohb@hotmail.fr

Abstract— A numerical study of the flow in a double skin flow with secondary ventilation has been conducted numerically, in order to understand the basic mechanisms of the free convection in an open channel asymmetrically heated with uniform heat flux density (510 W/m2). The vertical channel corresponds to a double skin façade, which was immersed in a tank filled with water. The tank corresponds to the environment which allows us to overcome pressure conditions at the inlet and the outlet of the channel. The use of water allows neglecting radiation effect. The mass conservation equations of momentum and energy are solved using the finite volume method and numerical simulations are performed using Ansys Fluent CFD (Computational Fluid Dynamics) software. Our interest, in this work, is the study of the influence of an opening on the unheated wall, which represents secondary ventilation on the heat transfer.

Keywords—Natural convection; CFD; Asymmetric heating; PV Panels; Vertical channel; Double-skin façade.

I. INTRODUCTION (*HEADING 1*)

Among the big energy consumers is the building sector. Buildings are responsible for 40% of energy consumption and 36% of CO_2 emissions in the EU [1]. Thus, it is crucial and necessary to develop a friendly solution by a new technic or improving the existent technics to enhance the energy efficiency in this sector and protect the environment. Due to the large surfaces offered by building envelope, the implementation of double skin façades technology with/without integration of PV panels (Photovoltaic panels), BIPV (Building Integrating Photovoltaic panels) is among the best ways that reduce energy consumption and CO_2 emissions and increase sound protection, daylighting, natural ventilation conditions. Furthermore, the design of building with DSF(Double-skin façade) offers the option to exploit the solar energy for hybrid purposes too (PV/T), and to reduce the risk of overheating PV devices. Thus, a well comprehension of the phenomenon occurring within such systems is essential for the enhancement and consequently the development of new efficient design of such systems.

In this work, we are interested in the study of laminar flow of natural convection in a vertical channel heated asymmetrically. The benefit of this study is that the plane channel is representative of many problems such as chimney, Trombe wall, solar collector or the double-skin façade. Also, note that the flow in an asymmetrically heated vertical plane channel at constant flux may be considered as a prototype of the semi-confined natural convection flows.

In [2], Experimentally observed two dynamic regimes for the flow in a vertical channel. The appearance of these regimes depends on the number of modified Rayleigh Ra^*; Indeed, for a relatively low Rayleigh number the author found a fully developed regime in which all the fluid entering from below exits through the top of the channel with a profile similar to Poiseuille. However, for high Rayleigh number, a boundary layer is observed near the heated wall; the fluid simultaneously enters from below and exits through the top of the channel via a reversed flow along the unheated wall. The formatter will need to create these components, incorporating the applicable criteria that follow. [3] Highlighted also in a numerical study for a channel heated symmetrically or asymmetrically and calculating the Nusselt number as a function of Rayleigh number, two regimes: the first is developed at low Rayleigh number and the second is of boundary layer type at high Rayleigh number. [4] Have also interested in the intensification of heat transfer for natural convection flows in a vertical plane channel by studying the influence of the opening on the unheated wall. In this configuration, it has been shown that the average Nusselt number on the heated wall is almost independent of the location and size of the opening.

[5] They have observed an upward flow of boundary layer type near the heated wall, coming from the bottom of the channel. The latter is accompanied by a downward flow

ISSN 2356-5608

developing on the opposite unheated side coming from the top of the channel. The dynamic structure of the flow has been demonstrated using a visualization technique by laser tomography using discrete tracer by [6]. These authors focused specifically on the influence of the aspect ratio (A/b) and Rayleigh number on the flow structure. Indeed, their results show that the reversed flow takes different proportions of the channel width for different modified Rayleigh numbers. This finding is verified by the comparative study between simulations and experimental results in [7]

Concerning the study of the radiation contribution in a vertical channel, [8] have shown that the heat radiation causes the disappearance of the recirculation in the upper part of the channel. Finally, [9] a recent study in a vertical plane channel asymmetrically heated, realized experimentally by laser tomography for only the very first moments of the implementation, allowed to observe simultaneously the appearance of the dynamic boundary layer and a reversed flow coming from the top of the channel.

This study focus specifically on the influence of the size of an opening, which is a secondary ventilation, on heat transfer. The vertical plane channel was immersed in a tank filled with water corresponding to the domain of study; to overcome the thermal radiation effect and pressure boundary conditions in the inlet and the outlet of the channel. In fact, these latter can make the study complicated, in the case of natural convection, if the channel is only the domain of study. The paper is organized as follows:

In section (II) we describe the channel configuration, then the section (III) shows the governing equations, and the physical model, the numerical method applied in this work is presented in section (IV), and finally section (V) summarizing the results.

II. Channel description

The channel consists of two parallel planar vertical walls of 376 mm high separated by a gap b (b = 36 mm). One wall is composed of a heated central part (height A = 188 mm) surrounding by two unheated parts (length A/2) while opposite wall remains entirely unheated (Fig. 1). In addition, for a better control of the flow conditions at the entrance, a quarter circle (R = 36mm) is added at the bottom of each wall. The channel is immerged in a vertical tank (500 x 500 x 1000 mm^3) made of 20 mm thick Plexiglas® plates and filled with water. The second wall is an adiabatic wall with an opening of size equals, respectively to b/3, 2b/3, b and 4b/3, and located at the bottom of the heated area.

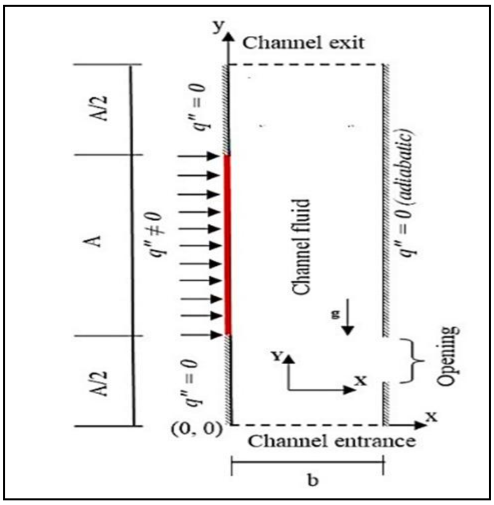


Fig. 1. Channel geometry

III. Physical model

Natural convection is governed by the continuity equation (1), the momentum equations (2, 3) and the conservation equation of energy (4).

$$(\partial\rho u/\partial x)+(\partial\rho u/\partial y) = 0 \quad (1)$$

$$[\partial u/\partial t + u(\partial\rho u/\partial x)+v(\partial\rho u/\partial y)] = - \partial p^*/\partial x +(\mu\, \partial^2 u/\partial x^2)+ (\mu\, \partial^2 u/\partial y^2)+[\partial u/\partial x\, \partial\mu/\partial x + \partial v/\partial x\, \partial\mu/\partial y] \quad (2)$$

$$[\partial v/\partial t + u(\partial\rho v/\partial x)+v(\partial\rho v/\partial y)] = - \partial p^*/\partial y +(\mu\, \partial^2 v/\partial x^2)+ (\mu\, \partial^2 v/\partial y^2)+[\partial u/\partial y\, \partial\mu/\partial x + \partial v/\partial y\, \partial\mu/\partial y] \quad (3)$$

$$\rho c_p[\partial T/\partial t + u(\partial T/\partial x)+v(\partial T/\partial y)] = \kappa(\partial^2 T/\partial x^2)+ \kappa(\partial^2 T/\partial y^2) \quad (4)$$

To simplify the model that describes the natural convection in the vertical channel, some simplifying assumptions have been used. In this study, it is assumed that the flow is laminar, two-dimensional and steady. The fluid is assumed Newtonian, incompressible and physical properties of the fluid depend only on the temperature as described on [9] and [10]. The simulations were conducted for three modified Rayleigh number (Ra*) between 2.25×10^6 and 6.75×10^6. The modified Rayleigh number is based on the flux density (q") and the width of the channel (b) and takes into account the aspect ratio of the heated portion (A/b) through Equation (5).

$$Ra^* = [\rho g \beta q'' b^4/\kappa \nu^2] Pr \quad (5)$$

IV. Numerical method

Transport equations i.e., (1) – (4) of mass, momentum and energy are solved numerically using the finite volume method [11]. This method is based on the spatial integration of transport equations on control volumes. In the solver configuration the pressure-based solver was used which can solve sequentially transport equations. The coupling between velocity and pressure is achieved with the Coupled algorithm that solves the equations of continuity and momentum simultaneously, and gives an advantage to treat flows with a strong interdependence between dynamic and thermal fields. Central-Differenced Schemes and Second-Order Upwind Schemes are used for the spatial discretization for diffusive terms, and convective terms respectively. The time advance uses the Second-Order Implicit formulation. The convergence criteria were based on the absolute residuals resulting from the integration of the conservation equations over finite control volumes. For all simulations performed in this study, converged solutions were achieved after a residuals decrease larger than 10^{-4} for all the governing equations. 2D numerical simulations are performed with ANSYS Fluent® CFD commercial software.

V. Results

Before beginning any studies on the dynamic behavior of the flows, it is necessary to check that the numerical model developed is reliable. To perform this step, numerical results are compared with experimental data [5]. A comparison of the streamlines between experimentation and numerical simulation is made for the same geometrical and thermal configuration at a modified Rayleigh number of 4.5×10^6 (q" = 510 W/m2) at t = 30 min which seems to be a sufficient time to reach the steady-state flow inside the channel.

In this comparative study, one may observe that the dynamical flow structures are similar; namely composed by a boundary layer upward flow near the heated wall and a recirculation zone at the channel outlet. One can observe that both numerical and experimental recirculation sizes are very close. This validates our numerical method (Fig.2).

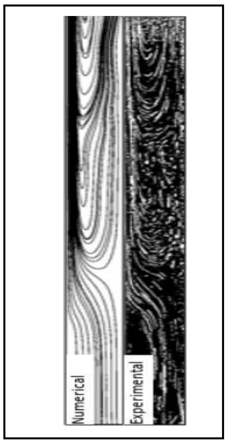


Fig. 2. Model validation between experimental and simulation at Ra* = 4.5×10^6 .

A. The influence of the opening size on temperature field

Figure (3) shows the temperature field within the channel. It can be seen that the presence and position of an opening on the unheated wall, in a vertical plane channel asymmetrically heated, has no effect on the thermal boundary layer, i.e., thermal boundary layer is the motor of the flow. The presence of the opening in the bottom of the channel increases the heat transfer more than the other positions.

B. The influence of the opening sizes on temperature at different heights of the channel

The results allow to observe the effect of the opening sizes (b/3, 2b/3, b and 4b/3) on the temperature variations at different heights of the channel at the entrance of the channel, the entrance of the heated zone, the middle of the heated zone , the exit of the heated zone, and the exit of the channel. Indeed, it can be seen in Figure (4a-d) that the temperature variations is similar for the various opening sizes. In addition, the temperature variation increases in the vicinity of the heated

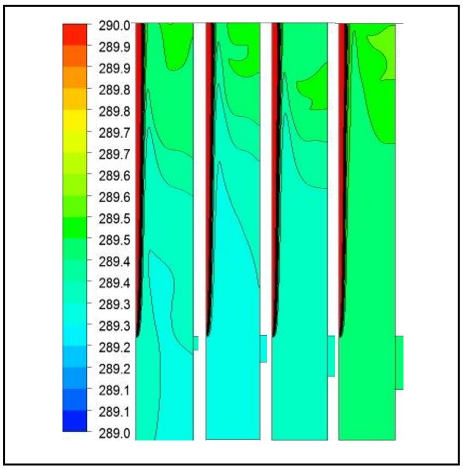
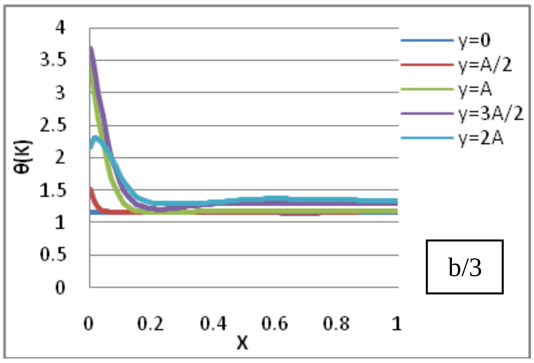
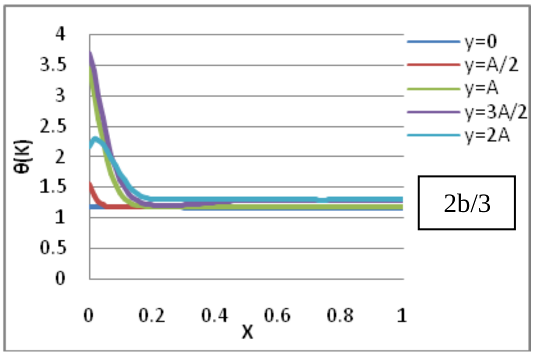
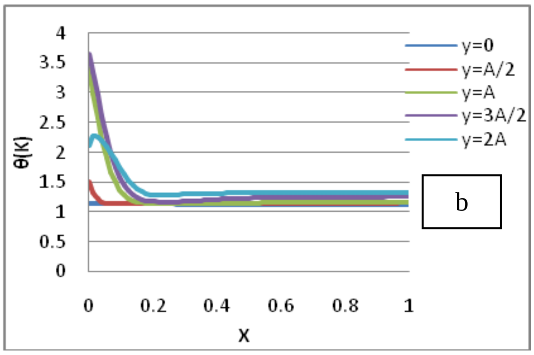


Fig. 3. Isotherm patterns in the channel for various opening sizes located at the bottom of the channel at Ra* = 4.5×10^6 .

wall, and decreases far from this one. Therefore, the opening size has no effect on the evolution shape of the temperature within the channel at the different studied heights.







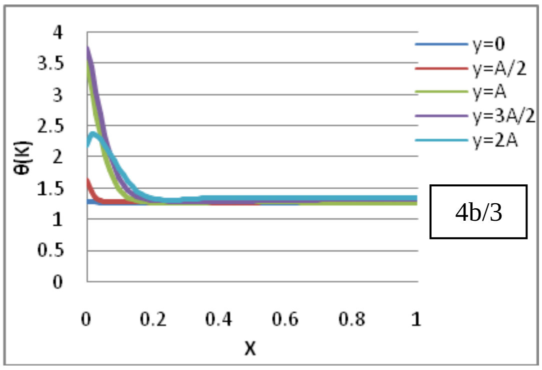


Fig. 4. Temperature variations at different heights of the channel for various opening sizes (b/3, 2b/3, b and 4b/3) located at the bottom of the channel at Ra* = 4.5×10^6 .

C. The influence of the opening sizes on Nusselt number for the various opening sizes at the heated wall.

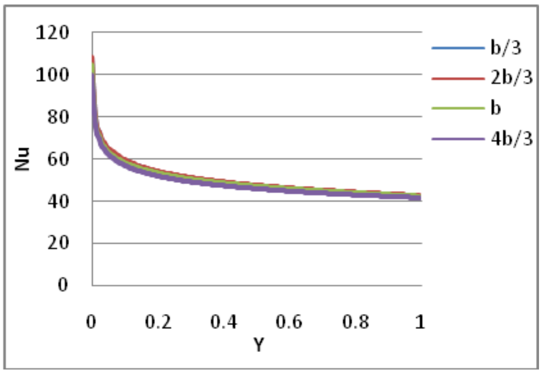


Fig. 5. Nusselt number along the heated wall for various opening sizes located at the bottom of the channel at Ra* = 4.5×10^6

From Figure (5), it can be seen that increasing the opening size at the bottom of the channel has no effect on Nusselt number along the heated wall, Also the presence of the opening within the channel has no effect on the thermal boundary layer. This finding is also verified in [4].

VI. Conclusion

The effect of an opening (secondary ventilation) on heat transfer in an asymmetrically heated channel has been numerically analyzed for a modified Rayleigh number equal to Ra* = 4.5×10^6, channel aspect ratio equals to 5.2 and for four sizes of the opening at the bottom of the channel . Overall, the thermal fields are strongly affected by the opening size. According to our experiment, we may draw the following remarks:

1- A large opening at the bottom of the channel affect the thermal field within the channel.
2- Increasing the opening sizes at the bottom in an asymmetrically heated channel has no effect on the thermal boundary layer and temperature variations at the different studied heights.
3- The variation of the Nusselt number along the heated wall is neglected with opening sizes.

NOMENCLATURE

b	*channel wall spacing (m)*
C_p	specific heat capacity (J/kg.*K)*
g	acceleration of gravity (m/s^2)
k	thermal conductivity (W/m.K)
P^*	driving pressure (Pa)
Pr	Prandtl number
Nu	Nusselt number
q''	heat flux density (W/m^2)
Ra	Rayleigh number
Ra^*	modified Rayleigh number
t	time (s)
T	temperature (K)
u,v	velocity (m/s)
x,y	coordinate (m)

Greek Symbols

β	volume expansion coefficient (1/K)
q"	heat flux density (W/m^2)
ρ	density (kg/m^3)
ν	kinematic viscosity (m^2/s)
θ	*Temperature difference (K)*

References

[1] W. Elenbaas, 1942, “Heat dissipation of parallel plates by free convection,” Physica 9(9), 1-23.

[2] W. Aung, L.S. Fletcher, V. Sernas, 1972, “Developing Laminar Free Convection Between Vertical Flat Plates with Asymmetric Heating,” International Journal Of Heat and Mass Transfer, 15, 2293–2308.

[3] L.F. Azevedo, and, E.M. Sparrow, 1986, “Natural Convection in a Vertical Channel Vented to the Ambient Through an Aperture in the Channel Wall,” International Journal of Heat and Mass Transfer, 29:819–830.

[4] B.W. Weeb, D.P. Hill, 1989, “High Rayleigh number laminar natural convection in an asymmetrical heated vertical channel”, ASME J. Heat transfer, 111, 649-656.

[5] D. Ospir., C. Popa, C. Chereches, G. Polidori, and S. Fohanno, 2012 "Flow Visualization of Natural Convection in a Vertical Channel with Asymmetric Heating," International Communications in Heat and Mass Transfer, 486–493.

[6] C. Popa, D. Ospir, S. Fohanno, and C. Chereches, 2012, "Numerical Simulation of Dynamics Aspects of Natural Convection Flow in a Double-skin Façade," Energy and Buildings, 229–233.

[7] R. Li, M. Bousetta, E. Chenier, and G. Lauriat, 2013, "Effect of surface radiation on natural convective flows and onset of flow reversal in asymmetrically heated channels," International Journal of thermal sciences, 9-27.

[8] G. Polidori, S. Fatnassi, R. Ben Maad, S. Fohanno, and, F. Beaumont, 2014, "Early-stage dynamics in the onset of free-convective reversal flow in an open-ended channel asymmetrically heated," International Journal of thermal sciences, 40-46,

[9] Abidi-Saad, A., Kadja, M., Popa, C., and Polidori, G., 2017, "Effect ofAdiabatic Square Ribs on Natural Convection in an Asymmetrically Heated Channel," *Heat Mass Transfer*, **53**(2), 743–752.

[10] M. Bouraoui, M. S. Rouabah, A. Abidi-Saad, A. Korichi, C. Popab, G. Polidori.,2017 , "Numerical simulation of a double skin with secondary ventilation flow on adiabatic WALL," Frontiers in Heat and Mass Transfer (FHMT), 8, 18 (2017).

[11] S.V. Patankar, 1980, "Numerical Heat Transfer and Fluid Flow," Hemisphere publication Co., NY.