Numerical Study of Natural Convection in an Open-Ended Channel with Surface Radiation: Comparison with Experimental Data

Agounoun Rachid¹, Zoubir Amine¹, Kadiri Imaï², Sbai Khalid¹, Rahmoune Miloud¹,
¹ Laboratoire d’Etude des Matériaux Avancés et Applications (LEM2A), Université Moulay Ismail, Faculté des sciences, Ecole Supérieure de Technologie Meknès, Maroc
² Université Cadi Ayyad, Faculté des sciences et techniques de Marrakech, Morocco
Corresponding author: ragounoun@yahoo.fr

Abstract - The present study deals with natural convection flow coupled with surface radiation in a vertical open-ended channel with wall constant heat flux. The experimental and numerical investigations are both conducted using air as the working fluid. The numerical code is developed using finite differences scheme to solve the Navier-Stokes equations under the Boussinesq assumption in two dimension. Surface radiation heat transfer is taken into account by using the radiosity method. Concerning the experimental apparatus, it consists of a vertical channel heated uniformly on one wall while the other is isolated. Temperature and velocity measurements are provided for two modified Rayleigh numbers Ra=3.9×10^4, 9.1×10^5. The numerical code is first validated with previous numerical investigations. Then, comparison between experimental and numerical results is performed with and without surface radiation and discrepancies on temperature and velocity profiles are discussed. It was found that surface radiation play a significant role in the evolution of the dynamical and thermal behaviour of the natural convection flow inside the channel.

Index Term - Radiation; Natural convection; Open-Ended Channel.

I. INTRODUCTION

The working temperature of building integrated photovoltaic components (BIPV) is a key factor in terms of efficiency. The configurations for integration of these components can cause an increase in operating temperature and a decrease of their electrical efficiency. In order to cool them by natural convection, they are mounted in a double-skinned configuration, meaning that they are separated from the building envelope by an open-ended air gap. Identifying and analyzing the physical phenomena that govern the air flow and fluid/wall heat transfer modes in this vertical channel are therefore essential to improve photovoltaic panel cooling. This can only be performed by detailed numerical simulations validated by experimental studies. Elenbaas [1] was the first who worked on natural convection between two isothermal parallel plates for cooling electronic components. He found that convective heat transfer is characterized by Nusselt, Grashof and Prandtl numbers and presented a correlation giving the Nusselt number as a function of the modified Rayleigh number. The convective heat transfer is calculated by deducting radiation exchange which is estimated from experimental measurements. This study cover a range of modified Rayleigh (Ra*) ranging from 0.1 to 10^5 thereby covering the two remarkable flow regimes (conductive and boundary layer regimes). Sparrow and Bahrami [2] conduct an experimental investigation on natural convection in a vertical channel with two walls symmetrically heated with uniform temperature. They used the sublimation technique of naphthalene which is a chemical crystalline substance placed in the channel. As the naphthalene sublimates at ambient temperature, a reduction of its thickness occurs. Measuring the thickness of this change and using an analogy between mass transfer and heat transfer, it is possible to deduce the convective heat transfer coefficient. This technique separates the convection of other heat transfer modes (radiation/conduction) while the experimental techniques commonly used to measure the convective heat transfer coefficient include inevitably the effects of radiative and conductive heat transfer. Sparrow and Bahrami [2] measured Higher averaged Nusselt numbers in comparison with those measured by Elenbass [1]. Differences are related to variable thermophysical properties of the air, to surface radiation exchange and to heat losses at the channel edges (conductive/convective). An interesting experiment is implemented by Web and Hill [3] they studied natural convection of air in a vertical channel with adiabatic extensions at both the entry and the exit. The adiabatic extensions are used to control the radiative losses at the ends of the channel, which would otherwise have been radiatively exposed to relatively cool environment. The study was conducted for a range of modified Rayleigh number ranging from 503 to 1.75×10^5. Webb and hill [3] have established correlations based on Ra* in terms of local and averaged Nusselt number and maximum wall temperature. Results in of average Nusselt number are close to those obtained by sparrow and Gregg [4] in the case of a flat plate isolated for large numbers of Ra*. However, the differences are about 11% between the two studies. They explain this by a probable influence of radiative losses, the effects of conduction and the variation of thermophysical properties of air a fuction of temperature.

The are many investigations done with assumption of pure naturel convection. These are either numerical studies in which only natural convection is taken into account [5][6][7][8][9], or experimental studies in which radiative heat transfer are neglected or not discussed. In order to characterize the pure convective heat exchange, some researchers have used water as the working fluid. The choice of the water here is used to neutralize the effects of radiation and to be interested only in natural convection [10][11][12][13][14].

Other researchers studied the effect of surface radiation on natural convection [15][16][17]. However, their geometries are different ours. An interesting study was done by Ruli [18]. He used a similar geometry but with adiabatic extensions at both the
channel entry and exit. They made a comparison of their numerical results with Webb and Hill experimental ones. They found a good agreement in terms of wall temperature and Nusselt number but only for small Rayleigh numbers.

Most of studies have been done using air as working fluid and the radiation part of heat transfer is still not completely known [2][3][19]. In order to contribute in the understanding of the physical mechanisms of natural convection with surface radiation, we present both experimental and numerical investigations of thermal and dynamic quantities for natural convection coupled with surface radiation. Although many experimental and numerical studies on naturel convection between parallel plates have been carried out, most studies were limited to thermal measurements and heat transfer analysis. Few studies were interested in the flow dynamics quantities. However, dynamic quantities are needed to qualify numerical simulations with the experimental measurements.

The experimental set up, instrumentation used and experimental procedure are described in the following. Secondly, we describe the physical model and the numerical study. Algorithms for solving temperature and pressure-velocity coupling are briefly clarified. Numerical methods are then validated by comparing numerical results with those of Ru Li [18] in terms of mass flow rate and Nusselt number. The studies addresses the problem of natural convection of air with surface radiation between two vertical walls, one wall is isolated while the other one is heated with uniform heat flux. Finally, a comparison between our experimental data and numerical results is made on wall temperature, mass flow rate and velocity profiles for two modified Rayleigh numbers.

II. EXPERIMENTAL APPARATUS

The experimental setup utilized in the present study is illustrated in Fig. 1. The test room has a volume of 126 m$^3$ and is situated in the larger inner volume of a training platform. The floor area of test is 6×6 m$^2$ and its ceiling height is 3.5 m. The channel consists of two parallel walls of 1.5 m in height and 0.7 m in depth separated by a distance of 0.1 m, the aspect ratio b/H is about 1/15. One of the plates is heated with uniform electrical heat flux, while the other is insulated. The channel input is beveled at 30° to eliminate the unsteady recirculation zones developing from the leading edge of the two walls. Additionally, horizontal sides have been installed at the inlet and outlet of the channel in order to streamline flows in these sections. In the span-wise direction, the channel was fitted with two vertical transparent sheets in order to prevent lateral infiltration. In order to prevent heat losses, the rear faces of each panel isolated, by a block of polyurethane (λ =0.027 W/mK and 12 cm) thick integrated in a wooden frame.

15 independent stainless steel strip 10 cm wide, 50 microns thick and low conductivity (λ=13 W/mK) are extended on each side of the inner channel walls. The plates emissivity $\varepsilon$ is 0.092 corresponding to the emissivity of the strip. Electrical power is led into each strip independent and dissipated in the channel by the joule effect. Each of the plates is instrumented using 75K – type thermocouples (120 microns diameter), located on the vertical axis of symmetry of each wall (along y). The reference temperature is measured by a thermocouple at the inlet of the channel. Back losses by conduction along x, are evaluated from 15 thermocouples (type K, 120 µm diameter) placed in the insulation 4 cm depth on a vertical line vis-à-vis surface thermocouples. These losses are about 3.5% of the electrical power injected into strips.

The measurements were performed after a period of steady state around 6 hours. Global field measurement speed is ensured by a PIV (particle image velocimetry). The system consists of a Nd: YAG laser 120 mJ, a CCD camera (pco.200) and processing software (Davis 7.2). The acquisition frequency is 11 Hz images and tests lasting between 40 and 60 minutes. The flow is first seeded with microparticles (<1µm) oil DEHS (Di-Ethyl-Hexyl-Sebacat) produced by an atomizer. With their desity of 912 Kg/m$^3$ and size, these particles exhibit a good compromise between the ability to follow the flow (the difference between the particule velocity and the flow velocity is of 0.01%) and light diffusion, according to [20]. The experiment was planted for 5 minutes and stabilization time and space homogenization of particles in the flow is 30 minutes elapse between the end of the seeding and early PIV measurements.

### Table I

<table>
<thead>
<tr>
<th>Experiences</th>
<th>$q_w$ W/m$^2$</th>
<th>Temperatures °C</th>
<th>$Ra_b$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Inlet</td>
<td>Outlet</td>
</tr>
<tr>
<td>1</td>
<td>20</td>
<td>21.98</td>
<td>3.9 × 10$^3$</td>
</tr>
<tr>
<td>2</td>
<td>50</td>
<td>23.75</td>
<td>9.1 × 10$^3$</td>
</tr>
</tbody>
</table>
experimental apparatus. The electrical wall heating produces a temperature gradient in the fluid leading to small density differences which gives rise to the buoyancy force. Thus, a natural convective flow occurs with the fluid entering at the bottom and leaving at the top. Throughout the analysis the following assumptions are made: fluid properties, except density, are independent of temperature; density variations are significant only in the buoyancy force; and the flow is two-dimensional, laminar and incompressible with negligible viscous dissipation. It is also assumed that the vertical walls are made of the same material emissivity \( \varepsilon \). The air flowing inside the channel is assumed to be transparent to thermal radiation. The external environment radiates on the walls through the open upper and lower sections. A black body model at ambient temperature is used to take into account the heat radiation of the ambient air. Thus, the well-known unsteady Boussinesq equations in their elliptic form, governing the flow are:

\[
\begin{align*}
\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} &= 0 \\
\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} &= -\frac{1}{\rho} \frac{\partial p}{\partial x} + \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \\
\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} &= -\frac{1}{\rho} \frac{\partial p}{\partial y} + \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{g \beta}{\rho} (T - T_0) \\
\frac{\partial T}{\partial t} + u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} &= \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2}
\end{align*}
\]

Where \( u, v, p \) are respectively, horizontal velocity, vertical velocity and pressure. \( \rho_0, \nu_0, \beta_0, \alpha_0 \) are respectively, mass density, kinematic viscosity, thermal expansion coefficient and thermal diffusivity calculated at some reference temperature \( T_0 \). By introducing the following non-dimensional quantities with reference variables \( (L_0, U_0, t_0) \):

\[
\begin{align*}
L & = b, \quad X = \frac{x}{L_0}, \quad Y = \frac{y}{L_0} \\
U_0 & = \left( \frac{\beta_0 \gamma \Delta T b}{\Pr} \right)^{1/3} \\
U & = \frac{u}{L_0}, \quad V = \frac{u}{L_0}, \quad \theta = \frac{T - T_0}{\Delta T}, \quad P = \frac{p - p_0}{\rho U_0^2}, \quad t = \frac{b}{U_0}
\end{align*}
\]

The governing equations are transformed as follows:

\[
\begin{align*}
\frac{\partial U}{\partial x} + \frac{\partial V}{\partial y} &= 0 \\
\frac{\partial U}{\partial t} + U \frac{\partial U}{\partial x} + V \frac{\partial U}{\partial y} &= -\frac{1}{\Pr \cdot Ra} \left( \frac{\partial^2 U}{\partial x^2} + \frac{\partial^2 U}{\partial y^2} \right) \\
\frac{\partial V}{\partial t} + U \frac{\partial V}{\partial x} + V \frac{\partial V}{\partial y} &= -\frac{1}{\Pr \cdot Ra} \left( \frac{\partial^2 V}{\partial x^2} + \frac{\partial^2 V}{\partial y^2} \right) + Pr \theta \\
\frac{\partial \theta}{\partial t} + U \frac{\partial \theta}{\partial x} + V \frac{\partial \theta}{\partial y} &= \frac{1}{Pr \cdot Ra} \left( \frac{\partial^2 \theta}{\partial x^2} + \frac{\partial^2 \theta}{\partial y^2} \right)
\end{align*}
\]

Where \( \Delta T \) used to define the adimensional temperature is \( q_u d / \lambda_0 \), with \( q_u \) is the uniform heat flux fixed at the wall. The Prandtl number \( Pr \) is defined by: \( Pr = \nu_0 / \alpha_0 \). Furthermore, the Rayleigh number, expressing the strength of buoyancy, is constructed with the plate spacing as follows:
where $A_b = L_b/b$ is the aspect ratio of the heated part of the channel (in Fig. 1-b $L_b = H$ and in Fig. 2 $A_b = A/2$).

$$Ra_b = \frac{\beta \rho g_c \vartheta_b^4}{\nu_0 \vartheta_0^4}$$  \hspace{1cm} (3)

This finally leads to, the modified Rayleigh number (or Elenbaas number):

$$Ra_p = \frac{Ra_b}{A_h}$$  \hspace{1cm} (4)

Where $A_h = L_h/b$ is the aspect ratio of the heated part of the channel (in Fig. 1-b $L_h = H$ and in Fig. 2 $A_h = A/2$).

$$\begin{align*}
\frac{\partial \theta}{\partial t} + \frac{\partial \theta}{\partial x} &= -1; \quad \text{for } X = 0; \quad A/4 \leq Y \leq 3A/4 \\
\frac{\partial \theta}{\partial t} + \frac{\partial \theta}{\partial x} &= 0; \quad \text{for } X = 0; \quad 0 \leq Y \leq A \\
\frac{\partial \theta}{\partial t} + \frac{\partial \theta}{\partial x} &= 0; \quad \text{for } X = 1; \quad 0 \leq Y \leq A
\end{align*}$$  \hspace{1cm} (5)

The selected boundary conditions for the thermally driven open channel corresponding to Fig. 2 are shown in system of equation (5). When the boundary conditions are expressed in pressure at the inlet, the generalized Bernoulli theorem is used to take into account the pressure loss between the upstream and the entry of the channel (upstream velocity assumed to be zero and the pressure is equal to $p(x, y) = p_{\text{atm}}$). Indeed, [6], [21] and [8] found that the driving pressure at the entrance of the channel, due to the velocity of the incoming fluid, cannot be neglected, the influence is even more important as the Rayleigh number increases. Thus, local Bernoulli pressure is applied at the channel entrance. Moreover, pressure at the outlet is taken equal to exterior pressure assumed to be zero (atmospheric pressure in dimensional form). When boundary conditions are written on velocity, zero Neumann boundary condition is applied on streamwise velocity $V$ and homogeneous Dirichlet boundary condition is applied on transverse velocity $U$. Actually, we assume that the incoming flow at the inlet is parallel to the channel. Concerning the thermal boundary conditions, we neglect the longitudinal conductive heat flux at the channel outlet, knowing that the transverse thermal gradients are more important. If in some case, an incoming flow can appear at the channel outlet, we assume it has the outside temperature. In this approximation, we neglect the fact that the incoming fluid may be slightly heated in the external surroundings.

In the presence of surface radiation, only thermal boundary conditions at walls changes.

As the net radiation flux density in dimensionless form is $q_{r}^* = qr / q_{wv}$, the thermal boundary conditions at walls are written (see Fig. 2):

$$\begin{align*}
\frac{\partial \theta}{\partial x} + q_r^* &= -1; \quad \text{for } X = 0; \quad A/4 \leq Y \leq 3A/4 \\
\frac{\partial \theta}{\partial x} + q_r^* &= 0; \quad \text{for } X = 0; \quad 0 \leq Y \leq A/4 \text{ et } 3A \leq Y \leq A \\
\frac{\partial \theta}{\partial x} - q_r^* &= 0; \quad \text{for } X = 1; \quad 0 \leq Y \leq A
\end{align*}$$  \hspace{1cm} (6)

Where the net radiation flux density $q_r$ on a surface $S$, the difference between the radiosity leaving $S$ and radiation coming from other surfaces, diffuse and gray or black is given by the expression:

$$q_r = J - G = J - 'J.F$$  \hspace{1cm} (7)

Where $F$ is the shape factor between $S$ and other surfaces. The radiosity $J$ is written as follow:

$$J = (1 - e)J.F + e\sigma(\Delta T + T_0)^4 / q_w$$  \hspace{1cm} (8)

With $e$ is the emissivity and $\sigma$ is the Stephane-Boltzmann constant.

III.2. Numerical Methods

Equations of conservation of mass, momentum, and energy, are discretized with finite differences schemes in space and time. The temporal scheme adopted to discretize the system of equations (2) is based on the three-level second-order Euler backward scheme with Adams-Bashforth extrapolation as proposed in [29]. In this scheme, we treat implicitly the diffusion terms and explicitly the convective ones. The pressure velocity decoupling is achieved by adopting an iterative solution based on a projection algorithm [30] [31]. This algorithm consists of finding first a velocity prediction based on pressure field approximation. Then an equation for pressure correction is derived from the continuity equation, it’s then solved to obtain the pressure correction to update the velocity fields and pressure. Concerning the spatial discretization, the computational domain is discretized on a non-uniform Cartesian grid. Use is made of a staggered variable arrangement. A second order forward and backward difference schemes are used to treat the Neumann boundary condition on vertical velocity respectively at the inlet and at the outlet. After discretization, we obtain a Poisson equation on pressure correction and Helmholtz equations on velocity and temperature. We solve the Poisson and Helmholtz equations respectively using partial diagonalization method and tridiagonal matrix algorithm. For more details see [32].

Even that combined convection and surface radiation occurs only on the thermal boundary conditions of vertical walls. The problem of surface radiation is discretized spatially on the same grid used for the Navier-Stokes equations in order to find the local distribution of the radiative flux on both walls. For temporal discretization, we choose a semi-implicit treatment for surface radiation. $\bar{q}_{r}^{*^{(n+1)}} = 2\bar{q}_{r}^{*^{(n)}} - \bar{q}_{r}^{*^{(n-1)}}$ [16]. For example,

$$-(\partial \theta^{(n+1)}) / (\partial X) - \bar{q}_{r}^{*^{(n+1)}} = -(\partial \theta^{(n+1)}) / (\partial X) - 2\bar{q}_{r}^{*^{(n)}} + \bar{q}_{r}^{*^{(n-1)}}$$

This treatment allow us to obtain first $\theta^{n+1}$ and then $\bar{q}_{r}^{*^{(n+1)}}$ solving the radiative transfer problem (7).

IV. Code validation

The aim of this section is to validate the numerical code in the case of natural convection between vertical walls with surface radiation. A first validation in the case of pure natural convection was done in a previous works Zoubir et al. [22] [23]. A good agreement was found with other works on characteristics...
quantities (mass flow rate and Nusselt number). The comparison was made for modified Rayleigh number of \( Ra^* = 10^7 \) and aspect ratio of 10 and different grid resolutions. A spatial grid analysis was also done and shows that our numerical code is globally of second-order in space [22]. In pure natural convection, the problem depends only on the modified Rayleigh number. But in coupled radiation and convection, other parameters such as \( \varepsilon, \lambda, T_0, b \) and \( \Delta T \) must be specified. In the present comparison with Ru Li et al. [18] results we have \( \varepsilon = 0.1; \lambda = 0.025 \text{ W/(mK)}; T_0 = 290 \text{ K}; b = 3.80 \text{ cm} \) and \( Ra^* = 6.69 \times 10^5 \). The channel geometry under investigation is shown in Fig. 2 and the corresponding height is \( H = 30.44 \text{ cm} \). Table II presents a comparison in term of Nusselt numbers and mass flow rates between the present study and Ru Li et al. [18] one. One can notice that our numerical results are in very good agreement in terms of radiative and convective Nusselt numbers and flow rates. The discrepancy is less than 2% on characteristic quantities for all tested grid resolutions. Fig. 3 and 4 shows horizontal profiles respectively of temperature and vertical velocity for different heights \( Y = A/4, 3A/4, A \) and \( \varepsilon = 0.1 \). The qualitative comparison shows also an excellent agreement between the results of both studies.

### Table II

**Comparison of Radiative and Convective Nusselt Number and Mass Flow Rate Between This Work and the Study of [18] for**

<table>
<thead>
<tr>
<th>Reference [18]</th>
<th>Present study</th>
</tr>
</thead>
<tbody>
<tr>
<td>* ( m_a )</td>
<td>* ( Nu_c )</td>
</tr>
<tr>
<td>40 ( \times ) 400</td>
<td>4.046</td>
</tr>
<tr>
<td>80 ( \times ) 800</td>
<td>4.509</td>
</tr>
<tr>
<td>120 ( \times ) 1200</td>
<td>4.516</td>
</tr>
<tr>
<td>160 ( \times ) 1600</td>
<td>4.458</td>
</tr>
</tbody>
</table>

(a)

<table>
<thead>
<tr>
<th>Reference [18]</th>
<th>Present study</th>
</tr>
</thead>
<tbody>
<tr>
<td>* ( m_a )</td>
<td>* ( Nu_c )</td>
</tr>
<tr>
<td>40 ( \times ) 400</td>
<td>5.991</td>
</tr>
<tr>
<td>80 ( \times ) 800</td>
<td>6.239</td>
</tr>
<tr>
<td>120 ( \times ) 1200</td>
<td>6.248</td>
</tr>
<tr>
<td>160 ( \times ) 1600</td>
<td>6.252</td>
</tr>
</tbody>
</table>

(b)

\[ q_c = 300 \text{ W/m}^2, b = 3.80 \text{ cm} \) and \( Ra^* = 6.65 \times 10^5 \) with \( \varepsilon = 0.1 \) (a), \( \lambda = 0.025 \text{ W/(mK)} \) and \( T_0 = 290 \text{ K} \); the comparison of results is done according to different grid resolutions.

V. Comparison Between Numerical and Experimental Results

The configuration of interest is an air-filled vertical channel and the corresponding natural convection flow with surface radiation is studied both experimentally and numerically. The boundary conditions on the heated walls were isoflux conditions and the external channel temperature was regulated to remain within the band of the Boussinesq approximation. Wall temperatures and velocity were measured for two heat fluxes (see Table I). Uncertainties on velocity and temperature measurements are respectively 0.1 m/s and \( T = \pm 0.12^\circ \text{C} \). Numerical simulations are also performed for the two experimental configurations. Computations are done on Intel Xeon E5520 2.27 GHz processor with 4 GB memory. Numerical simulations led to steady laminar flows and experiments are in unsteady state regime. Despite of the difference observed on the time behavior, flow rates and local Nusselt numbers (calculated at the channel outlet) are compared between experimental and numerical results (see Table III). The experimental flow rate is calculated at the inlet and the outlet by integrating the measured velocity assuming a 2D profile. In pure natural convection case, numerical results in terms of mass flow rate are different from experimental ones of 70% and 60% whereas they are different of 20% and 15% in the case of coupled convection and surface radiation respectively for \( Ra^* = 3.9 \times 10^7 \) and \( Ra^* = 9.1 \times 10^7 \). In term of Nusselt number, agreement is better for the first experiment and discrepancy is important for the second configuration. Discrepancies between numerical and
experimental results in terms of mass flow rates and Nusselt numbers decrease when adding surface radiation.

### TABLE III

<table>
<thead>
<tr>
<th>$Ra^*$</th>
<th>$m_{\text{exp}}$</th>
<th>$m_{\text{num}}$</th>
<th>$N_{\text{u exp}}$</th>
<th>$N_{\text{u num}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>(P.C)</td>
<td>(S.R)</td>
<td>(P.C)</td>
<td>(S.R)</td>
</tr>
<tr>
<td>3.9 $\times 10^3$</td>
<td>15.12</td>
<td>4.68</td>
<td>12.221</td>
<td>11.400</td>
</tr>
<tr>
<td>9.1 $\times 10^3$</td>
<td>22.8</td>
<td>8.88</td>
<td>19.32</td>
<td>13.746</td>
</tr>
</tbody>
</table>

Fig. 5. Comparison between experimental and numerical data on vertical velocity at $Y=A/2$ for natural convection with and without surface radiation, with $Ra^*=3.9 \times 10^3$.

Fig. 6. Comparison between experimental and numerical data on vertical velocity at $Y=A/2$ for natural convection with and without surface radiation, with $Ra^*=3.9 \times 10^5$.

Possible sources of discrepancies are the following. As the experiment is done in a perturbed external medium, discrepancies could come from the fact that the code fails to predict the perturbations at the channel extremities. In simulations, the inlet boundary conditions do not introduce perturbations and the head losses can decrease the mass flow rate inside the channel. In addition, a thermal stratification of 1°C/m has been measured between the inlet and the outlet. It can also create pressure losses which may slow down the air flow and reduce therefore the channel flow rate. These hypothesis are suggested by the comparison of velocity profiles between experiments and numerical simulations. For the second experiment, there can be in addition a change of flow regime for the second modified Rayleigh numbers. This change can be seen from the wall temperature distributions plotted in Fig. 7 against the channel height: in experiment 2 wall temperature decreases in the downstream part. This behavior can also be explained by the heat energy losses by conduction and radiation at the channel exit as it was described by Webb and Hill [3] in his experiments. Note that the code provides a good prediction of the wall temperature but only for the smallest modified Rayleigh simulation but it doesn’t exist in the corresponding experiment. This can be explained by the surface radiation effects. In pure convection, the air heated at the wall rises by natural convection and creates a dynamic boundary layer along the heated wall. To aliment this boundary layer, the air enters from the top when the bottom alimentation is not sufficient. In the presence of surface radiation, the redistribution of the heat energy supplied by the heated wall results in a decrease in the intensity of the flow along the left wall and the appearance of an ascending movement along the unheated wall. This new flow eliminates a possible re-circulation at the exit near the adiabatic wall and the mass flow rates increases significantly (2.5 and 2 times higher respectively for the two numerical simulations).
numbers. Nevertheless, numerical simulations give a good trend on the behavior of the flow found in experiments especially in the case of coupled natural convection and surface radiation. Then, we can summarize the surface radiation effects as follows:

- Increase the mass flow rate inside the channel
- Increase the heat exchange between the wall and the air and then reduce the wall temperature
- Eliminate flow reversals

VI. Conclusion

We have presented a numerical code for studying natural convection with and without surface radiation in an open-ended channel. Finite difference scheme of second order in time and space was adopted to discretize the Navier-Stokes equations under the Boussinesq assumption. The code was first validated with the Ru Li et al. [18] numerical study in the case of coupled convection and radiation in the case of vertical channel with adiabatic extensions at both the inlet and the outlet. The difference observed is of order of 2% on characteristics quantities. Finally, numerical simulations were performed for an air-filled channel and results were confronted to experimental data for two modified Rayleigh numbers $Ra^t = 3.9*10^5$ and $Ra^t = 9.1*10^5$ and aspect ratio of $A = 15$. Comparison on flow rate, averaged Nusselt number, vertical velocity and wall temperature profiles has been done between numerical and experimental data and possible reasons for these discrepancies are discussed. Further studies need to be carried out in order to improve both experimental and numerical approaches and understand the physical phenomena. Furthermore, numerical and experimental studies emphasize that surface radiation change completely the structure of the natural convection flow and enhance more heat exchange. Then, in experimental point of view, it seems essential to control the radiative properties of surfaces, even when the temperature differences are small. Similarly, numerical simulations of natural convection must integrate the sensitivity of the structure of the flow and heat transfer in thermal radiation when surface temperatures are not regulated.

REFERENCES
