# TURBULENT MIXED CONVECTION IN HEATED VERTICAL CHANNEL

by

***Ameni MOKNI*** [a, b*], ***Hatem MHIRI*** [b], ***Georges LE PALEC*** [a], ***and Philippe BOURNOT*** [a]

[a] Institut Universitaire des Systèmes Thermiques Industriels UMR CNRS, Marseille, France
[b] Unité de Thermique et Environnement, Ecole Nationale d'Ingénieurs de Monastir, Monastir, Tunisia

Original scientific paper
UDC: 66.045.1:532.517.4
DOI: 10.2298/TSCI1001125M

*In this paper an investigation of mixed convection from vertical heated channel is undertaken. The aim is to explore the heat transfer enhancement obtained by adding a forced flow, issued from a flat nozzle located in the entry section of a channel, to the up-going fluid along its walls. Combined forced and free convection are studied in order to increase the cooling requirements. The study is conceded for two Rayleigh number. The first case corresponds to two separate boundary layers so the channel acts as two independent plates. For the second case the two boundary layers are attached. Calculations are carried out with air as the working fluid by changing the jet velocity in order to optimize the system to give the maximum heat flow rate over the chimney. The system of governing equations is solved with a finite volumes method and an implicit scheme. The results obtained show that the jet-wall activates the heat transfer, as does the drive of ambient air by the jet.*

Keywords: *channel, wall jet, mixed convection, boundary layer*

## Introduction

Natural convection is an energy transport process that takes place as a result of buoyancy induced fluid motion occurring in the presence of a body force field (gravity). The density of gases and liquids depends on their temperature, generally decreasing with increasing temperature (fluid expansion). In such a way, it is possible that a convection current exist even when there is no additional forced convection flow.

In mechanical and environmental engineering, mixed convection is a frequently encountered thermal fluid phenomenon, which exists in atmospheric environment, urban canopy flows, ocean currents, gas turbines, heat exchangers, and computer chip cooling systems, *etc*. In the early development of convective heat transfer studies, forced and natural convections were considered separately and the interaction between these two physical processes was ignored. Modern research combining forced and natural convection was initiated in the 1960's and mainly based on experimental approaches [1].

* Corresponding author; e-mail: ameni26@yahoo.fr

Two-dimensional channel formed by parallel plates is the most frequently used configuration in convection air cooling of electronic equipment. The passive character of cooling by natural convection makes it attractive for applications in electronic devices. However, in order to increase the cooling requirements, researches for methods to improve the heat transfer are crucial [2-5].

Heat transfer by natural convection from vertical plates with uniform wall temperature or heat flux has received considerable attention, and extensive discussions about this subject are available in the literature. Hugot [6] undertook an experimental study of the interaction of the boundary layers developing along two large parallel vertical plates. This study enabled him to measure the local heat transfer coefficients for various spacing and temperatures corresponding to Grashof numbers ranging from $5 \cdot 10^5$ to $2 \cdot 10^{11}$. The boundary layers interaction was defined by comparison with the single plate results. Moreover, information provided by the velocity, temperatures profiles and their fluctuations lead to a better knowledge of the turbulence of the flow.

Myamoto *et al*. [7] and Auletta *et al*. [8] studied experimentally natural convection flow and heat transfer in a heated vertical channel. From a theoretical point of view, the literature gives extensive results for both turbulent and laminar flows. Turbulent heat transfer resulting in a channel have been reported in references [4, 9-12]; Fedorov [10] proposed scaling correlations for induced flow rate and heat characteristics in an asymmetrically heated channel. Later Versteegh *et al*. [11, 12] also studied an asymmetrically heated channel in order to find wall functions from the scaling behaviour of the flow. Other authors treated this problem for laminar flows [13]. The development of laminar buoyancy-driven convection between two vertical plane plates asymmetrically heated by a constant heat flux was studied by Dalbert *et al.* [14, 15]. Desrayaud *et al.* [16] undertook a numerical study of the laminar natural convection in an isothermal vertical channel in which rectangular ribs are symmetrically located on each wall. Kheireddine [17] studied the influence of the pressure loss on the induced mass flux for a buoyancy-driven flow. The calculation domain was extended far from the channel, and they showed that the free pressure boundary location is negligible if placed at a distance greater than 4 times the channel width. However theory [18] and experiments show that both laminar and turbulent flows can occur at the same time if the length of the plate is rather significant. For the sake of completeness, a few studies concerned the mixed convection regime: we can cite Penot *et al*. [15] who proposed useful correlations to determine the flow rate, the fluid temperature, and the Nusselt number. More recently, Najam *et al*. [19] studied numerically the mixed convection in a "T" form cavity heated with a constant heat flux and subjected to an air blast entering by the bottom. They showed the competition between natural and forced convection. The heat transfer was found maximal in the zone where the role of natural convection is more significant.

The present theoretical study is concerned with mixed convection in a heated vertical channel submitted to a vertical jet of fresh air entering by the bottom. The channel is long enough that the flow becomes turbulent before the exit. The influence of this forced additional jet is analyzed by using the low Reynolds number $k$-$\varepsilon$ turbulence model in order to simulate the turbulent flow. Numerical results are reported for dry air as coolant, and for two Rayleigh numbers corresponding: (1) to two attached boundary layers, and (2) to two separated ones.

## Assumptions and governing equations

The geometry of the problem investigated herein is described in fig.1. We consider a vertical channel which simulates a chimney. A gas jet is issued from a flat nozzle located at the

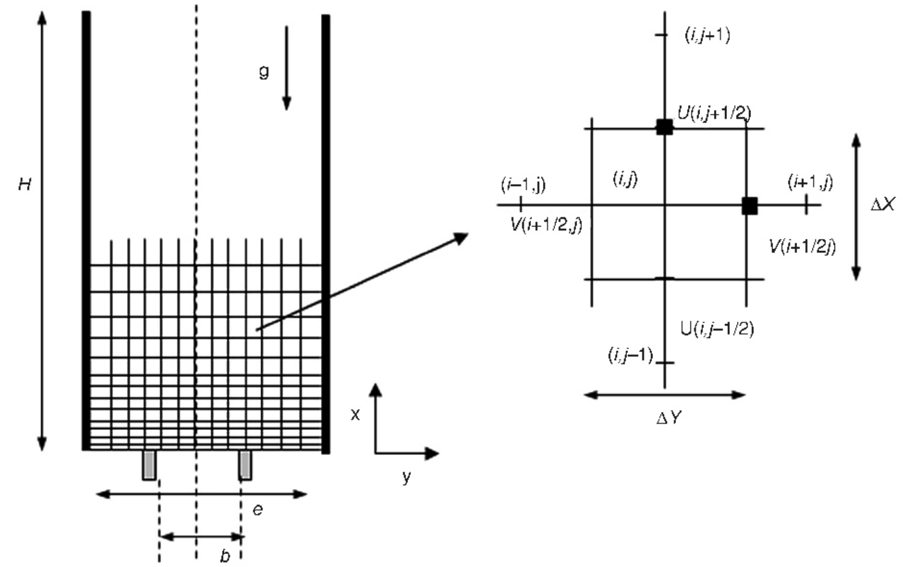


**Figure 1. Physical domain and coordinates system**

bottom of the channel. The chimney walls are subject to a constant heat flux. The flow is assumed steady and incompressible. Both natural and mixed convection cases are considered by using the Boussinesq approximation in which the density varies linearly with temperature. Other thermo-physical quantities are assumed to be constant.

Let us introduce the dimensionless variables defined by:

$$X \quad \frac{x}{e}, \; Y \quad \frac{y}{e}, \; U \quad \frac{ue}{\alpha}, \; V \quad \frac{ve}{\alpha}, \; P \quad \frac{(p \quad \rho g x)e^2}{\rho\alpha^2}, \; \theta \quad \frac{T \quad T_\infty}{\varphi e}\lambda, \; K \quad \frac{k\varepsilon^2}{\alpha^2}, \; E \quad \frac{\varepsilon e^4}{\alpha^3} \tag{1}$$

The dimensionless governing equations for two-dimensional buoyancy-driven flows, with no viscous dissipation, can be written as:

*Continuity equation*:

$$\frac{\partial U}{\partial X} \quad \frac{\partial V}{\partial X} \quad 0 \tag{2}$$

*Momentum equation in x-direction:*

$$U\frac{\partial U}{\partial X} \quad V\frac{\partial U}{\partial Y} \quad \frac{\partial P}{\partial X} \quad (\mathrm{Pr} \quad \mathrm{Pr_t}) \quad \frac{\partial^2 U}{\partial X^2} \quad \frac{\partial^2 U}{\partial Y^2} \quad \frac{2}{3}\frac{\partial K}{\partial X} \quad \mathrm{Ra}\,\mathrm{Pr}\,\theta \tag{3}$$

*Momentum equation in y-direction:*

$$U\frac{\partial V}{\partial X} \quad V\frac{\partial V}{\partial Y} \quad \frac{\partial P}{\partial Y} \quad (\mathrm{Pr} \quad \mathrm{Pr_t}) \quad \frac{\partial^2 V}{\partial X^2} \quad \frac{\partial^2 V}{\partial Y^2} \quad \frac{2}{3}\frac{\partial K}{\partial Y} \tag{4}$$

*Energy equation:*

$$U\frac{\partial \theta}{\partial X} \quad V\frac{\partial \theta}{\partial Y} \quad 2 \quad \frac{\partial^2 \theta}{\partial X^2} \quad \frac{\partial^2 \theta}{\partial Y^2} \tag{5}$$

*Turbulent kinetic energy equation:*

$$U\frac{\partial K}{\partial X} \quad V\frac{\partial K}{\partial Y} \quad \mathrm{Pr} \quad \frac{\mathrm{Pr_t}}{\sigma_k} \quad \frac{\partial^2 K}{\partial X^2} \quad \frac{\partial^2 K}{\partial Y^2} \quad E \quad G_{\mathrm{DK}} \quad G_{\mathrm{DB}} \tag{6}$$

*Rate of dissipation of turbulent kinetic energy equation:*

$$U\frac{\partial E}{\partial X} \quad V\frac{\partial E}{\partial Y} \quad \mathrm{Pr} \quad \frac{\mathrm{Pr_t}}{\sigma_\varepsilon} \quad \frac{\partial^2 E}{\partial X^2} \quad \frac{\partial^2 E}{\partial Y^2} \quad C_1\frac{E}{K}G_{\mathrm{D}} \quad C_2\frac{E^2}{K} \tag{7}$$

where

$$G_{\mathrm{DK}} \quad \mathrm{Pr_t} \quad \frac{\partial U_i}{\partial X_j} \quad \frac{\partial U_j}{\partial X_i} \quad \frac{\partial U_i}{\partial X_j} \quad \frac{2}{3}K\delta_{ij} \quad \frac{\partial U_i}{\partial X_j} \quad \text{and} \quad G_{\mathrm{DB}} \quad \frac{1}{\mathrm{Fr}}\frac{\nu_{\mathrm{t}}}{\mathrm{Pr_t}}\frac{\partial \theta}{\partial X} \tag{8}$$

$E$ stands for the turbulent kinetic energy production due to shear, while $G_{\mathrm{DK}}$ is the turbulent kinetic energy production due to the mean velocity gradients, and $G_{\mathrm{DB}}$ is the turbulent kinetic energy production due to the buoyancy.

The standard *k*-$\varepsilon$ model is used, so that constants are those given by Jones *et al*. [20]: $C_1 = 1.44$, $C_2 = 1.92$, $C_3 = 0.7$, $C_\mu = 0.09$, $\sigma_\varepsilon = 1.0$, $\sigma_{\mathrm{k}} = 1.30$, and $\mathrm{Pr_t} = 1.0$.

The boundary conditions are:

$$\text{at } Y \quad \frac{e}{2H}; \quad U \quad 0; \quad V \quad 0, \quad \frac{\partial \theta}{\partial Y}_p \quad 1, \quad K \quad 0$$

$$\text{at } Y \quad \frac{e}{2H}; \quad U \quad 0, \quad V \quad 0; \quad \frac{\partial \theta}{\partial Y}_p \quad 1, \quad K \quad 0,$$

at $X = 0$;

$$-\frac{e}{2H} \quad Y \quad \frac{b}{2H}; \quad \frac{\partial U}{\partial X} \quad 0, \; V \quad 0, \; P_{\mathrm{g}} \quad \frac{Q_1^2}{2}, \; \theta \quad 0, \; K \quad \frac{3}{2} \; I_{\mathrm{t}}U^2$$

$$\frac{b}{2H} \quad Y \quad \frac{b}{2H}; \quad \frac{\partial U}{\partial X} \quad 0, \; V \quad 0, \; P_{\mathrm{g}} \quad \frac{Q_2^2}{2}, \; \theta \quad 0, \; K \quad \frac{3}{2} \; I_{\mathrm{t}}U^2, \; \text{for free convection}$$

$$U = 1, \quad V \quad 0, \; \theta = 0, \quad K \quad 0.001, \; \text{for mixed convection (presence of the jet)}$$

$$\frac{b}{2H} \quad Y \quad \frac{e}{2H}; \quad \frac{\partial U}{\partial X} \quad 0, \; V \quad 0, \; P_{\mathrm{g}} \quad \frac{Q_1^2}{2}, \; \theta \quad 0, \; K \quad \frac{3}{2}I_{\mathrm{t}}U^2$$

at $X \quad 1$;

$$\frac{\partial U}{\partial X} \quad \frac{\partial V}{\partial X} \quad \frac{\partial \theta}{\partial X} \quad 0, \qquad P \quad 0, \qquad K \quad \frac{3}{2}I_{\mathrm{t}}U^2 \quad E \quad \frac{2eK^{0.5}}{b}$$

where $Q_1 \quad \int_{b/2H}^{e/2H} UdY \quad \int_{e/2H}^{b/2H} UdY$ and $Q_2 \quad \int_{b/2H}^{b/2H} UdY$. $I_{\mathrm{t}}$ is the turbulence intensity.

The governing equations reported above are discretized on a staggered, non-uniform Cartesian grid using a finite volumes procedure. In this method, for stability considerations, scalar quantities $P, \theta, K$, and $E$ are calculated at the centre (*i, j*) of the cells where as the velocity components ($U$ and $V$) are computed on the faces of the cells ($i, j \pm 1/2$), ($i \pm 1/2, j$).

## Results and discussion

In the following sections, heat transfer along the hot wall, flow and temperature fields are examined for two Rayleigh number (Ra = 9 $10^4$; Ra = 2 $10^7$) and various Reynolds numbers based on the velocity of the jet at the nozzle exit, ranging between Re = 0 (natural convection flow) and Re = 5 $10^4$.

Based on the velocity and the temperature results, mass flow rate and heat transfer coefficient are analyzed.

### *Case 1 – attached boundary layers*

First, he problem was simulated with $H/e$ = 12.5 and a constant heat flux $\phi$ = 100 W/m$^2$, which corresponds to a Rayleigh number equal to Ra = 9 $10^4$. The Prandtl number is 0.71 (air). For low values of imposed heat flux – *i. e.* low Rayleigh numbers – the thickness of boundary layers growing up along each plate is large enough so that layers merge on the axis, before reaching the exit section of the channel.

The development of the flow with different velocities is compared. Figure 2(a) shows the natural convection drawing developed along the channel because of the thermal gradient which exists between the fluid and the hot walls. This thermal drawing will cause a vertical aspiration of air with a non-negligible mass flow rate. For this reason, we considered Re values up to 3 $10^3$, for forced convection cases. The flow structure for mixed convection cases are presented in figs. 2(b)-(c). Generally, the flow structures induced in this case are different from those observed in the last one. The difference is mainly due to the presence of the jet (forced flow). Increasing the jet Reynolds number, the flow impact the channel walls at ascending $X$ values. Be-

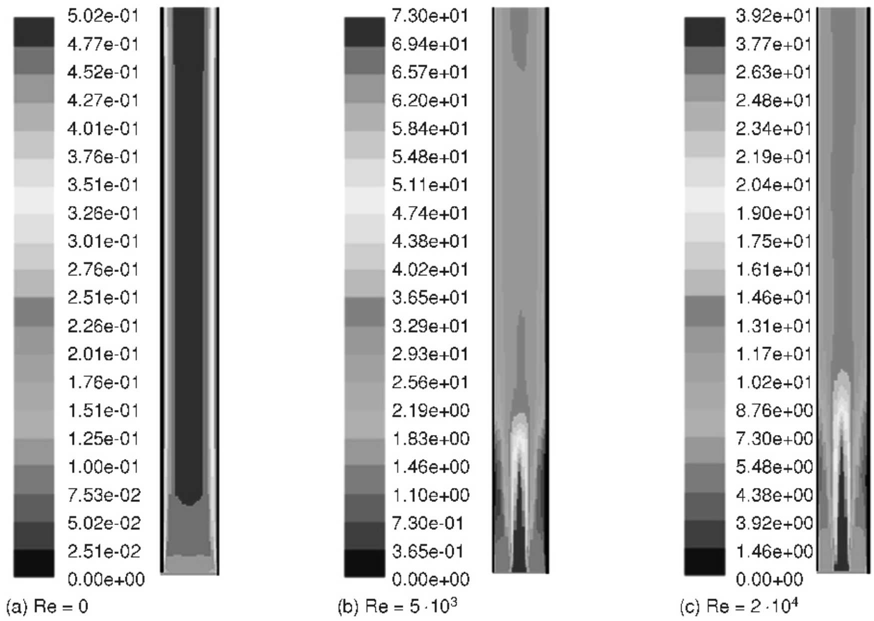


**Figure 2. Velocity development in the channel [ms$^{-1}$]; Ra = 9·10$^4$**

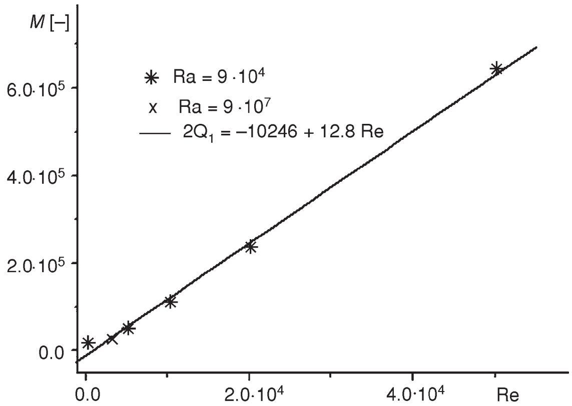


**Figure 3. Mass flow rate in both sides of the nozzle at the channel entry section**

hind this impact zone the natural convection flow is decelerated and we notice the presence of recirculation cells.

Keeping Ra = 9 $10^4$; the development of the convection flow on both sides of the nozzle for different jet Reynolds number are compared. The larger Re is, the stronger the flow is. This can be seen in fig. 3. In fact for the same Rayleigh number (same imposed heat flux) the flow increases according to the Reynolds number. We notice that the dimensionless mass flow rate is well correlated according the relationships:

$$2Q_1 = 10246 + 12.8 \text{ Re} \tag{9}$$

This flow happens in large majority due to the drive caused by the ambient air blast in the vicinity of the entry. Although the flow becomes more significant in the vicinity of the entry for important Reynolds numbers. Indeed the impacts of the jet on the walls stop the adjacent flow, which causes the apparition of hot points on the walls (fig. 4).

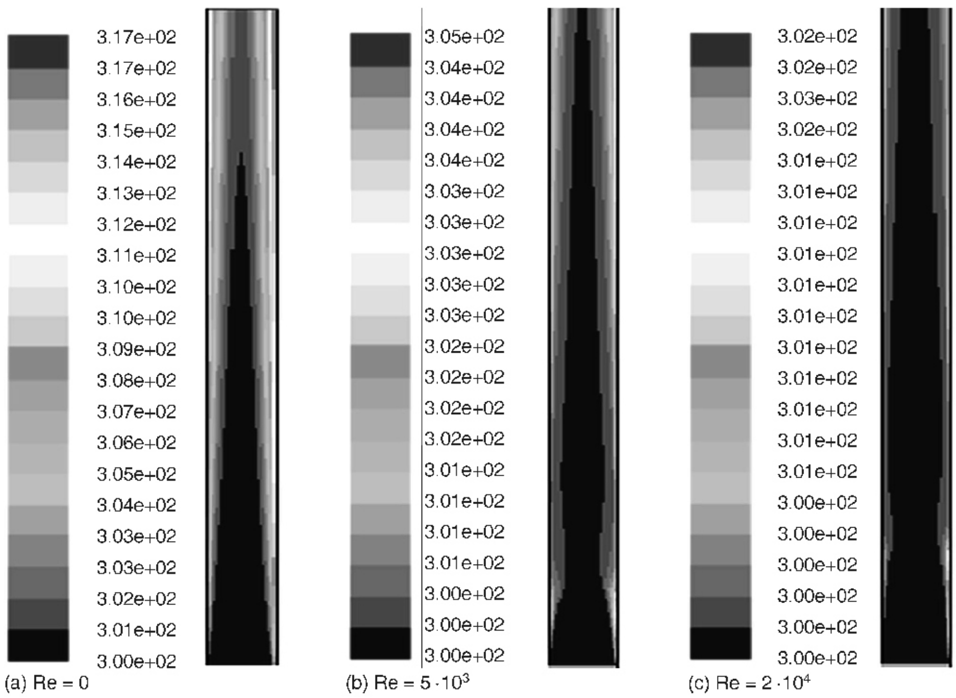


**Figure 4. Temperature development in the channel [K]; Ra = 9·$10^4$**

Figure 4 shows the evolution of the channel temperature for different Reynolds number. There is no big difference in the temperature evolution especially in the centre of the channel. The major difference is seen at the vicinity of the wall, we notice that all along the walls the temperature decreases for ascending Reynolds number (fig. 5), because the drive of ambient air by the jet is more and more significant, which also explains why the cooling is observed as of the entry of the channel. The jet interacts with the walls more and more far from the nozzle as the Reynolds number increases. This impact causes the formation of a swirl with weak intensity because it opposes to buoyancy forces, particularly close to the heated walls. When the Reynolds number increases, the maximum cooling region – in which wall temperatures are lower – moves towards the exit section of the channel.

Figure 6 presents the corresponding local Nusselt numbers. We notice that an increase of the inlet Reynolds number generates a more intense heat exchange. This is a predictable result since a high Reynolds number implies a more significant drive of the surrounding fluid and, consequently, a higher wall to fluid temperature gradient. We also observe that the Nusselt number decreases with the $X$ distance for all the inlet conditions tested.

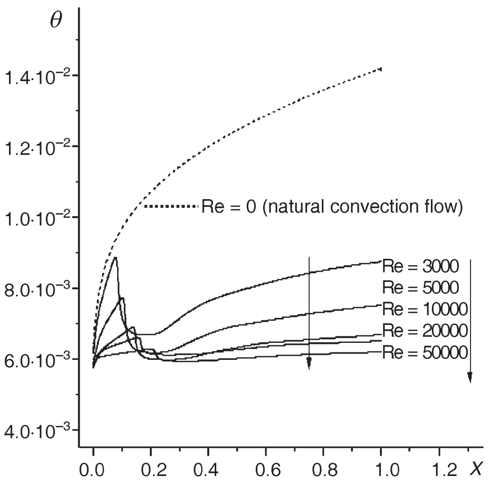


**Figure 5. Dimensionless wall temperature for various Reynolds number; Ra = 9 $10^4$**

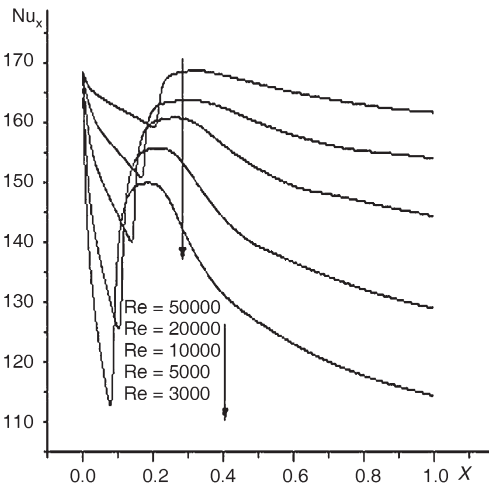


**Figure 6. Local Nusselt number of the wall for various Reynolds number; Ra = 9 $10^4$**

### *Case 2 – separated boundary layers*

Last flow behaviour for the case of separated boundary layers is compared to the case of attached one. For the second case the problem was simulated with $H/e$ = 12.5 and a constant heat flux $\phi$ = 2 $10^4$ W/m$^2$, which corresponds to a Rayleigh number equal to Ra ~2 $10^7$. The Prandtl number is 0.71 (air).

For high Rayleigh numbers the thickness of boundary layers decreases symmetrically with an increasing value of the x-component velocity. This result can be seen in fig. 7(a).

The evolution of the mass flow rate in both side of the jet nozzle is showed also in fig. 3. For highest Rayleigh number the evacuated flow is almost the same of the previous case (Ra = 9 $10^4$) the flow increases according to the relation (9). These results confirm that

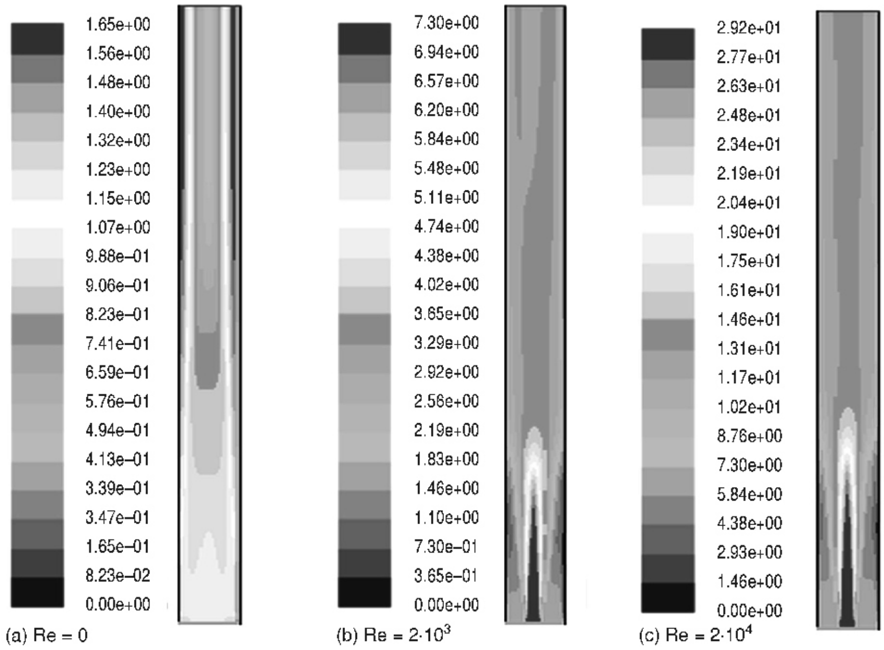


**Figure 7. Velocity development in the channel [ms$^{-1}$]; Ra = 2 10$^7$**

this flow don't result from the natural convection flow but in a large majority from the drive caused by the surrounding fluid in the vicinity of the entry.

The channel temperature for different Reynolds numbers is depicted in fig. 8 for Re = 0. The corresponding temperature is too high near the wall. When Re increases, a small cell appears down of the jet, fig. 8(b)-(c). Its size increases with increasing Re and cause the formation of hot zones. These zones coordinate increases for ascending Reynolds number values.

We represent on fig. 9 the dimensionless wall temperature for various Reynolds numbers ranging from 0 – free convection (dotted line) – to 5 10$^4$. The influence of the jet on the wall temperature is noticeable especially for high Rayleigh numbers, because a higher wall heat flux yields to a more significant variation of the wall temperatures, especially in the region where the jet interacts with the wall. Just in the lower part of this zone, the recirculation flow minimizes the heat exchange.

Figure 10 presents the corresponding local Nusselt numbers. We notice that an increase of the inlet Reynolds number generates a more intense heat exchange. This is a predictable result since a high Reynolds number implies a more significant drive of the surrounding fluid and, consequently, a higher wall temperature gradient. We also observe that the Nusselt number decreases with the $X$ distance for all the inlet conditions tested.

Increasing the Reynolds number enhances the Nusselt number. This effect is more significant at Ra = $2 \cdot 10^7$, for which the effect of the jet is more important on the temperature profile. As soon as the jet operates, the local Nusselt number increases according to the Reynolds number; which can be explained by the fact that the thickness of the boundary layer, which acts as a heat insulator, decreases when the Reynolds number increases. It follows that the convective exchange between the flow and the heated plate increases (fig.10).

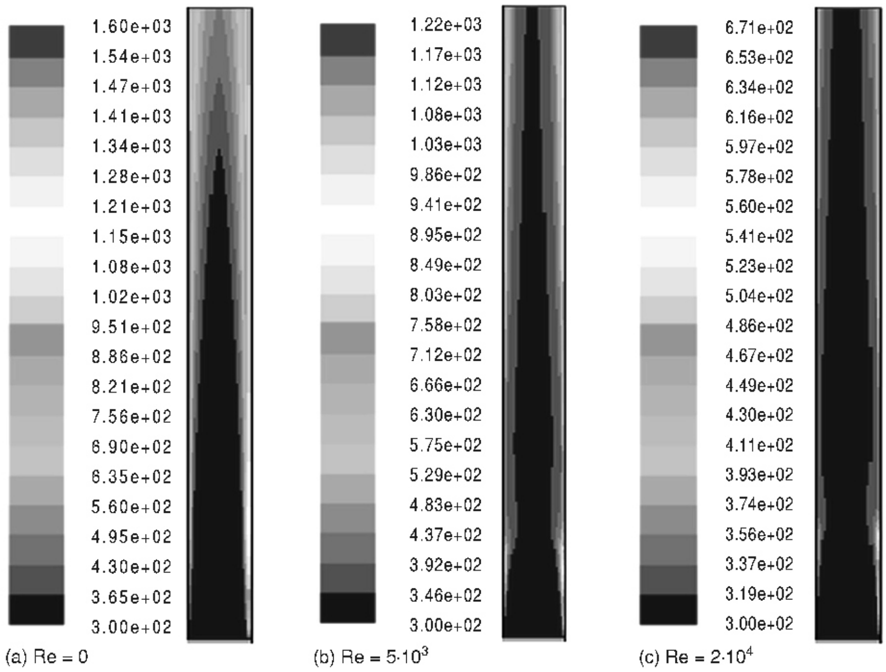


**Figure 8. Temperature development in the channel [K]; Ra = 2 10$^7$**

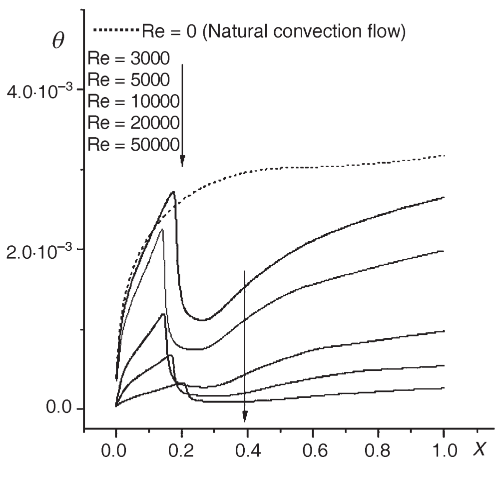


**Figure 9. Dimensionless wall temperature for various Reynolds number; Ra = 2 10$^7$**

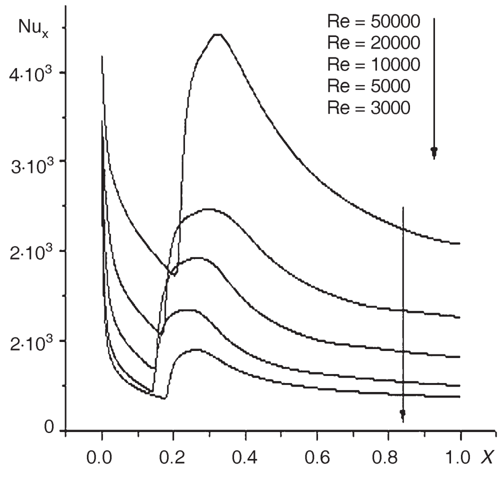


**Figure 10. Local wall Nusselt number for various Reynolds number; Ra = 2 10$^7$**

We notice for the two Rayleigh number treated the presence of hot point for which the heat transfer is minimal. These zones are due to the recirculation flow behind the jet impact zone. The maximum heat exchange located at the jet impact zone increases according to the Reynolds number.

## Conclusions

A numerical study of both natural and mixed convection in a 2-D-channel submitted to a constant wall heat flux was performed. The mixed flow is obtained by using an ascending jet located at the entry section of the channel. Special attention has been carried to the thermal behaviour of the flow, especially in the jet-wall interaction zone. Moreover, competition between natural and forced convection are focused; for the two studied cases separated and attached boundary layers. For Reynolds number Re < 3000 competition between forced and natural convection flows is noticeable; for Re > 3000 natural convection flow is overwhelmed by the forced convection flow. The drive of ambient air by the jet enhances the heat transfer: the vertical jet at the entrance allows a good ventilation of the high part of the cavity and then it's favourable to heat exchange from the cavity towards the exterior.

## Nomenclature

- $b$ – width of the nozzle, [m]
- $E$ – dimensionless rate of dissipation of turbulent kinetic energy, [–]
- $e$ – width of the channel, [m]
- Fr – Froude number $[u_0^2/g\beta(T_0 - T_\infty)b]$, [–]
- Gr – Grashof number ( $g\beta\varphi H^4/\nu^2\lambda$), [–]
- g – gravitational acceleration, [ms$^{-2}$]
- $H$ – length of the channel, [m]
- $h$ – local heat transfer coefficient, [Wm$^{-2}$K$^{-1}$]
- $K$ – dimensionless turbulent kinetic energy, [–]
- $k$ – turbulent kinetic energy, [m$^2$s$^{-2}$]
- $M$ – dimensionless mass flow rate at the inlet section of the channel ($=2\dot{m}_1/(e - b)\alpha\rho$), [–]
- Nu – Nusselt number (= $hx/\lambda$ , [–]
- $P$ – dimensionless pressure, [–]
- $p$ – pressure, [Pa]
- Pr – Prandtl number (= $\mu c_p/\lambda$), [–]
- $Q_1$ – dimensionless mass flow rate at the inlet section of the channel (= $\dot{m}_1/(e - b)\alpha\rho$, [–]
- $Q_2$ – dimensionless mass flow rate at the exit section of the nozzle, [–]
- Ra – Rayleigh number (= $g\beta$ $\varphi/\lambda\alpha\nu$), [–]
- Re – Reynolds number (= $bu_0/\nu$), [–]
- $T$ – temperature, [K]
- $U, V$ – dimensionless velocity components along X and Y, respectively, [–]
- $u, v$ – velocity components along x-and y-axis, respectively, [ms$^{-1}$]
- $x, y$ – co-ordinates, [m]
- X, Y – dimensionless co-ordinates, [–]

*Greek symbols*

- $\alpha$ – thermal diffusivity, [m$^2$s$^{-1}$]
- $\varepsilon$ – turbulent kinetic energy dissipation, [m$^2$s$^{-3}$]
- $\Phi$ – wall heat flux, [Wm$^{-2}$]
- $\lambda$ – thermal conductivity, [Wm$^{-1}$K$^{-1}$]
- $\mu$ – dynamic viscosity, [kgm$^{-1}$s$^{-1}$]
- $\nu$ – kinematic viscosity, [m$^2$s$^{-1}$]
- $\rho$ – fluid density, [kgm$^{-3}$]
- $\theta$ – dimensionless temperature, [–]

*Subscripts*

- m – average
- p – wall value
- t – turbulent
- x – local value
- $\infty$ – external (ambient conditions value)

## References

[1] Metais, B., Eckert, E. R. G., Forced, Mixed and Free Convection Regimes, *Journal of Heat Transfer, 86* (1964), pp. 295-296
[2] Bar-Cohen, A., Rohsenow, W. M., Thermally Optimum Spacing of Vertical Natural Convection Cooled Parallel Plates, *Journal of Heat Transfer, 106* (1984), 1, pp. 116-123
[3] Betts, P. L., Dafa'nela, A. A., Turbulent Buoyant Air Flow in a Tall Rectangular Cavity, *Proceedings,* (ASME HTD) Winter Annual Meeting, New York, USA, 1986, Vol. 60, p. 83
[4] Cheung, F. B., Sohn, D. Y., Numerical Study of Turbulent Natural Convection in An Innovative Air Cooling System, *Numerical Heat Transfer, Part A , 16* (1989), 4, pp. 467-487

[5] Gebhart, B., *et al*., Buoyancy-Induced Flows and Transport, Hemisphere Publ. Corp., Washington, DC, 1987
[6] Hugot, G., Study of Laminar Natural Convection between Two Isothermal Vertical Parallel Plates (in French), *Entropie, 46* (1972), pp. 55-66
[7] Miyamoto, M., *et al*., Turbulent Free Convection Heat Transfer from Vertical Parallel Plates, in: Heat Transfer (Eds. C. L. Tien, V. P. Carey, J. K. Ferrell) Hemisphere Publ. Corp., Washington DC, 1986, Vol. 4., pp. 1593-l598
[8] Auletta, A., Manca, O., Heat and Fluid Flow Resulting from the Chimney Effect in a Symmetrically Heated Vertical Channel with Adiabatic Extensions, *International Journal of Thermal Sciences, 41* (2002), 12, pp. 1101-1111
[9] Korbut, V. P., Paladenko, Yu. V., Turbulent Natural Convection Flow and Heat Transfer in a Fat Vertical Slot with Asymmetrically Heated Walls, *Heat Transfer Research, 25* (1993), pp. 313-320
[10] Fedorov, A. G., Viskanta, R., Turbulent Natural Convection Heat Transfer in an Asymmetrically Heated Vertical Parallel Plate Channel, *International Journal of Heat Mass Transfer, 40* (1997), 16, pp. 3849-3860
[11] Versteegh, T. A. M., Nieuwstadt, F. T. M., Turbulent Budgets of Natural Convection in an Infinite, Differentially Heated, Vertical Channel, *International Journal of Heat and Fluid Flow, 19* (1998), 2, pp. 135-149
[12] Versteegh, T. A. M., Nieuwstadt, F. T. M., A Direct Numerical Simulation of Natural Convection Between Two Infinite Vertical Differentially Heated Walls Scaling Laws and Wall Functions, *International Journal of Heat and Mass Transfer, 42* (1999), 19, pp. 3673-3693
[13] Elenbas, W., Heat Dissipation of Parallel Plates by Free Convection, *Physica, 9* (1942), 1, pp. 1-28
[14] Dalbert, A. M., Penot, F., Peube, J. L., Laminar Natural Convection in a Vertical Heated Channel (in French), *International Journal of Heat and Mass Transfer, 24* (1981), 9, pp. 1463-1473
[15] Penot, F., Dalbert, A. M., Mixed Convection and Forced into a Vertical Thermosyphon (in French), *International Journal of Heat and Mass Transfer, 26* (1983), 11, pp. 1639-1647
[16] Kheireddine, A. S., *et al*., Numerical Prediction of Pressure Loss Coefficient and Induced Mass Flux for Laminar Natural Convective Flow in a Vertical Channel, *Energy, 22* (1997), 4, pp. 413-423
[17] Desrayaud, G., Fichera, A., Laminar Natural Convection in a Vertical Isothermal Channel with Symmetric Surface-Mounted Rectangular Ribs, *International Journal of Heat and Fluid Flow, 23* (2002), 4, pp. 519-529
[18] Ben Maad, R., Natural Convection in Plane Wall with Prandtl High Numbers (in French), Ph. D. thesis, University of Pierre et Marie Curie, Paris VI, France, 1979
[19] Najam, M., *et. al*., Numerical Study of Mixed Convection in a T Form Cavity Subjected to a Constant Heat Flux and Ventilated by a Vertical Air Jet (in French), *Compte Rendu de Mecanique, 330* (2002), 7, pp. 461-467
[20] Jones, W. P., Launder, B. E., The Calculation of the Low Reynolds Number Phenomena with a Two-Equation Model of Turbulence, *International Journal of Heat and Mass Transfer, 16* (1973), 6, pp. 1119-1130

Paper submitted: November 19, 2008
Paper revised: April 5, 2009
Paper accepted: April 14, 2009