Mixed Convection in a 2D-channel with a Co-Flowing Fluid Injection: Influence of the Jet Position

Ameni Mokni, Hatem Mhiri, Georges Le Palec, And Philippe Bournot

Abstract—Numerical study of a plane jet occurring in a vertical heated channel is carried out. The aim is to explore the influence of the forced flow, issued from a flat nozzle located in the entry section of a channel, on the up-going fluid along the channel walls. The Reynolds number based on the nozzle width and the jet velocity ranges between 3 \(10^3\) and 2.\(10^5\); whereas, the Grashof number based on the channel length and the wall temperature difference is 2.57 \(10^{10}\). Computations are established for a symmetrically heated channel and various nozzle positions. The system of governing equations is solved with a finite volumes method. The obtained results show that the jet-wall interactions activate the heat transfer, the position variation modifies the heat transfer especially for low Reynolds numbers; the heat transfer is enhanced for the adjacent wall; however it is decreased for the opposite one. The numerical velocity and temperature fields are post-processed to compute the quantities of engineering interest such as the induced mass flow rate, and the Nusselt number along the plates.

Keywords—Channel, Heat flux, Jet, Mixed convection.

I. INTRODUCTION

Natural convection is unquestionably regarded as a very attractive mode of cooling because of its little cost, minimal maintenance and low noise [1]. Natural convection between heated vertical 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 very attractive for applications in electronic devices. However, in order to increase the cooling requirements, researches for methods to improve the heat transfer parameters or to analyze standard configurations to carry out optimal geometrical parameters for a better heat transfer rate [2]–[9]. Adding a forced convection flow is a solution to enhance heat transfer.

In many applications although forced convection heat transfer is involved, the effect of buoyancy is not negligible. Such flows are known as mixed convection flows. When the Reynolds number (Re) or the Grashof number (Gr) is high enough, the flow become turbulent.

Pioneering work on turbulent mixed convection particularly from vertical tubes has been done by Jackson and co-workers [10]–[12]. Nakajima et al. [13] studied the effect of buoyancy on the turbulent transport processes in mixed convection for both aiding and opposing flows. Correlations for dimensionless mass flow rate, maximum wall temperature and average Nusselt number, in terms of Rayleigh number and dimensionless geometric parameters are presented by several authors in order to compute the quantities of engineering interest [14]–[21]: we can cite Penot and al. [20] who proposed useful correlations to determine the flow rate, the fluid temperature, and the Nusselt number according to the heat flux density, the pressure difference and the Reynolds and Grashof numbers. M. Najam et al. [21] 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 asymmetrically heated vertical channel submitted to a vertical jet of fresh air entering by the bottom. Numerical results are presented in terms of dimensionless induced mass flow rates and dimensionless wall temperatures for a Rayleigh number \(Ra=2.57 \times 10^{10}\) based on the heat transfer and the channel length and different Reynolds numbers. Moreover, Nusselt numbers varying with the dimensionless axial coordinate \(X\) pave the way for the calculation of the average Nusselt numbers. Computations are established for a symmetrically heated channel and various jet positions. Some optimal geometrical configurations have been identified corresponding to the maximum average Nusselt number.

II. ASSUMPTIONS AND GOVERNING EQUATIONS

A computational domain of finite dimension illustrated in Fig. 1 is employed to simulate the vertical channel. A gas jet is issued from a flat nozzle located at various position at the bottom of the channel. The chimney walls are subject to a constant heat flux. Numerical results are reported for dry air
as coolant. The unconfined airflow lies far away from the region of the disturbance induced by the presence of the jet flow. 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-ε turbulence model. Mixed convection is considered by using the Boussinesq approximation in which the density varies linearly with temperature. Other thermo-physical quantities are assumed to be constant. The flow is assumed steady and incompressible.

Fig.1 Coordinate system of the configuration flow

Dimensionless variables are defined by:

$$X = \frac{x}{L}, \quad Y = \frac{y}{L}, \quad U = \frac{u}{u_0}, \quad V = \frac{v}{u_0}, \quad P = \frac{(p + \rho g x) \epsilon^2}{\rho \alpha},$$

$$\theta = T - T_\infty, \quad K = \frac{ke^2}{\alpha^2}, \quad E = \frac{\alpha^2}{\alpha},$$

(1)

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

Continuity equation:

$$\frac{\partial U}{\partial X} + \frac{\partial V}{\partial Y} = 0$$

(2)

Momentum equation in X direction:

$$\frac{\partial U}{\partial X} + V \frac{\partial U}{\partial Y} = \frac{\partial P}{\partial X} + (\text{Pr} + \text{Pr}_f) \left[ \frac{\partial^2 U}{\partial X^2} + \frac{\partial^2 U}{\partial Y^2} \right] - \frac{2}{3} \frac{\partial K}{\partial X} + Ra \Pr \theta$$

(3)

Momentum equation in Y direction:

$$\frac{\partial V}{\partial X} + U \frac{\partial V}{\partial Y} = \frac{\partial P}{\partial Y} + (\text{Pr} + \text{Pr}_f) \left[ \frac{\partial^2 V}{\partial X^2} + \frac{\partial^2 V}{\partial Y^2} \right] - \frac{2}{3} \frac{\partial K}{\partial Y}$$

(4)

Energy equation:

$$\frac{\partial T}{\partial X} + U \frac{\partial T}{\partial Y} = \frac{\partial}{\partial Y} \left[ \frac{\partial T}{\partial Y} \right] - \frac{\partial}{\partial Y} \left[ \frac{\partial (\frac{U}{\alpha} + \frac{V}{\alpha})}{\partial Y} \right] + \frac{\partial}{\partial X} \left[ \frac{\partial (\frac{U}{\alpha} + \frac{V}{\alpha})}{\partial X} \right]$$

(5)
\[ U \frac{\partial \theta}{\partial X} + V \frac{\partial \theta}{\partial Y} + \frac{2}{\partial X^2} \left( \frac{\partial \theta}{\partial Y} \right) \]

Turbulent kinetic energy equation:

\[ U \frac{\partial K}{\partial X} + V \frac{\partial K}{\partial Y} = \left( \frac{\partial U}{\partial X} \right)^2 + \frac{2}{\partial X^2} \left( \frac{\partial K}{\partial Y} \right) - E + G_{DK} + G_{DB} \]

Rate of dissipation of turbulent kinetic energy equation:

\[ U \frac{\partial E}{\partial X} + V \frac{\partial E}{\partial Y} = \left( \frac{\partial U}{\partial X} \right)^2 + \frac{2}{\partial X^2} \left( \frac{\partial E}{\partial Y} \right) - C_1 E \frac{J}{K} + C_2 \frac{E^2}{K} \]

where:

\[ K_{DK} = Pr_1 \left( \frac{\partial U}{\partial Y} + \frac{\partial U}{\partial Y} \right) \frac{\partial U}{\partial Y} \frac{2}{3} K \frac{\partial U}{\partial Y} \]

and

\[ G_{DB} = \frac{1}{Fr} \frac{\partial \theta}{\partial X} \]

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

The standard k-ε model is used, so that constants are those given by Jones and Lauder [21]: \( C_1 = 1.44 \); \( C_2 = 1.92 \), \( C_3 = 0.7 \), \( C_\mu = 0.09 \), \( \sigma_\epsilon = 1.0 \), \( \sigma_\mu = 1.30 \), \( Pr_1 = 1.0 \).

The boundary conditions are the following:

At \( X = 1 \):

\[ \frac{\partial U}{\partial X} = 0, \frac{\partial V}{\partial X} = 0, P = 0, K = \frac{3}{2} \theta \]

And

\[ Q_1 = \int_{b/2L}^{b/2L} UdY \]

\[ Q_2 = \int_{-b/2L}^{-b/2L} UdY \]

\( \nu_1 \) is the turbulence intensity.

III. NUMERICAL RESULTS

The Results of the present investigation are carried out for air, \( Pr = 0.71 \), the Rayleigh number based on the channel length range is \( 2.57 \times 10^5 \), the Reynolds number based on the jet velocity ranges between \( 3 \times 10^3 \) and \( 2 \times 10^4 \). We study the influence of the velocity and the position of the jet on the heat transfer enhancement. Computations deals with a symmetrically heated channel.

For purpose of validation, the computations were performed first for a simple channel - i.e. without gas injection from the nozzle - and numerical results were compared with the experimental ones published by A. Auletta et al. [22]. The local Nusselt defined by

\[ Nu = \frac{\phi(x) x}{(T_x - T_{in})} \]

is reported in figure 2. The resulting free convection problem was simulated with \( L/e = 2.5 \) which corresponds to a Rayleigh number equal to \( 1.16 \times 10^{11} \). The Prandtl number is 0.71 (air).

![Fig. 2. Local Nusselt number variation according to the local Rayleigh number. Natural convection flow Comparison with experiments.](image-url)

Differences between measurements and numerical predictions are very low and they mainly concern the highest part of the channel. This may be due to the fact that there were some heat losses caused by insufficient thermal insulation in...
experiments. Moreover, the imposed wall heat flux was not uniform because of the space between two successive heaters. At least, the use of thermocouples also modifies the flow structure, especially in the vicinity of the plates.

The study is carried out for a constant channel geometrical characteristics \( L/e = 12.5 \). We notice, for this aspect ratio, a linear increase of the local wall Nusselt number according to the relation:

\[
Nu = 0.76 Ra_x^{0.19} 
\]

(10)

A. Mass flow rate

Note that the introduction of the disturbance at the channel entrance activates the convection flow; the natural convection flow increases according the jet velocity; indeed the natural convection flow is higher in presence of the jet. The buoyancy influence is noticeable for low Reynolds numbers, so it has a significant impact on the driven flow. In contrast to height Reynolds numbers, the flow is mainly driven by the mean pressure gradient, so it increases according the jet velocity. The natural convection flow is independent also, of the nozzle position for low values of Reynolds number. A notable dependence on the nozzle position starts to be seen gradually from \( Re = 5000 \). The difference is noted between the central position and the two others. Induced mass flow rate is almost the same for the two shifted positions.

The mass flow rate on both sides of the jet involves according to the following correlation for a jet Reynolds number \( Re > 3 \times 10^5 \):

For central position:

\[
Q = 2 Q_1 = -11158 + 30.1 Re 
\]

(11)

For shifted positions:

\[
Q = 2 Q_1 = -9366 + 33.6 Re 
\]

(12)

The flow structure is, essentially, composed of the open lines, which represent the forced flow, and closed cells which are due to the recirculation movement up of the jet or to natural convection phenomena (Fig. 4.). The thermal gradient which exists between the fluid and the hot walls will cause a vertical aspiration of air with a non-negligible mass flow rate. The vertical jet of fresh air entering by the bottom of the heated channel interacts with the heated walls. As the Reynolds number increases, the jet interacts with the walls more and more far from the nozzle.

This impact causes, in addition of the coldest zones; the formation of a rotating cell with weak intensity behind the impact zone, because it opposes to buoyancy forces. These rotating cells do not support heat transfer and generate the formation of hot zones.

These zones move towards the exit section of the channel. For the two shifted positions the symmetry breaks down, the jet interacts with the opposite wall more and more far from the nozzle. The swirling size increases and constrains the jet of forced flow to pass close to this plane. In the other hand, the jet pass close to the adjacent wall; more the jet is closer to the wall, more the impact zone is closer to the entrance section, more it penetrates into the cavity.

B. Skin friction

The skin friction coefficient \( C_f = \frac{2 \tau_w}{\rho U^2} \) is plotted on figure 5.

For all the treated cases, note that the two walls behave in the same manner for the central position; however they operate in an opposite way for the two shifted positions.

The skin friction decreases from the leading edge. A peak is noticed, further, located on the impact zone. The skin friction maximum on the adjacent wall, correspond to the minimum on the opposite wall and vice versa.
D. Average Nusselt number

In order to better quantify the total heat transfer between the channel and the flow, let us introduce the average Nusselt number, defined as:

$$\bar{Nu} = \frac{\bar{h}L}{\lambda}$$

where

$$\bar{h} = \frac{1}{L} \int_0^L h_x dx = \frac{1}{L} \int_0^L \frac{\phi}{T_p(x) - T_w} dx$$

The average Nusselt number is plotted according the jet Reynolds number on Figure 7. We notice that the average Nusselt number increases according to the Reynolds number for both channel walls, which can to be explained by 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 walls increases. Fig. 7 shows also a little increase of the channel Nusselt number according to shifted position for low Reynolds numbers. Indeed for these cases the heat transfer enhancement of the adjacent plate is larger than the decrease of the opposite one. A weak increase of the total channel Nusselt number is noted.

For high Reynolds numbers the average heat transfer is...
unchanged even when we change the nozzle position. Predictable results since the heat transfer is almost the same for high Reynolds numbers (Fig.6.b). This can be explained by the fact that the induced mass flow rate ensures the maximum exchange.

![Graph showing evolution of the average Nusselt number](image)

**IV. CONCLUSION**

A numerical study of both natural and mixed convection in a 2D-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. Computations were performed for \( Ra=2.57 \times 10^{10} \), Reynolds number ranging between 3 103 and 2104 and three nozzle positions. The numerical procedure was validated by comparing our results with the experimental ones by Auletta [22]. Special attention has been carried to the thermal behaviour of the flow, especially the influence of the jet position on the heat transfer. The mass flow rate induced increases according to the Reynolds number. The nozzle position affects the induced mass flow rate for important Reynolds numbers. The difference is noted only between the central position and the two others shifted.

The vertical jet at the entrance permits good ventilation the cavity and then it’s favourable to heat exchange from the cavity towards the exterior. It was shown that for high Reynolds number, the flow field is mainly controlled by the external flow and heat transfer is almost similar for all the considered positions. The mean influence concerns the lowest Reynolds numbers: The heat transfer is enhanced for the adjacent wall; however it is decreased for the opposite one. The total Nusselt number is weakly increased.

**REFERENCES**


