

# Numerical Analysis of Laminar Mixed Convection within a Complex Geometry

Y. Lasbet, A. L. Boukhalkhal, K. Loubar

**Abstract**—The study of mixed convection is, usually, focused on the straight channels in which the onset of the mixed convection is well defined as function of the ratio between Grashof number and Reynolds number,  $Gr/Re$ . This is not the case for a complex channel wherein the mixed convection is not sufficiently examined in the literature. Our paper focuses on the study of the mixed convection in a complex geometry in which our main contribution reveals that the critical value of the ratio  $Gr/Re$  for the onset of the mixed convection increases highly in the type of geometry contrary to the straight channel. Furthermore, the accentuated secondary flow in this geometry prevents the thermal stratification in the flow and consequently the buoyancy driven becomes negligible. To perform these objectives, a numerical study in complex geometry for several values of the ratio  $Gr/Re$  with prescribed wall heat flux (H2), was realized by using the CFD code.

**Keywords**—Complex geometry, heat transfer, laminar flow, mixed convection, Nusselt number.

## I. INTRODUCTION

HEAT transfer modes are: conduction, convection and radiation. Convection is the process in which the heat transfer is due to a flowing fluid. It is classified into three forms: forced convection, free convection and mixed convection. The motion in forced convection is caused by external forces like a pumps or fans. In the natural convection, the motion is due only to the difference of temperature between the wall and the fluid. Mixed convection is the combined flow between the forced convection and the natural convection. It occurs when both forced and natural (free) convection mechanisms significantly and concurrently contribute to the heat transfer. Extensive works have been carried out to study the mixed convection through ducts of various geometries. Most of these works have been focused to the mixed convective heat transfer study in straight ducts. In the straight channels, the main parameter with which it can be characterize the mixed convection is the ratio between the Grashof number,  $Gr$ , and the Reynolds number,  $Re$ . In the literature, the value of the ratio  $Gr/Re$  corresponds to the onset of the mixed convection regime in the straight channel is well determined and the kinematic of the flow is well characterized, [1], [2]. Elliott et al. [1] have studied

numerically the mixed convection within the horizontal duct with constant wall temperature. The authors reported that the buoyancy driven fluid motion within the duct in the vertical direction dependent upon the Grashof and Reynolds numbers as well as the ratio of the duct height to its width. Barletta et al [2] have performed analytically the mixed convection in vertical rectangular duct with a prescribed uniform wall heat flux (H2 boundary condition). The authors showed that the buoyancy and the Nusselt number are affected by the ratio  $Gr/Re$  and the aspect ratio. However, it has been seen from the literature that the analysis of mixed convective heat transfer in complex duct has been not examined well. So, the main contribution of this paper is to investigate numerically by using CFD computational the mixed convection in complex geometry with constant wall heat flux conditions (H2 boundary conditions).

In this study, authors analyzed numerically the heat transfer and three-dimensional flow behavior of mixed convection, and evaluated thermal performance using Nusselt number and the intensity of the secondary flow with and without buoyancy induced flow. For this aim, we solved the conservation equations (NS equations and the energy equation) and the calculations were performed for the following values of the ratio  $Gr/Re$  (0, -100, 100, 200, and 300). This paper is organized as follows: Section II presents the geometry considered and the numerical procedure adopted. Section III presents the results and discussions and Section IV is devoted to the conclusions and future work.

## II. DESCRIPTION OF THE GEOMETRY AND NUMERICAL METHOD

Fig. 1 presents the basic element of the C-shaped geometry. It, initially introduced by [3], generates spatially chaotic flows as shown by the work of [4]. The channel cross-section is square with an aspect ratio of 1 (1 cm  $\times$  1 cm). The hydraulic diameter  $D_h$  is 1 cm. The unfolded length of a period is equal to 9 cm.

The mass conservation, Navier–Stokes equations and Energy equation are numerically solved by using the commercial CFD code Fluent©. The fluid is considered incompressible and the flow is steady and laminar.

The applied boundary conditions are:

- at the inlet, uniform velocity profile.
- at solid walls, no-slip conditions and a uniform wall heat surface flux.
- at the outlet, the outflow condition is considered.

Y. Lasbetis with the Laboratory LDMM, Ziane Achour University, Route Moudjbara BP 3117 Djelfa, Algeria (corresponding author, phone: +213552686134; e-mail: peeryahia@yahoo.fr).

A. L. Boukhalkhal is with the Laboratory LDMM, Ziane Achour University, Route Moudjbara BP 3117 Djelfa, Algeria (e-mail: lamine\_aa@univ-djelfa.dz).

K. Loubaris with the GEPEA, UMR 6144, Ecole des Mines de Nantes, France (e-mail: khaled.loubar@mines-nantes.fr).

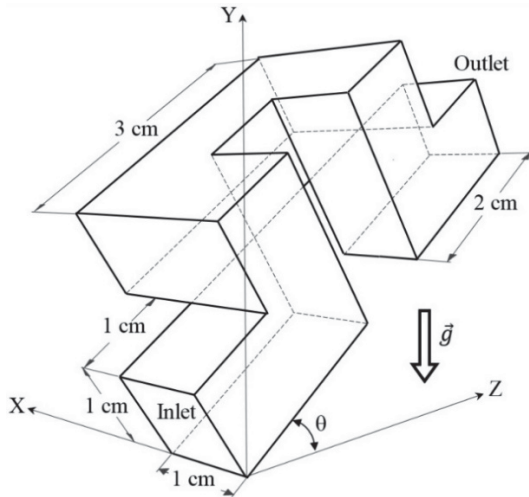


Fig. 1 C-shaped geometry

A parametric study was performed to identify the grid mesh that provides the greatest resolution close to the walls where fluid velocity and temperature gradients are the highest. A spatial resolution of  $40 \times 40$  meshes in the cross-section and 40 meshes in 1 cm in the z-direction was found fully satisfactory. The convergence of computations was stopped when the residues of the conservation equation were less than  $10^{-6}$ .

In this paper, we calculate the evolution of the Nusselt number and the intensity of the secondary flow with the curvilinear coordinate in the C-shaped geometry in mixed convection regime (horizontal and vertical orientations) for several values of the ratio Gr/Re number (Gr/Re: 0, 100, 200, and 300). The local Nusselt number is given by:

$$Nu_s = \frac{q_w}{T_w - T_b} \frac{D_h}{\lambda} \quad (1)$$

where  $T_b$  is the bulk mean fluid temperature over the cross section area of the channel,  $T_w$  is the wall temperature over the same cross section area,  $D_h$  is the hydraulic diameter,  $q_w$  is the surface heat flux and  $\lambda$  is the thermal conductivity of the fluid (water,  $\lambda = 0.6 \text{ W m}^{-1} \text{ K}^{-1}$ ).

The mean Nusselt number is calculated by:

$$Nu_m = \frac{1}{S} \int_0^S Nu_s ds \quad (2)$$

where  $s$  is the curvilinear coordinate along the channel and  $S$  is the unfolded length of one period and it is equal to 9 cm.

The intensity of the secondary flow is defined as:

$$I = \frac{(\sqrt{v^2 + w^2})_{\max}}{u_m} \quad (3)$$

where  $v$  and  $w$  are the transversal component velocities over the cross section area of the channel and  $u_m$  is the mean velocity.

Finally, the momentum flux correction factor,  $K_d$ , and the kinetic energy correction factor,  $K_e$ , as defined in [5] are given by:

$$K_d = \frac{1}{S} \int_0^a dx \int_0^b \left( \frac{u}{u_m} \right)^2 dy \quad (4)$$

$$K_e = \frac{1}{S} \int_0^a dx \int_0^b \left( \frac{u}{u_m} \right)^3 dy \quad (5)$$

The nondimensional parameters that characterize the flow regime are the classical Reynolds number, Re, and the Grashof number, Gr, respectively defined by (6) and (7):

$$Re = \frac{u_m D_h}{\nu} \quad (6)$$

$$Gr = \frac{g \beta \Delta T D_h^3}{\nu^2} \quad (7)$$

where  $\Delta T$  is defined as:

$$\Delta T = \frac{q_w D_h}{\lambda} \quad (8)$$

The ratio between the Grashof number and the Reynolds number is:

$$\frac{Gr}{Re} = \frac{g \beta \Delta T D_h^2}{\nu u_m} \quad (9)$$

### III. VALIDATION OF THE CFD CODE

In this section, we solve numerically by using computational CFD code (Fluent) the governing equations for mixed convection in a vertical square duct with prescribed uniform wall heat flux (H2 boundary condition) for several values of the ratio between the Grashof number, Gr, and the Reynolds number, Re, (Gr/Re = -100, 0, 100, 200, and 300). We will compare our solution with those provided in literature in order to check the reliability and the precision of the CFD computation.

Let us consider a flow of a Newtonian fluid (water) in straight channel with square cross section (1 cm x 1 cm), see Fig. 2.

We assume that the flow is steady and laminar. The viscous dissipation in the fluid is neglected. All proprieties of the fluid are treated as constants and are equal to those of the water (the thermal conductivity  $\lambda$ , the thermal diffusivity and the dynamic viscosity  $\mu$ ). Except for the mass density for which

the Boussinesq approximation is employed and is given by:

$$\rho = \rho_0 (1 - \beta (T - T_0)) \quad (10)$$

where  $\beta$  is the volumetric coefficient of thermal expansion of the work fluid (water) and  $T_0$  is the reference temperature. In this study, we chose  $T_0$  identical to the inlet section temperature. We imposed at the inlet a uniform velocity profile and at the outlet, the outflow condition is considered. At the solid walls, no-slip conditions and a uniform wall heat surface flux are imposed.

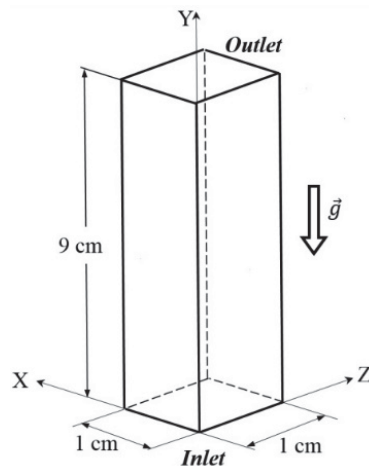


Fig. 2 Straight Channel

Table I presents a comparison of the values of the Nusselt number obtained in present paper, [2], [5] Morini [6] for the case of forced convection ( $Gr/Re=0$ ).

TABLE I  
VALUES OF THE NUSSLETT NUMBER IN SQUARE DUCT IN FORCED CONVECTION REGIME ( $Gr/Re=0$ )

	Present paper	Barletta [2]	Shah and London [5]	Spiga and Morini [6]
Nusselt number, Nu	3.088	3.0874	3.091	3.091

TABLE II  
VALUES OF THE NUSSLETT NUMBER IN SQUARE DUCT IN MIXED CONVECTION REGIME FOR THE VALUES OF THE RATIO  $Gr/Re$  RANGING FROM -100 TO 300 OBTAINED BY PRESENT PAPER AND [2]

	Nu		$K_d$		$K_e$	
$Gr/Re$	Present paper	Barletta [2]	Present paper	Barletta [2]	Present paper	Barletta [2]
-100	2.731	2.743	1.547	1.547	2.795	2.795
100	3.427	3.423	1.266	1.273	1.7609	1.778
200	3.748	3.63	1.204	1.204	1.549	1.550
300	4.055	4.156	1.161	1.171	1.424	1.449

Table I shows a good agreement between the values of the Nusselt number in present paper and that calculated by the others authors. Thereby, this result can be taken as a good validation of the CFD computations. Table II reports the values of the Nusselt number, the momentum flux correction factor,  $K_d$ , and the kinetic energy correction factor,  $K_e$ , for

several values of the ratio  $Gr/Re$  in the range  $-100 < Gr/Re < 300$ .

The values reported by the present paper and those obtained by [2] are similar which presents another argument of validation of CFD computational. We can see that there are a barely difference between our measures and those of [2] when the  $Gr/Re$  is large. This difference is due, first, to the method of resolution and, second, to the choice of the reference temperature. Our reported results for the Nusselt number were obtained by CFD numerical resolution while those of [2] were obtained analytically. The reference temperature in our simulations is that of the entrance section and it is considered as constant. However, [2] have chosen  $T_0$  as the mean fluid temperature in a cross-section, namely:

$$T_0 = \frac{1}{ab} \int_0^a dx \int_0^b T(x, y, z) dy \quad (11)$$

#### IV. PRESENTATION OF THE RESULTS

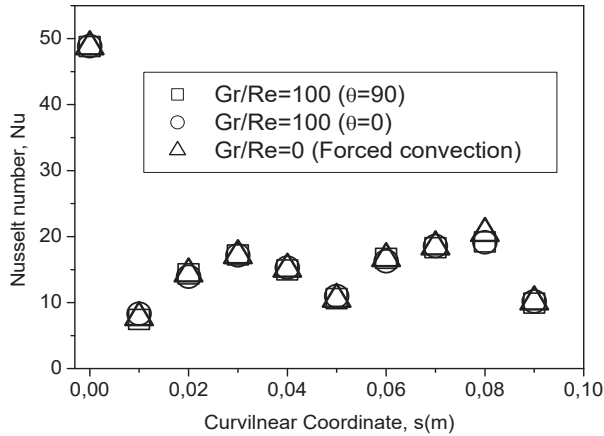
We present in this section the local Nusselt number and the intensity as defined in section (2) within the C-shaped geometry in one period. The evolutions of these parameters are given along the channel with the curvilinear coordinate. The Reynolds number is taken equal to 100 and the Grashof number has the following values: 0,  $10^4$ ,  $2 \cdot 10^4$  and  $3 \cdot 10^4$ . So, the ratio  $Gr/Re$  takes the values: 0, 100, 200, and 300.

Fig. 3 shows the evolutions of the local Nusselt number with the curvilinear coordinate within the C-shape geometry in mixed convection regime ( $\theta = 0^\circ, \theta = 90^\circ$ ) for the different values of the ratio  $Gr/Re$ .

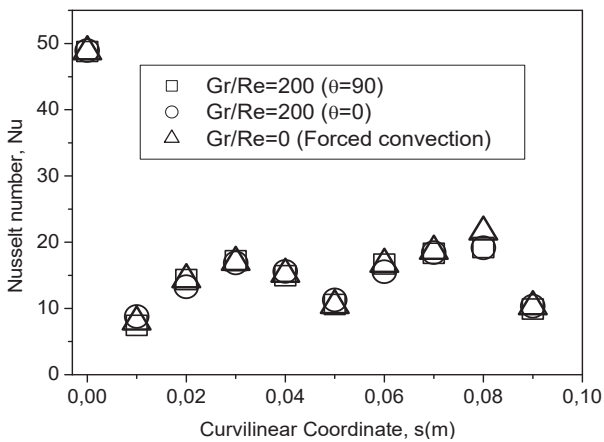
For all values of the ration  $Gr/Re$ , a fast decay of the local Nusselt number is observed in entry region ( $s < 0.01m$ ). After this decrease, fluctuated evolution of the local Nusselt number is observed with the curvilinear coordinate. The chaotic behavior of this parameter along the channel is due to the continued destruction of the boundary layer along the curvilinear coordinate which prevents the establishment of flow. This destruction highly contributes to the improvement of the heat exchange between the fluid and the wall and consequently the value of the Nusselt number enhances greatly.

The evolutions of the Nusselt number with the curvilinear coordinate in the both forced convection flow ( $Gr/Re=0$ ) and mixed convection flow (horizontal and vertical orientations) are perfectly similar whatever the ratio  $Gr / Re$ . This result shows that the natural convection effect is negligible comparing to the forced convection in the type of flow. This is explained by the fact that the secondary flow is very intense within the C-shaped geometry.

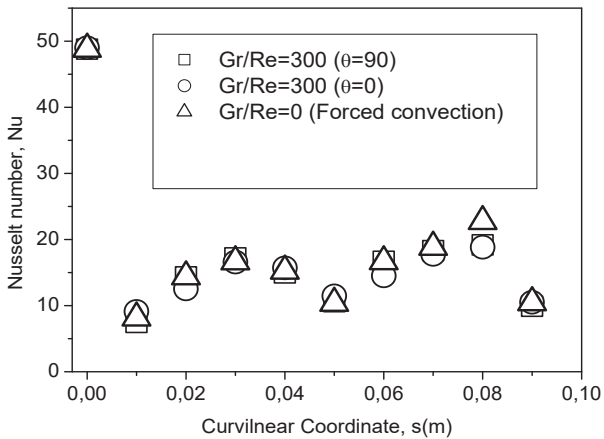
The mean value of the Nusselt number obtained for all simulations is around 16.45 and it is very noticeable compared to that calculated in the straight channel, see Table III. The maximum relative variation of the Nusselt number is about 10 in C-Shaped geometry for the three values of the ratio  $Gr/Re$ .



(a)



(b)



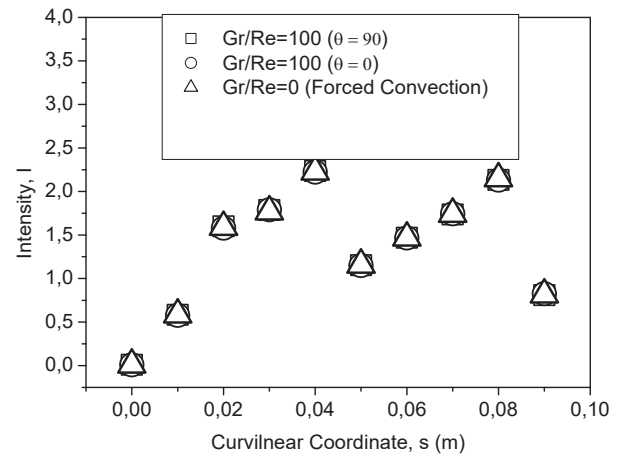
(c)

Fig. 3 Evolutions of the Nusselt number, Nu, versus the curvilinear coordinate, (a) Gr/Re = 100, (b) Gr/Re = 200, and (c) Gr/Re = 300.

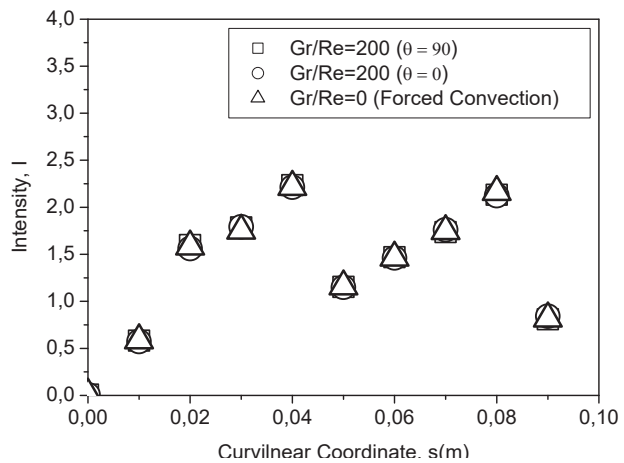
The improvement of the heat transfer is due to the fully erratic kinematic behavior as reported by our previous work [7]. This behavior can be quantified by calculating of the intensity which given in (3).

TABLE III  
COMPARISON OF THE NUSSELT NUMBERS OBTAINED IN BOTH COMPLEX GEOMETRY AND THE STRAIGHT CHANNEL FOR THE RATIO GR/RE EQUAL TO 100, 200, AND 300

	100	200	300
NuStraight-channel	3.427	3.748	4.055
NuC-shaped-geometry	16.45	16.45	16.45
Rate of Improvement $(N_{u_{C-Shape}} - N_{u_{Straight}}) \times 100$	380.01	338.90	305.67



(a)



(b)

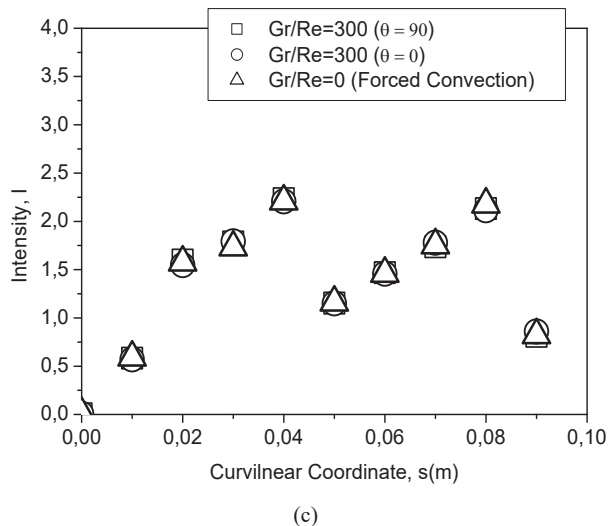


Fig. 4 Evolutions of the Intensity,  $I$ , versus the curvilinear coordinate, (a)  $Gr/Re = 100$ , (b)  $Gr/Re = 200$ , and (c)  $Gr/Re = 300$

Fig. 4 shows the evolutions of intensity with the curvilinear coordinate within the C-shape geometry in the mixed convection regime (horizontal and vertical orientations) for the values of the ratio  $Gr/Re$  ranging from 0 to 300. This parameter quantifies the rate of agitation in the fluid created by secondary flows in cross section of the C-shaped geometry. We notice that the intensity is not affected by the ratio  $Gr/Re$ . So, the evolutions of the intensity are almost identical for all cases. This result confirms, as previously shown, that the forced convection dominates the natural convection. In the others words, the secondary flows dues to the forced convection are most important compared to those created by the natural convection. To highlight the effect of natural convection in this geometry, we should increase significantly the ratio  $Gr/Re$ . It was observed that the onset of the mixed convection regime in the complex channel is delayed and it can be detected by a suitable selection of  $Gr/Re$  ratio ranges.

#### V.CONCLUSION

In this paper, we deal numerically by using CFD computational the mixed convection regime in complex channel called C-shaped in both horizontal and vertical orientations for which the walls are maintained at specified constant heat flux  $H_2$ . These simulations reveal that the important rate stirring in the C-shaped geometry prevents the thermal stratification in the flow. Consequently, the buoyancy driven is negligible compared to the forced convection across the range  $Gr/Re$  values studied. Furthermore, our study showed that the ratio,  $Gr/Re$ , correspond to the outbreak of the mixed convection in such geometry is very great compared to that found in the straight channel. This geometry obtains high rates of heat transfer and these results are pertinent to heat transfer improvement in laminar flow in mini-channel, [8].

#### REFERENCES

- [1] L. Elliott, D. B. Ingham and J. D. Wood, "Mixed Convection Flow of Newtonian and Non-Newtonian Fluids in a Horizontal Rectangular Duct," *Numerical Heat Transfer Part A*, vol. 32, pp. 831-860, 1997.
- [2] A. Barletta, E. Rossi di Schio and E. Zanchini, "Combined forced and free flow in a vertical rectangular duct with prescribed wall heat flux," *International Journal of Heat and Fluid Flow*, vol. 24, pp. 874-887, 2003.
- [3] H. Robin, A.M. Stremler, K.V. Sharp, M.G. Olsen, J.G. Santiago, R.J. Adrian, H. Aref and D.J. Beebe, "Passive mixing in a three-dimensional serpentine microchannel," *Journal of Micro electro mechanical Systems*, vol 9(2), 190-197, 2000.
- [4] D. Beebe, R. Adrian., M. Olsen, M. Stremler, H. Aref and B. Jo, "Passive mixing in micro-channels: fabrication and flow experiments," *Mécanique et Industries*, vol 2, 334-348, 2001.
- [5] R.K. Shah and A.L. London, "Laminar forced convection in ducts," *Academic Press*, New York, 1978.
- [6] M. Spiga and G.L. Morini, "Nusselt numbers in laminar flow for H2 boundary conditions," *Int. J. Heat Mass Transfer*, vol 39, 1165-1174, 1996.
- [7] Y. Lasbet, B. Auvity, C. Castelain and H. Peerhossaini, "Thermal and Hydrodynamic Performances of Chaotic Mini-Channel: Application to the Fuel Cell Cooling," *Heat Transfer Engineering*, Vol 28(8-9), 795-803, 2007.
- [8] Y. Lasbet, B. Auvity, C. Castelain and H. Peerhossaini, "A chaotic heat-exchanger for PEMFC cooling applications," *Journal of Power Sources*, Vol 156, 114-118, 2006.