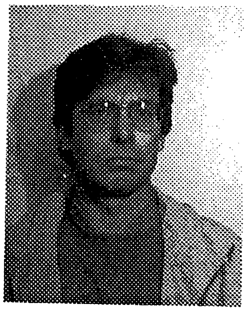
5th International Conference on Air Distribution in Rooms
ROOMVENT '96, July 17 - 19, 1996



NUMERICAL STUDY OF THE INFLUENCE OF INLET BOUNDARY CONDITIONS ON THE AIR MOVEMENT IN A VENTILATED ENCLOSURE

Joubert, P. (*); Sandu, A. (**); Béghein, C. (*); Allard, F. (*)

(*) LEPTAB, Université de La Rochelle, Av. Marillac, 17042 La Rochelle cédex 1, France
(**) Universitatea Tehnica de Constructii Bucuresti, Facultatea de Instalatii, B-dul Carol I, n°66, sector 2, RO-72323, Bucuresti, Romania

ABSTRACT
A numerical study of the influence of dynamic and turbulent inlet boundary conditions has been carried out for a forced convection flow in a two-dimensional enclosure. The turbulent flow is predicted via a k-ε model. In the first part, it is shown that the turbulent k and ε inlet boundary conditions do not significantly affect the flow pattern. The second part of this work deals with the influence of the inlet jet profile (plane or parabolic) on the flow. It is reported here that while the average flow is not greatly affected by the inlet profile, the fluctuating part of the flow is modified by the jet profile. If the inlet Reynolds number is kept constant, this phenomenon is amplified for an increasing inlet dimension.

KEYWORDS
Ventilated cavity, numerical simulations, turbulent inlet conditions

1. INTRODUCTION
CFD codes are more and more used in order to predict air flow patterns or heat and pollutant transfers in rooms and they are becoming design tools for scientists or engineers. The more usual codes use the turbulent viscosity concept in order to model the fluctuating terms due to turbulence effect. Two additional equations enable to define the turbulent viscosity field in any location of the space: the conservation of kinetic turbulent energy, k and the conservation of its dissipation rate, ε.

However, when making comparisons between the results obtained by various authors for the same physical configuration, we can notice different results (EUROTHERM 1993).

As we already noticed in a previous study (Béghein et al. 1993), one of the origin of this discrepancy can be the k-ε model itself. In our applications, the wall functions are no more relevant to describe the behavior of boundary layer flows along the walls of the rooms, and the usual solution consisting in the introduction of low Reynolds number models leads to a better modeling but can be sensitive to the selected model. In this paper we focus on two other sources of uncertainty relative to the modeling of the turbulence parameters at supply inlet and to the influence of the shape of the incoming jet.

2. INFLUENCE OF TURBULENT BOUNDARY CONDITIONS AT SUPPLY INLET

2.1. Studied configuration
For this study we used as support the experimental study carried out for Annex 20 of IEA (Nielsen 1990). In this ventilated cavity, we can assume the flow to be two-dimensional. The cavity is in steady isothermal condition and the flow pattern is generated by the air injection. At first we consider a constant velocity profile at the inlet (U_o = 0.455 m/s). The outlet condition is obtained by mass conservation and normal gradients of turbulent kinetic energy and dissipation rate equal to zero. Figure 1 describes the studied configuration.

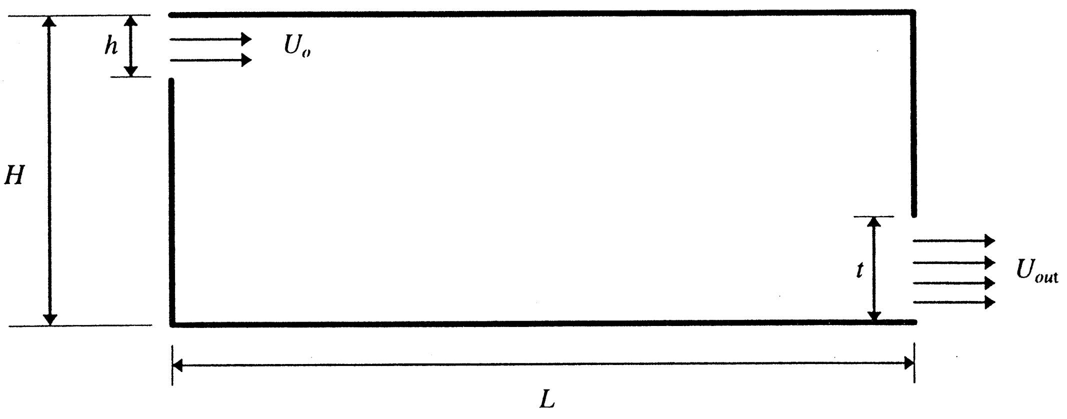


Figure 1: Two-dimensional ventilated cavity.
$H = 3$ m, $L/H = 3$, $t/H = 0.16$, $h/H = 0.056$

2.2. Governing set of equations

The governing set of equations for the flow in the cavity is based on the classical set of equations for k-ε model of turbulence. The values of constants appearing in the equations of the turbulent quantities are chosen following the model proposed by Henkes (1990). No wall functions are employed, so the boundary conditions for the kinetic energy and its rate of dissipation read $k|_{wall}=0$ and $\varepsilon|_{wall}=\infty$.

2.3. Inlet boundary conditions

A review of existing works has been done by Saïd et al. (1993). When most authors consider an isotropic turbulence and impose the turbulent kinetic energy at the inlet as $k_o = (3/2)\, I_o^2\, U_o^2$, where I_o is the turbulence intensity at the inlet: $I_o = u'_o/U_o$ (u'_o being the fluctuating term of the injection velocity and U_o its mean value); Saïd found at least six different formulations in the literature for defining the rate of dissipation of the turbulent kinetic energy at the inlet, ε_o.

2.4. Numerical procedure

The numerical integration of the set of equations governing the flow in the cavity is based on a control volume method using SIMPLER algorithm (Patankar 1980). The non-linear terms in the advection-diffusion equations are represented by a power law scheme, and a Tri-Diagonal-Matrix-Algorithm (TDMA) is used to solve the resulting linear systems.

2.5. Results

Two parametric studies have been considered. At first we vary the dissipation rate of the turbulent kinetic energy at the inlet,ε_o, and then the turbulence intensity, I_o.

2.5.1. Influence of the rate of dissipation of the turbulent kinetic energy at the inlet. For this first study we used a fixed turbulence intensity of 4% and we studied three different models reported by Saïd et al. (1993) for evaluating ε_o, that is the model of Lamers et Velde (1989), the model of Nielsen (1990), and the model of Skovgaard et Lemaire (1990). Table 1 presents the characteristics of these models. The Reynolds number at the injection, based on the height, h, of the inlet and on the mean velocity, U_o, is constant and equal to 5096.

In fact, the results presented in Figures 2 show that the influence of the formulation of the dissipation term is very weak. We cannot notice any relevant variation neither of the mean velocity nor of the fluctuating terms.

Source	Formula	Parameters
Lamers et Velde, (1989)	$\varepsilon_o = C_\mu k_o^{3/2}/l_o$	$k_o = (3/2)\, u_o'^2$ u'_o = measured fluctuating velocity = $0.04\ U_o$ $l_o = 0.1\ h/2$; h = width of inlet slot diffuser $C_\mu = 0.09$
Nielsen, (1990)	$\varepsilon_o = k_o^{3/2}/l_o$	$k_o = (3/2) I_o^2 U_o^2$ $I_o = 0.04$ $l_o = 0.1\ h$; h = width of inlet slot diffuser
Skovgaard et Lemaire, (1990)	$\varepsilon_o = (C_\mu)^{3/4} k_o^{3/2}/l_o$	$k_o = (3/2) I_o^2 U_o^2$ $I_o = 0.04$ $l_o = 0.03\ D_h$ D_h = hydraulic diameter of inlet diffuser

Table 1: Description of the three different models for ε_o, from Saïd et al. (1993).

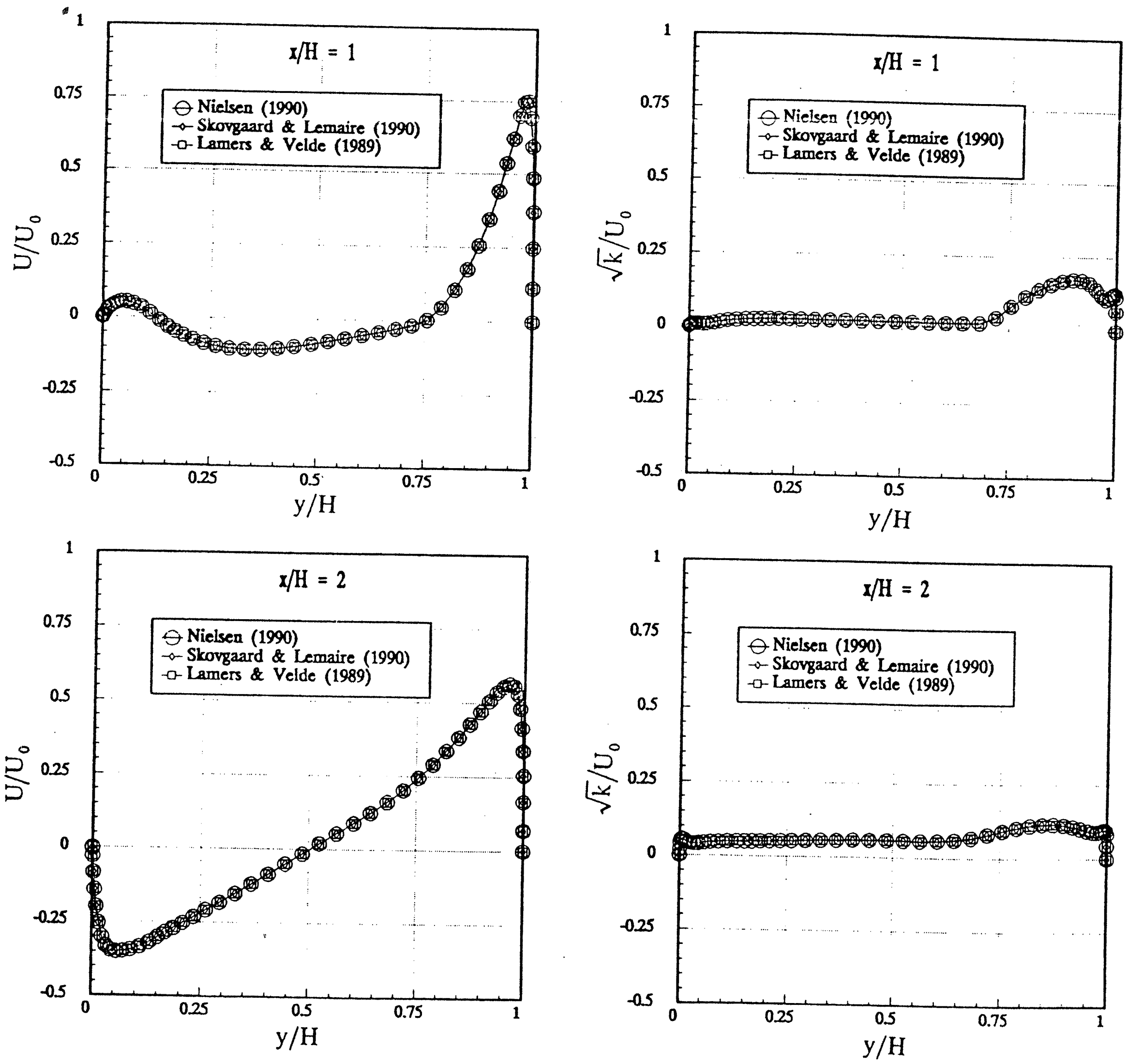


Figure 2: Vertical profiles of mean velocity and fluctuating term at $x/H = 1$ and $x/H = 2$.

2.5.2. Influence of the turbulence intensity at the inlet. For this second study we selected Nielsen formulation (cf. Table 1), in order to define the turbulent boundary conditions at the inlet, and we vary the turbulence intensity, I_o, from 4% to 9%, 14% and 37.4 %.

Figures 3 presents the resulting velocity profiles in two vertical planes located at $x/H = 1$ and $x/H = 2$. We can notice here that this variation of turbulent intensity does not modify the overall air flow patterns within the cavity.

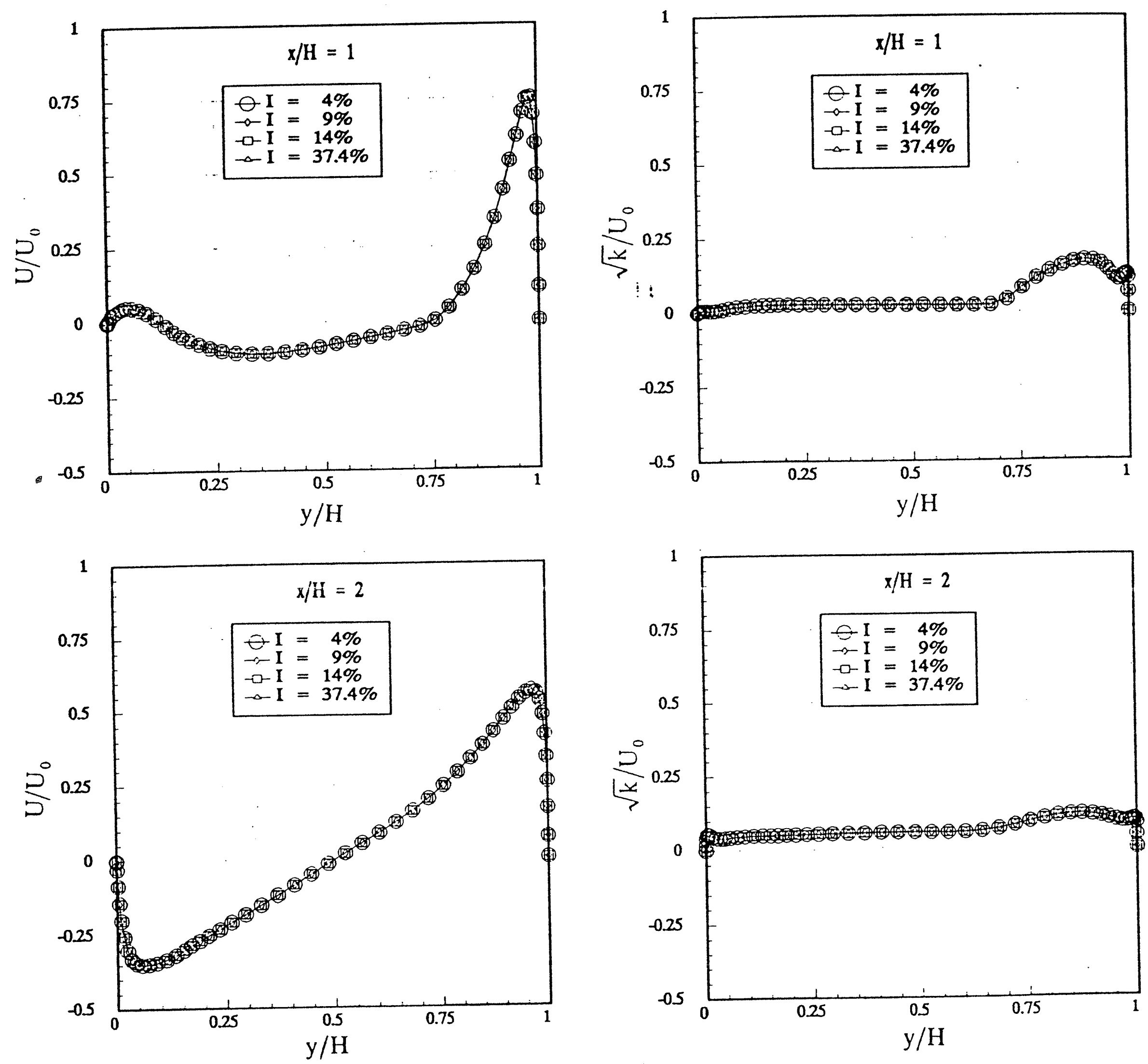


Figure 3: Vertical profiles of mean velocity and fluctuating terms at $x/H = 1$ and $x/H = 2$.

These results confirm the previous ones obtained by Saïd et al. (1993), using a different numerical scheme, and are finally very important because it is usually very difficult to characterize precisely the turbulent quantities within the inlet jet. This shows finally that their definition is not a leading parameter in order to predict air flow patterns in a ventilated room.

3. INFLUENCE OF JET GEOMETRY AT THE INLET

3.1. Configuration

For this second study, simulations have been performed with three different heights of the inlet and two configurations of the jet: a plane jet and a parabolic jet. The inlet Reynolds number is constant and equal to 5096. The physical configuration stays equal to the previous one, excepted for the outlet, which size has been reduced to $t/H = 0.014$. The studied cases are reported below.

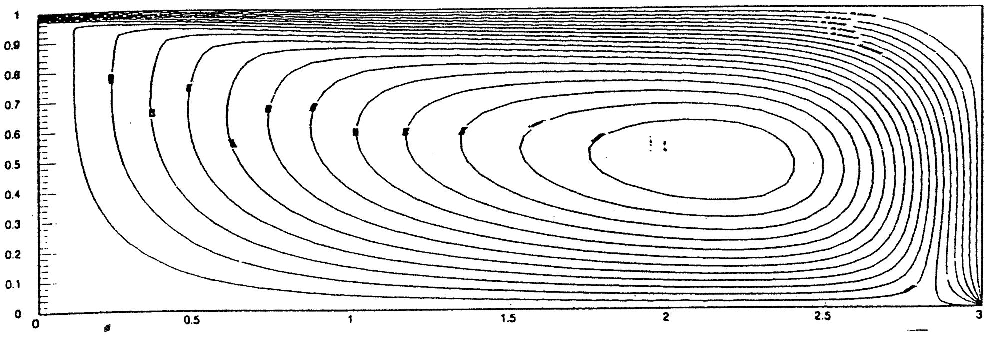
Case number	Inlet profiles		h [cm]	U_o [m/s]
1	flat	parabolic	8.4	0.910
2	flat	parabolic	16.8	0.455
3	flat	parabolic	33.6	0.2275

Table 2: Studied configurations.

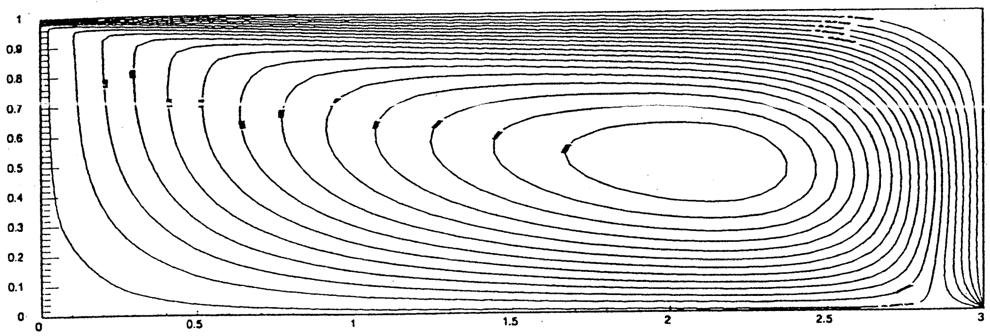
3.2. Results

The following figures show the stream line patterns obtained for each configuration.

case n° 1

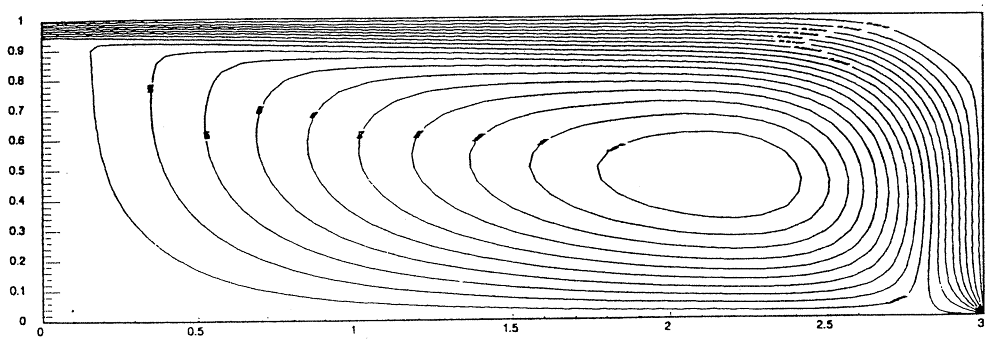


flat profile

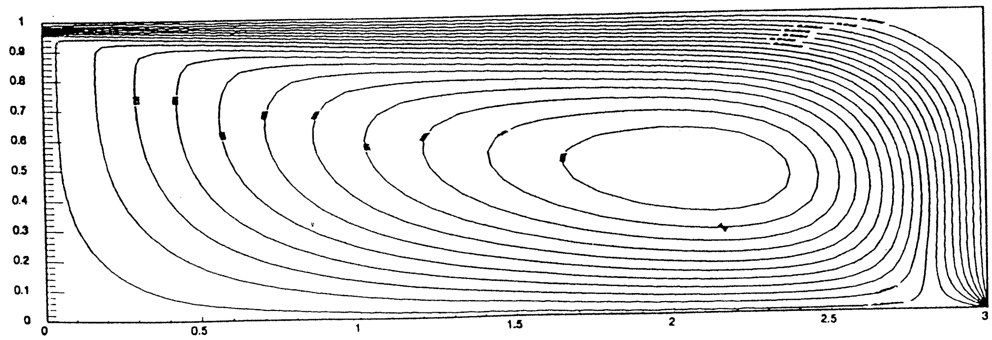


parabolic profile

case n° 2



flat profile



parabolic profile

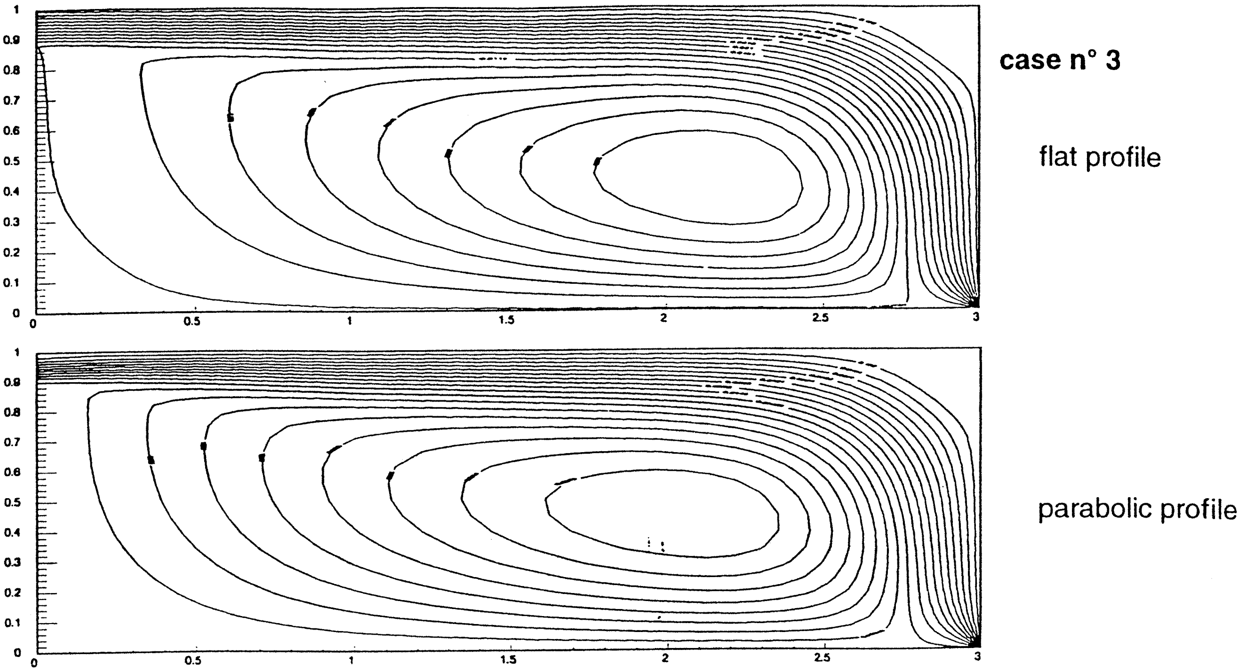


Figure 4: Stream line patterns of the various configurations.

We can see here that changing the injection profiles from plane to parabolic leads to a smooth translation of the dead zone of the cavity toward the outlet. The location and amplitude of the secondary flows do not seem to be affected by the profile of the injection jet.

Figure 5 demonstrates also that the mean velocity profiles within the cavity do no seem to be strongly modified in the central part of the cavity. The strongest modification is obviously found directly in the jet zone in the upper part of the enclosure.

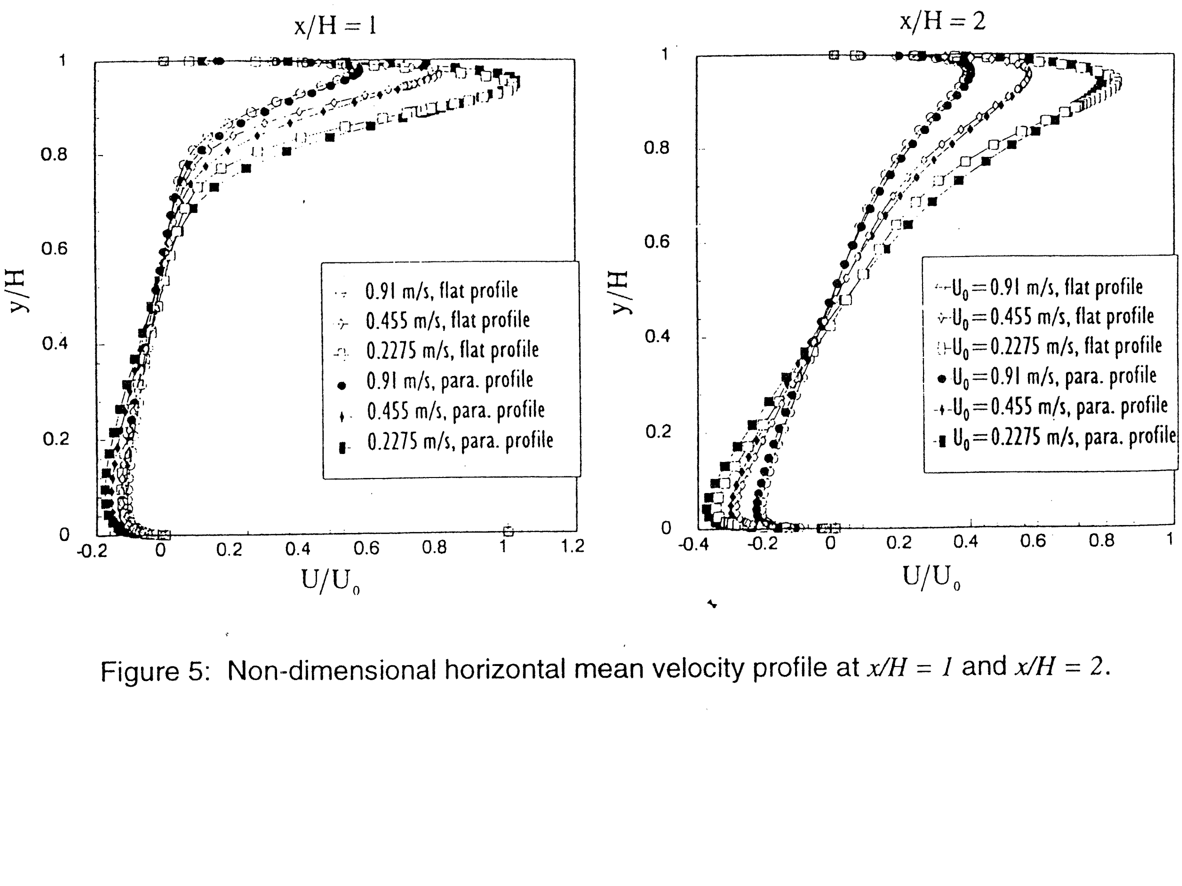


Figure 5: Non-dimensional horizontal mean velocity profile at *x/H = 1* and *x/H = 2*.

At the contrary, the turbulent quantities, namely the turbulent kinetic energy, appear to be very different when changing the injection profile. Furthermore, this difference in behavior increases when the dimension of the inlet increases. In fact, in case of a parabolic profile, the flow is sheared off by the jet and it generates a more important rate of turbulent kinetic energy. When the dimension of the inlet increases, this phenomenon is amplified because the shear zone is larger. Figure 6 presents this last result.

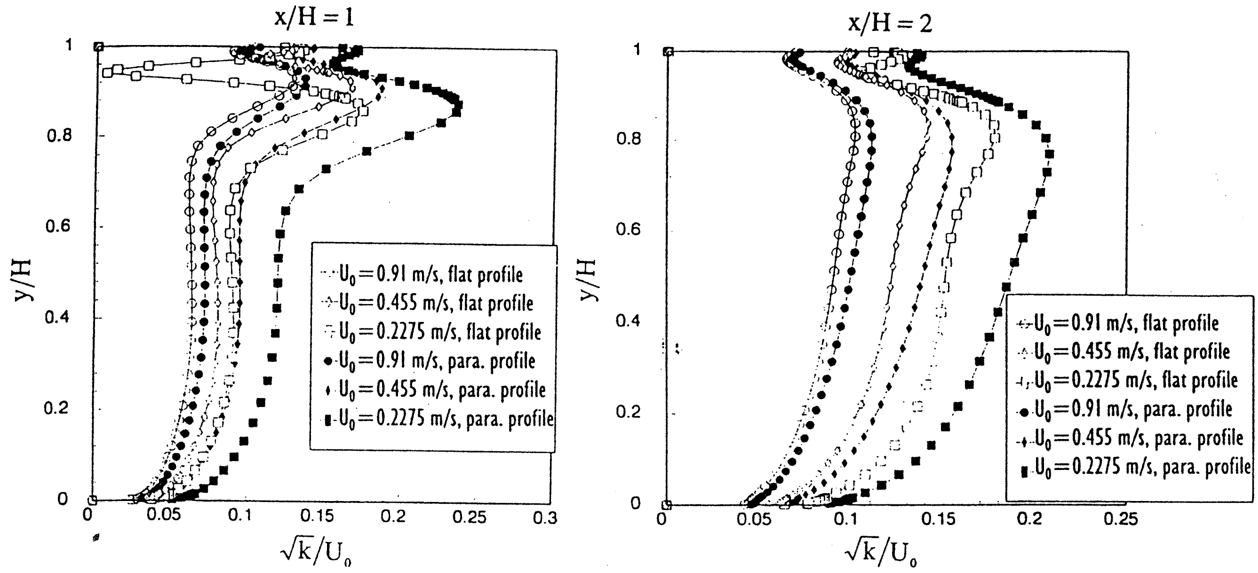


Figure 6: $\sqrt{k}\,/\,U_0$ for the three different cases.

4. CONCLUSION

The main objectives of these study were to check the influence of inlet boundary conditions on the prediction of air flow patterns within a ventilated enclosure.

The results obtained here are encouraging because they show that a rough knowledge of the turbulent quantities within the injection jet is sufficient to predict a reasonable air flow pattern within the cavity. This fact is important because in a design approach, this knowledge is in general quite poor. At the contrary, the knowledge of the shape of the injection jet is important in order to predict accurately the development of turbulence within a ventilated room. This fact may be important when dealing with comfort studies in the occupation zone of a room or when predicting pollutant diffusion in a room.

ACKNOLEDGMENTS

Computations have been performed on the IBM SPS 32 computer of CNUSC (Centre National Universitaire Sud de Calcul), Montpellier, France.

REFERENCES

Béghein, C.; Penot, F.; Mergui, S.; Allard, F. (1993)
"Numerical and Experimental Evaluation of Turbulent Models for Natural Convection Simulation in a Thermally Driven Square Cavity", ASME Winter Annual Meeting, New-Orleans, Nov. 28-Dec. 3, ASME paper 93-WA/HT-46.

EUROTHERM (1993)
Seminar 22 Proceedings, "Turbulent Natural Convection in Enclosures: a Computational and Experimental Benchmark Study", March 25-27, 1992, Delft, Netherlands, Henkes, R.A.W.M. and Hoogendoorn, C.J. editors, Editions Européennes Thermique et Industrie.

Henkes, R.A.W.M. (1990)
"Natural Convection Boundary Layers". Ph.D. thesis. Delft University of Technology, Holland,1990, 198 p.

Lamers, A.P.G.G.; Velde, R. van de (1989)
"Air Flow Patterns in Ventilated Rooms". Phoenics Journal, Vol. 2, No. 2, pp. 219-238.

Nielsen, P.V. (1990)
"Specification of a Two-dimensional Test Case". International Energy Agency Annex 20, Research Item 1.45, Technical Report, Aalborg University, Aalborg, Denmark, ISSN 0902-7513 R9040, Nov. 1990.

Patankar, S.V. (1980)
"Numerical Heat Transfer and Fluid Flow". London, Mac Graw Hill, 1980. 197 p.

Saïd, M.N.A.; Juini, D.B.; Plett, E.G. (1993)
"Influence of Turbulence Tarameters at Supply Inlet on Room Air Diffusion". ASME Winter Annual Meeting, New-Orleans, Louisiana, Nov. 28 - Dec. 3, 1993.

Skovgaard, M.; Lemaire, A.D. (1990)
"Basic Model, Representation of Boundary Conditions at Supply Opening", Technical Note on Research Item 1.11, International Energy Agency Annex 20, TNO, Delft, Holland, Feb. 1990.