

INFLUENCE OF ASPECT RATIO IN THE TURBULENT CONVECTION IN A RECTANGULAR CAVITY

Renata Munhoz Guimarães, renatamunhoz2607@hotmail.com
Viviana Cocco Mariani, viviana.mariani@pucpr.br
Pontifícia Universidade Católica do Paraná

Kátia Cordeiro Mendonça, katia.cordeiro_mendonca@univ-lr.fr
Universidade de La Rochelle

Abstract. *The purpose of this study is to know the air distribution in a conditioned room, through the numerical investigation of the influence of aspect ratio in the thermal and fluid dynamics behavior of a turbulent flow. To achieve that objective, some simulations were done of the flow inside a rectangular room, where the air enters through an opening in the top of one wall and leaves the room through an opening in the bottom of the opposite wall. The Reynolds mean equations are used, with the turbulence model RSM BSL (Reynolds Stress Model - Baseline) to solve four cases, with different geometries. It was concluded that, in general, the turbulence model used in this work is capable to predict quite well the fluid dynamics behavior of the flow, which is influenced by the room length, but not by its width.*

Keywords*: turbulence, CFD, Reynolds Stress Model.*

1. INTRODUCTION

A very important piece of the population spends most part of the day inside enclosed places, which normally are conditioned to offer adequate sanitary and comfort conditions to the occupants. However, the available air conditioning systems produce heterogeneous conditions of air properties that can be unpleasant for the occupants, even when the global thermal perception is satisfactory. Besides that, the airflow impacts the heat and mass exchanges between air and the environment envelope, and consequently the energy consumption. The air quality inside an enclosed environment also depends on the airflow, because the pollutant sources can get concentrated in some places and affect the occupants in different ways. Thus, in order to design environments energetically efficient, maintaining adequate comfort and quality conditions to the occupants, it is necessary to determine air and its properties distribution inside the room.

Detailed information about the thermal and fluid dynamics behavior of airflows inside conditioned environments can be obtained through Computational Fluid Dynamics (CFD), that can be described as the numerical simulation of all physical and chemical processes present in the airflow.

Other authors have done experimental and numerical studies about the prediction of the airflow inside environments with natural, forced or mixed convection. Nielsen (1976) has experimentally investigated the isothermal and non isothermal airflows in conditioned rooms and has obtained velocity and temperature results for two and three dimensional environments, that are used as reference in this study. In 1990, Nielsen studied the same model of 1976 to obtain dimensionless mean velocity and turbulence intensity results. Gan (1995) has used CFD to predict the thermal comfort in a room with forced convection, in order to optimize the air conditioning system. Chen (1996) has used different turbulence models to determine the behavior of airflow inside rooms with natural, forced or mixed convection and noticed that the Reynolds Stress Model (RSM) was very efficient, but required more computational resources than standard k-ε model. Costa *et al*. (2000) have studied the influence of geometric, dynamics and thermal parameters in the behavior of the two dimensional turbulent airflow, through numerical simulation using the model defined in a previous study of the authors (1999). Schalin *et al*. (2004) have analyzed the performance of two turbulence models in the fluid dynamics prediction of an isothermal airflow, and have concluded that the Reynolds Stress Model presented better results than standard k-ε model. Susin *et al*. (2008) have studied the same model of Nielsen (1976), using three different two-equation models, and then compared the numerical results to the experimental of the literature. The authors have realized that all the models had well predicted the mean flow, but not the turbulence intensity, because of the anisotropy assumption of the models used.

The main objective of this study is to investigate numerically the influence of geometry aspect ratio, that means the ratio between the width and the length of the room, in the thermal and fluid dynamics behavior of the isothermal turbulent flow inside it. Thus, this work aims to contribute with the improvement of air quality, occupants thermal comfort and air conditioning systems efficiency in enclosed places, through the airflow analysis. To achieve those objectives, the airflow is simulated in CFD, using the Baseline Reynolds Stress Model, which was chosen because it is one of the most accurate turbulence models. This work also analyzes if this turbulence model shows some advantages comparing to more common ones, like two-equation turbulence models.

2. MATHEMATICAL MODEL

The airflow inside the room in study is described by the continuity equation, Eq. (1), together with the energy conservation equation and the Reynolds mean equations, Eq. (2), (3) and (4), which are obtained by the time-averaging of Navier-Stokes momentum equations. The application of this operator despise the details of all turbulent fluctuations, what results in six additional terms, the Reynolds stresses, which can be predicted by a turbulence model. The BSL Reynolds Stress Model applies the transport equation for the dissipation of kinetic energy (ω), Eq. (5), and for the six Reynolds stresses equations (R_{ij}), Eq. (6), once it considers the turbulent viscosity as anisotropic.

$$\frac{\partial \rho}{\partial t} + div(\rho \mathrm{U}) = 0 \tag{1}$$

$$\frac{\partial U}{\partial t} + div(U\mathrm{U}) = -\frac{1}{\rho}\frac{\partial \mathrm{P}}{\partial x} + v\ div\ grad\ U + \left[-\frac{\partial \overline{u'^2}}{\partial x} - \frac{\partial \overline{u'v'}}{\partial y} - \frac{\partial \overline{u'w'}}{\partial z}\right] \tag{2}$$

$$\frac{\partial \mathrm{V}}{\partial \mathrm{t}} + div(\mathrm{VU}) = -\frac{1}{\rho}\frac{\partial \mathrm{P}}{\partial \mathrm{y}} + v\ div\ grad\ \mathrm{V} + \left[-\frac{\partial \overline{\mathrm{u'v'}}}{\partial \mathrm{x}} - \frac{\partial \overline{\mathrm{v'^2}}}{\partial \mathrm{y}} - \frac{\partial \overline{\mathrm{v'w'}}}{\partial \mathrm{z}}\right] \tag{3}$$

$$\frac{\partial W}{\partial t} + div(W\mathrm{U}) = -\frac{1}{\rho}\frac{\partial \mathrm{P}}{\partial z} + v\ div\ grad\ W + \left[-\frac{\partial \overline{u'w'}}{\partial x} - \frac{\partial \overline{v'w'}}{\partial y} - \frac{\partial \overline{w'^2}}{\partial z}\right] \tag{4}$$

$$\rho U_j \frac{\partial \omega}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\omega}\right)\frac{\partial \omega}{\partial x_j}\right] + \alpha \mu_t \frac{\omega}{\mu_t}\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)\left(\frac{\partial U_i}{\partial x_j}\right) - \beta \rho \omega^2 \tag{5}$$

$$\frac{DR_{ij}}{Dt} = P_{ij} + D_{ij} - \varepsilon_{ij} + \Pi_{ij} + \Omega_{ij} \tag{6}$$

Where ρ is the specific weight, U, V and W are the mean velocity in the directions x, y and z, P is the pressure and u', v' and w' are the fluctuations of velocity in the directions x, y and z. And R_{ij} are the Reynolds stresses, P_{ij} is the rate of production of R_{ij}, D_{ij} is the transport of R_{ij} by diffusion, ε_{ij} is the rate of dissipation of R_{ij}, Π_{ij} is the transport of R_{ij} due to turbulent pressure-strain interactions and Ω_{ij} is the transport of R_{ij} due to rotation.

3 NUMERICAL MODEL

3.1 Physical Model

The physical model used in this study is based on the experimental device created by Nielsen (1976, 1990), that represents a room where the air enters horizontally parallel to the ceiling (left wall) and leaves through an opening near the floor (right wall), as shown in Fig. 1.

The height of the room, H, is 3m, the height of the inlet slot, h, is 0.168m and height of the outlet opening, t, is 0.48m. To analyze the influence of the aspect ratio in the behavior of the flow, the ratios L/H and W/H varies case to case, according to Tab. 1.

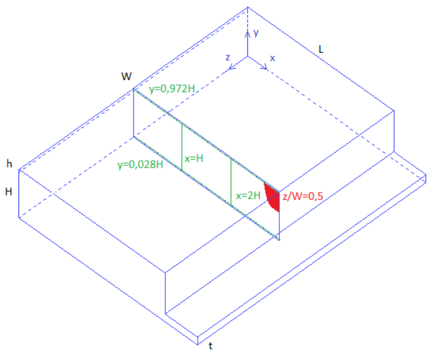

Figure 1. Schematic representation of the model.

Table 1. Aspect ratio of the geometry in the cases studied.

Case	L/H	W/H
1	3	4.7
2	1	1
3	3	1
4	4.7	1

The case 1 has the exact same dimensions of the device studied by Nielsen (1976, 1990), thus the validation of the present model is made by comparing the results of dimensionless mean velocity and turbulent intensity of this case to the experimental data obtained by that author. The analysis are made in four lines in the symmetry plan, being two horizontals, positioned in y=0.028H and y=0.972H, and two verticals positioned in x=H and x=2H , according to Fig. 1.

After the validation, the results are compared between the cases 1 and 3, to analyze the influence of the width of the room, W, and between the cases 2, 3 and 4, to analyze the influence of the length of the room, L, in the resulting airflow.

3.2 Spatial Discretization

At least three meshes are created for each case, in order to define the most adequate spatial discretization to simulate the airflow in study. To initiate the meshes development, the computational domain is created through the software ANSYS ICEM CFD (Version 11), and divided into four blocks that distinguish the inlet, the outlet and the outlet slot. After this, the meshes are created by the definition of three parameters, the distance between the first node and the wall, the expansion factor, considered as 1.25 for all the directions, and the kind of function used by the software to distribute the nodes, considered as exponential. The characteristics of the three meshes used in case 1 are shown in Tab. 2.

Table 2. Characteristics of the three meshes used in case 1.

Case	Meshes	Discretization (x, y, z)	Number of Elements
1	1	30x20x50	42916
	2	60x40x50	145680
	3	60x80x50	284672

Tab. 2 shows that mesh 1 is coarse and mesh 3 is the finest, according to the number of elements. The mesh 2 is defined through the analysis of y+ results of mesh 1 and its refinement in *x* and *y* directions. The same way, mesh 3 is defined through the refinement of mesh 2 in *y* direction. Fig. 2 shows the details of the three meshes discretization in the lateral plane. The meshes for the other cases were defined through the same methodology, using y+ results to help the refinement.

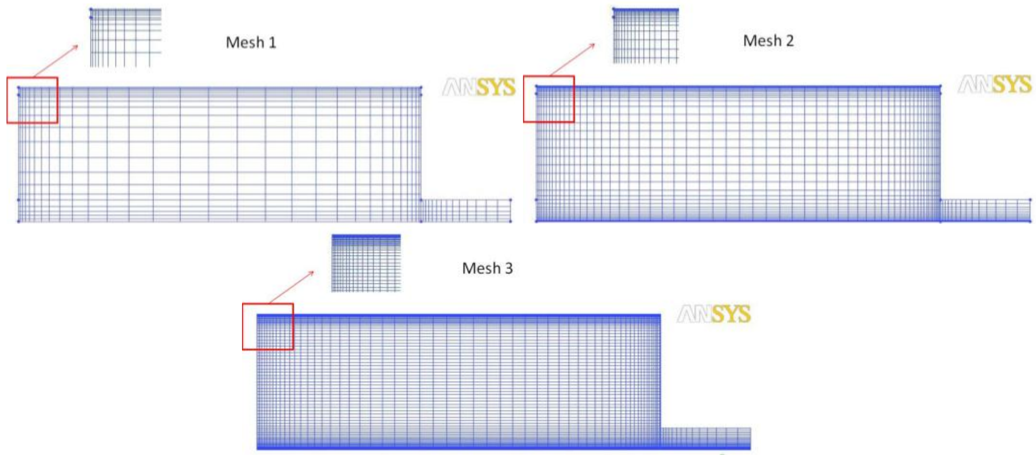

Figure 2. Lateral section of the three meshes used in case 1.

In order to calculate the advection terms in the finite-volume equations, the advection scheme must be chosen, because it significantly impacts the convergence and accuracy of the numerical results. The advection scheme can be of first order precision, upwind, second order precision, high resolution, or a intermediate scheme, defined through the blend factor. Upwind advection scheme is robust and stable, although less accurate than the high resolution scheme, which considers a correction factor in the advection terms. Thus, in this study, the upwind scheme is used at first for all the cases, in order to obtain the initial results for the next simulations, where the blend factor is increased until the high resolution scheme can be used.

3.3 Boundary Conditions

The boundary conditions are defined so that the governing equations of the airflow can be solved. The boundary condition in the inlet is prescribed velocity, which can be obtained through the definition of Reynolds number, based on the height of the inlet slot, $Re=\rho h U_0/\mu$. $Re=4700$, $\rho=1.166$ kg/m^3 and $\mu=1.7825\text{x}10\text{-}5$, then $U_0=0.4277$ m/s. Considering that the inlet airflow is horizontal, the velocity in the other directions are null. According to the equations for the turbulent kinetic energy, k_0, and the dissipation of turbulent kinetic energy, ε_0, presented by Nielsen (1990), $k_0=0.000438986$ m^2/s^2 and $\varepsilon_0=0.000547477$ m^2/s^3. Besides that, the pressure inside and outside the room are 101325 Pa.

In the walls, the velocity are null, U=V=W=0, then k_0=0 and ε_0=0. In the outlet, the relative pressure is null and the flow is considered fully developed, as well as in the symmetry plan, then $\partial U/\partial \hat{n} = \partial V/\partial \hat{n} = \partial W/\partial \hat{n} = \partial k/\partial \hat{n} = \partial \varepsilon/\partial \hat{n} = \partial \omega/\partial \hat{n} = 0$.

4 RESULTS AND DISCUSSIONS

4.1 Numerical Validation

In order to validate the numerical simulations of case 1, the numerical results obtained by the three meshes are compared with the experimental data (Nielsen 1976) and the numerical results obtained through the k-ω turbulence model (Susin, 2008), in terms of dimensionless mean velocity and turbulent intensity, as shown in Fig. 3 and 4.

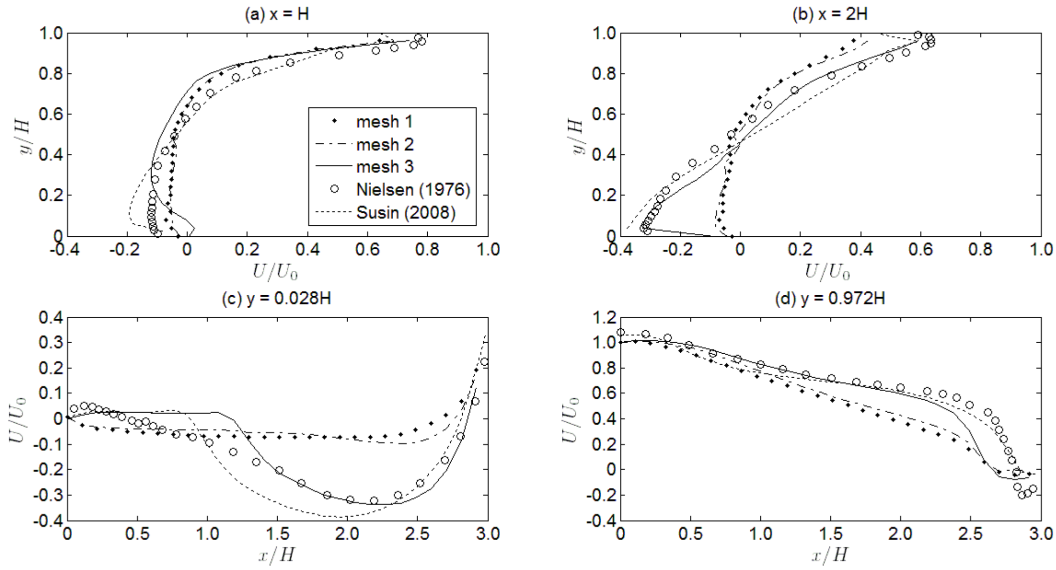

Figure 3. Dimensionless mean velocity in the symmetry plan.

In general, Fig. 3 shows that meshes 1 and 2 have obtained similar results and mesh 3 could predict the airflow mean velocity more accurately than the others, in all regions. However, the difference between numerical and experimental results near the floor, where 0.5<x/H<1.5, and near the ceiling, where 2.5<x/H<3, shows that the velocity was underestimated by the model. Comparing mesh 3 velocity results to the ones obtained through k-ω turbulence model (Susin, 2008), it is noticed that both models presented similar behaviors in x=H and x=2H, although near the floor RSM has underestimated the velocity while k-ω turbulence model has overestimated it. Fig. 3 (c) also shows that RSM has estimated the velocity much better than k-ω turbulence model in 1.5<x/H<2.3.

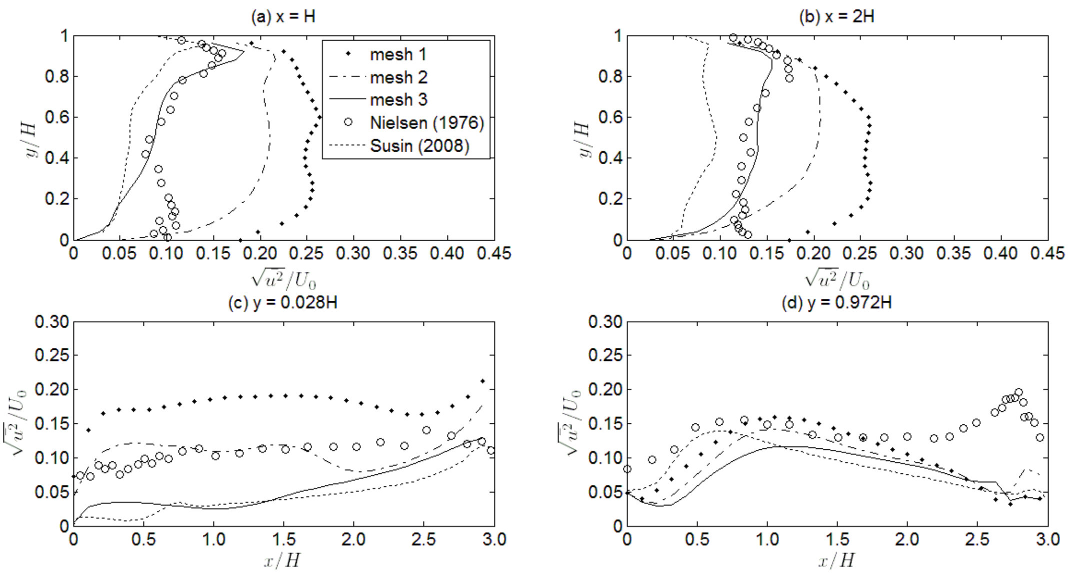

Figure 4. Dimensionless turbulent intensity in the symmetry plan.

It is noticed in Fig. 4 that mesh 3 was the only one capable to predict the turbulence intensity in x=H, though it presents discrepancy to the experimental results as it gets near to the floor, in y/H<0.4. In x=2H, the same behavior occurs, but only in y/H<0.2. Fig. 4 (c) shows that mesh 2 has obtained good results near the floor through all the length of the room, but analyzing Fig. 4 (a) and (b), it is possible to notice that the similarity between those results is only punctual. Fig. 4 (d) shows that mesh 1 presented the best results compared to the experimental ones near the ceiling, although in x/H>2, none of them, not even k-ω turbulence model, were able to predict the turbulence intensity. Comparing mesh 3 turbulence intensity results to the ones obtained through k-ω turbulence model (Susin, 2008), it is possible to say that RSM has predicted more accurately than the other model.

To summarize, Figures 3 and 4 shows that the mesh 3 could predict the mean velocity and turbulent intensity of the airflow with precision, but presented some difficulty near the right wall and the floor, as well as the k-ω turbulence model (Susin, 2008).

4.2 Aspect Ratio Analysis

The results of dimensionless mean velocity and turbulent intensity are compared between the cases 1 and 3, to analyze the influence of the width of the room, W, and between the cases 2, 3 and 4, to analyze the influence of the length of the room, L, in the airflow behavior, as shown in Fig. 5 to 8.

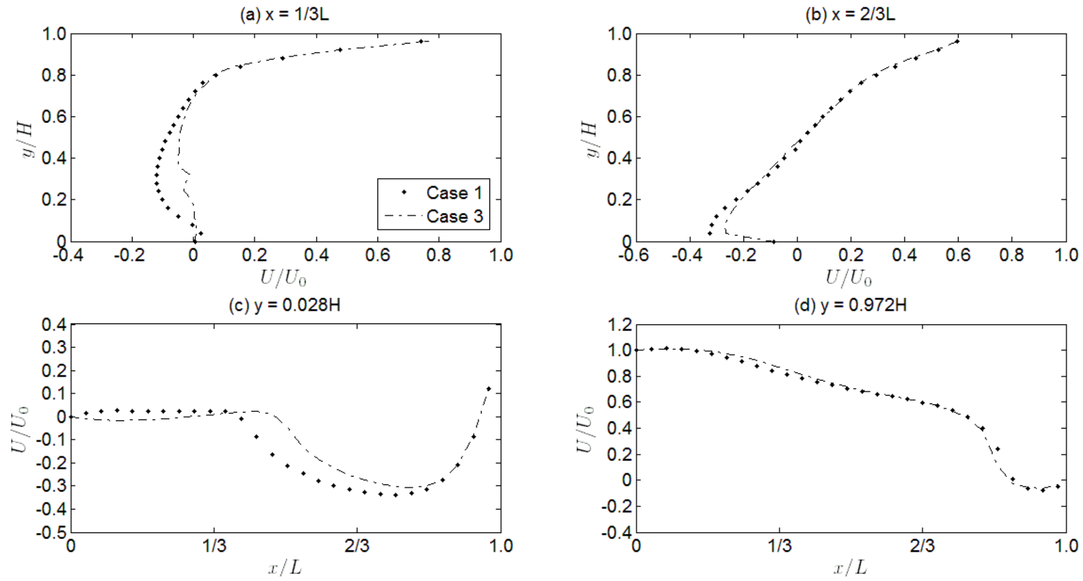

Figure 5. Dimensionless mean velocity in the symmetry plan, for cases 1 and 3.

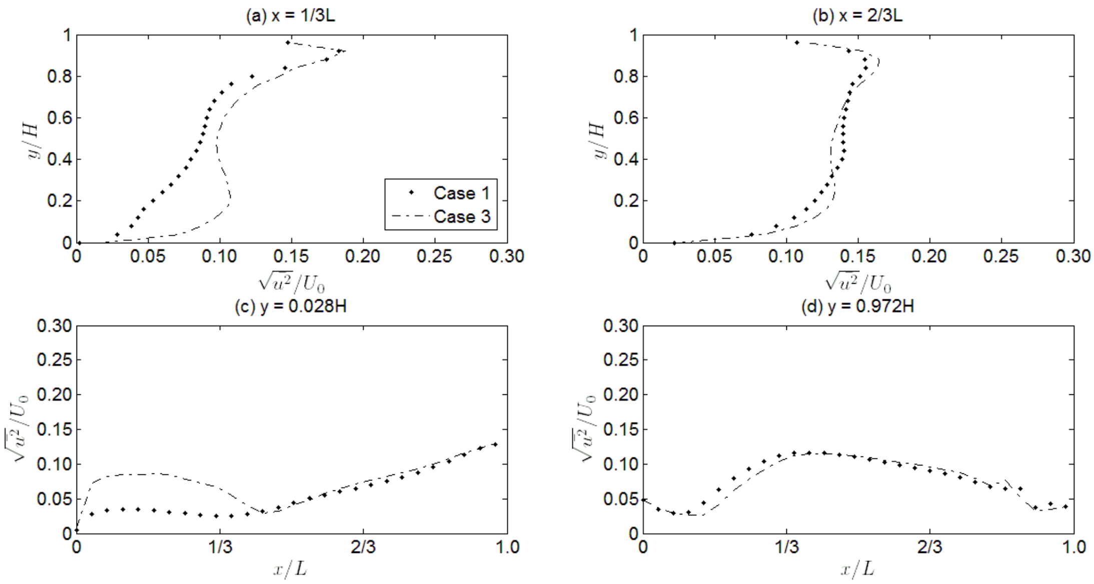

Figure 6. Dimensionless mean turbulent intensity in the symmetry plan, for cases 1 and 3.

Figures 5 and 6 shows that the mean velocity and turbulent intensity results of cases 1 and 3 are very similar, which indicates that the width of the room, W, does not influence the behavior of the flow.

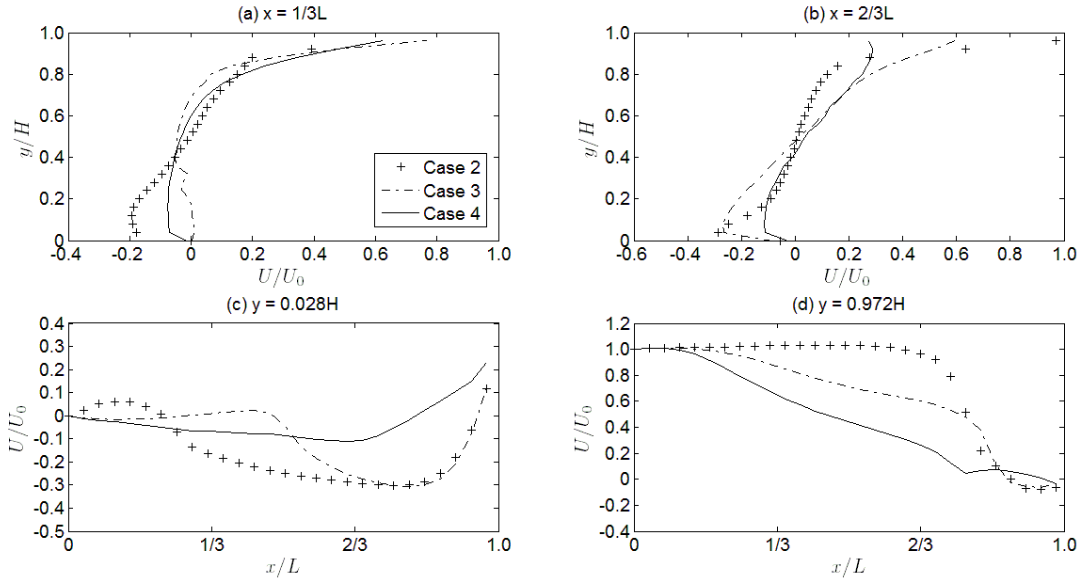

Figure 7. Dimensionless mean velocity in the symmetry plan, for cases 2, 3 and 4.

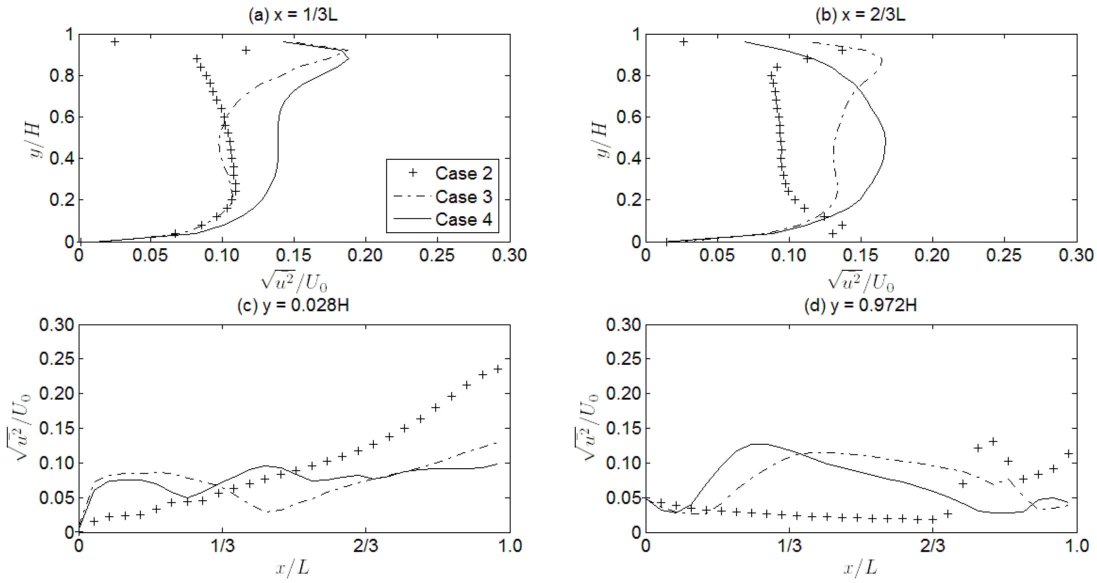

Figure 8. Dimensionless mean turbulent intensity in the symmetry plan, for cases 2, 3 and 4.

The comparison of the mean velocity and turbulent intensity results of cases 2, 3 and 4 shows visible differences among them. It is possible to realize that the point where the inlet velocity decreases varies case by case. It is also noticed that the turbulence intensity presents three dimensional effects. Thus, it is concluded that the length of the room, L, influences the behavior of the flow inside it.

5. ACKNOWLEDGEMENTS

In the validation, it was concluded that, in general, the turbulence model used in this work, Baseline Reynolds Stress Model, is capable to predict quite well the fluid dynamics behavior of the flow, even though the mean velocity had been underestimated in some regions of the ceiling and floor of the room. After the validation, the influence of aspect ratio in the flow was analyzed, changing the width and length of the room. It was concluded that the behavior of the flow is influenced by the room length, because the dimensionless profiles of velocity and turbulence intensity were different between the cases with different lengths, but not by the room width.

6. REFERENCES

Chen, Q., 1996. Prediction of room air motion by Reynolds-Stress models. Building and Environment, Vol. 31, pp. 233-244.

Costa, J. J., Oliveira, L. A. e Blay, D., 1999. Test of several versions for the k-e type turbulence modeling of internal mixed convection flows. International Journal of Heat and Mass Transfer, Vol. 42, pp. 4391-4409.

Costa, J. J. *et al*., 2000. Turbulent airflow in a room with a two-jet heating-ventilation system - a numerical parametric study. Energy and Buildings, Vol. 32, pp. 327-343.

Fox, R. W. e Mc Donald A. T., 1995. Introdução à Mecânica dos Fluidos, 4a edição, Rio de Janeiro: LTC.

Gan, G., 1995. Evaluation of room air distribution systems using computational fluid dynamics. Energy and Buildings, Vol. 23, pp. 83-93.

Launder, B. E. *et al*., 1975. Progress in the development of a Reynolds-Stress turbulence closure. Journal of Fluid Mechanics, Vol. 68, pp. 537-566.

Nielsen, P. V., 1990. Specification of a two-dimensional test case. Technical Report, International Energy Agency, Annex 20: Air Flow Pattern within Buildings.

Nielsen, P. V., 1976. Flow in air conditioned rooms. Model experiments and numerical solution of the flow equations. Tese de PhD, Technical University of Denmark, Copenhagen.

Schalin, A. e Nielsen, P. V., 2004. Impact of turbulence anisotropy near walls in room airflow. Indoor Air, Vol. 14, pp. 159-168.

Susin, R. M. *et al*., 2008. Evaluating the influence of the width of inlet slot on the prediction of indoor airflow: Comparison with experimental data. Building and Environment, Vol. 44, pp. 971-986.

K. e Malalasekera, W., 1995. An Introduction to computacional fluid dynamics. Longman Scientific & Technical, New York.

Voigt, L. K., 2000. Comparison of turbulence models for numerical calculation of airflow in an annex 20 Room. Technical Report, Technical University of Denmark.

7. RESPONSABILITY NOTICE

The authors are the only responsible for the printed material included in this paper.