CFD ANALYSIS OF COLD AIR DISTRIBUTION CENTERING ON THERMAL COMFORT

Rafael Costa Engel, rafaelengel@bs2.com.br
Alexandre M. S. Costa, amscosta@uem.br
Karla Silva, karla @utfpr.edu.br
Universidade Estadual de Maringá, Av. Colombo 5790 – CEP. 87020-900 - Maringa – Parana - Brazil

Abstract. *An available CFD package was used in this project to simulate different ways of air conditioning distribution focusing on thermal comfort. Based on field practices, the simulation conditions were set up and different cases with differences in the inlet and outlet places. After the convergence of the problem to the steady situation, the results of temperature, humidity and velocities obtained were analyzed and compared with the Brazilian standards (ANVISA and ABNT NBR 16401-1).* .

Keywords*: thermal comfort, numerical analysis, CFD*

1. INTRODUCTION

The need for a thermally comfortable indoor environment is object of ongoing studies. For summer conditions some authors, as Kessler and Nikol (1998), point that thermal comfort can be reached with well projected natural ventilation. On the other hand, some researchers point the need for mechanical ventilation, stressing the balance between the air conditioning and the thermal regulation of the human body (Umeno et al, 2001).

The ASHRAE 781-RP report, by Fountain and Huizenga (1994), lists some relevant thermal comfort models such as the PMV-PPD (Fanger, 1972). During the last three decades several comfort models have been applied in situations such as different indoor geometries, human activities, and geographic location. Some authors introduced some additional variables aiming more realistic issues. As example, De Dear et al. (1998) considered variable indoor temperature, behavioral and psychological adjustment in hot and cold places. Based on the models of Fanger (1972) and Gagge et al (1986), Yigit (1999) developed a two dimensional computational model that considers clothing effect with drying and evaporation. Lamberts and Xavier (2000) considered the thermal comfort indexes and indoor variables in a Brazilian class room experiment.

Nowadays, Computational Fluid Dynamics (CFD) application in the indoor environment is important elements in the study of energy consumption, thermal comfort and indoor air quality in buildings. Indoor environmental design requires detailed information about air distribution, such as airflow pattern, velocity, temperature, humidity, and pollutant concentrations. In this sense, CFD can aid in situations when experimental measurement cannot be a practical design tool. In HVAC, the use of computational fluid dynamics allows different approaches : macroscopic analysis of indoor environments, simulating temperature, velocity and humidity profiles, as the case of the current study, contaminant transport (e.g. Kowalski et al. 2003; Chang, et al, 2006; Zhu et al, 2007, Nielsen et al, 2008; Tung et al. 2009), refrigerant multiphase flow (Gang et al. 2005), radiation exchange (Murakami, 2001), data centers(Nishihara et al. 1998; Herrlin, 2008). Microscopic analysis are also performed using CFD, normally in the proximity of human body and using experimental thermal manikin data (Nielsen, 2002; Nilsson and Holmér, 2003, Murakami, 2004; Ooka et al. 2008). On the other hand, as pointed by Sorensen and Nielsen (2003) and Chen (2007), it is important to consider the quality of the CFD predictions in HVAC in order to obtain a sufficient level of reliability. With the above CFD shortcomings and pitfalls in mind, in the current work is performed a study of cold air distribution with simulation conditions based on air conditioning field practices.

1.1 Brazilian Standards and Codes

Agencies, such as ANVISA and ABNT, are the rulers for Brazilian thermal comfort. These agencies establish the codes and standards for thermal comfort indexes, based on international standards, such as ISO 7730. Some Brazilian studies validated the thermal comfort models based on the above codes and standards. (Lamberts and Xavier, 2000).

ANVISA and ABNT NBR 16401 establish an indoor thermal comfortable temperature as 23 °C. For this temperature there is a 2 °C tolerance. This tolerance is the maximum acceptable temperature spatial variation between any two points in the indoor environment. Besides, the air velocity range should be in the 0.025 to 0.25 m/s range and the relative humidity should be in the 50 to 60 % range. It must be pointed; the reading of the above parameters should be in the habitable zone. The habitable zone is located up to 1.5 m height from the floor.

2. MATHEMATICAL MODEL

The mathematical model is based on the equations of global mass, momentum, energy and species transport. (Eq.1 to 4).

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = 0 \tag{1}$$

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \nabla \cdot (\rho \vec{v}\vec{v}) = -\nabla p + \nabla \cdot (\overline{\overline{\tau}}_{eff}) + \rho \vec{g} + \vec{F} \tag{2}$$

$$\frac{\partial}{\partial t}(\rho E) + \nabla \cdot (\vec{v}(\rho E + p)) = \nabla \cdot \left(k_{eff} \nabla T - \sum_i h_i \vec{J}_i + (\overline{\overline{\tau}}_{eff} \cdot \vec{v}) \right) \tag{3}$$

$$\frac{\partial}{\partial t}(\rho Y_i) + \nabla \cdot (\rho \vec{v} Y_i) = \nabla \cdot \vec{J}_i \tag{4}$$

where: ρ is the density of mixture composed by two chemical species (dry air and steam) $\vec{v}$ is the instantaneous velocity field for the mixture and p is the pressure field. In the stress field "$\overline{\overline{\tau}}_{eff}$" is included the effect of the turbulence model (Reynolds-averaged Navier Stokes-RANS). $\vec{g}$ stands for the gravitational acceleration and in the $\vec{F}$ term is included the buoyancy term given by the Boussinesq approximation. In the energy equation (Eq. 3) the E is the sum of the internal and kinetic energy by unitary mass. The term k_{eff} is the effective thermal conductivity and its formulation is accounted the turbulence model. T is the temperature field, h_i is enthalpy for the chemical specie j, and $\vec{J}_i$ the diffusive flux. The 1st three terms on the right-hand side of Eq. (3) represent the energy transfer due to conduction, species diffusion, and viscous dissipation, respectively. It must be pointed that in our current modeling we are neglecting the condensation and surface radiation exchange phenomena. In Eq. (4) Y_i is the local mass fraction for the chemical specie i.

All those aforementioned terms in the above equations are already accounted in the CFD package kernel and they can be considered on the analysis by choosing them on the software interface and using some specific language script. The reader is directed to the package documentation (Open CFD, 2008a, b) for more details about the numerical solution algorithm and mathematical model.

3. METHODOLOGY

The available CFD package was used as a tool of analysis for the distribution of temperature and air speed over a standard enclosure. The enclosure was standardized as a room with dimensions 6.0 × 6.0 × 3.5 meters.

As shown in Figure 1, four different inlet and outlet configurations for air distribution were analyzed. In the figure is also sketched the presumed path followed by the air for the enclosure´s central plane. A more detailed view of the enclosure is also given in Fig. 2, for case (c).

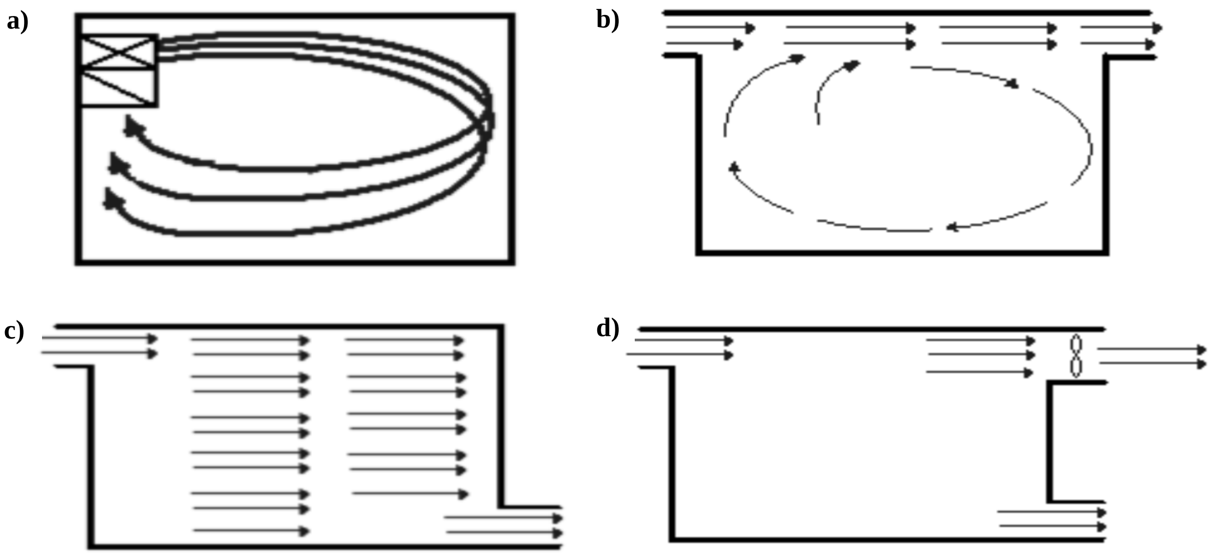

Figure 1. Air distribution: (a) Cross flow with inlet and outlet at the same wall; (b) Cross flow with inlet and outlet in opposite walls; (c) Bottom; (d) Mixed.

Copyright © 2010 by ABCM

Boundary conditions adopted for simulation are presented at Table 1.

Table 1 – Boundary conditions for simulations

Variable	**Value**
Supply air temperature	11 °C
Room initial temperature	32 ºC
Room initial relative humidity	50 %
Total sensible heat flux (walls and roof)	70 W/m²
Supply air velocity	2.0 m/s
Sensible heat flux per person	43W/m²
Supply air area	0.3 m²
Return air area	0.3 m²
Supply air relative humidity	80 %
Turbulence model	RANS
Regime	Transient
Total simulation time	420 s
Wall area	21 m²
Roof area	36 m²
Area per person	1.7625 m²

The total sensible cooling load was defined as 8.8 kW based on field practices for office applications. The ratio between cooling load and roof and wall areas is 70 W/m^2. With the defined cooling load, the volumetric flow rate at inlet is obtained by:

$$\dot{Q} = \frac{\dot{V}}{\vartheta}(h_i - h_s) \tag{5}$$

where:

$\dot{Q}$ = Total cooling load (*W*);

$\dot{V}$ = volumetric flow rate (*m³/s*);

ϑ = specific volume (*m³/kg-dry air*);

h_i = initial enthalpy of air in the room (*kJ/kg-dry air*);

h_s = supply air enthalpy (*kJ/kg-dry air*).

The velocity at inlet is defined by the cold air supply volumetric rate ($\dot{V}$) needed to balance the cooling load as given by Equation (6):

$$\dot{v} = \frac{\dot{V}}{A} \tag{6}$$

where:

$\dot{v}$ = velocity (*m/s*);

A = supply air area (*m²*).

It can be pointed, that the relative humidity (ϕ) relates to the specific humidity (ω), total pressure (p) and water saturation pressure (p_{g}) at mixture temperature by Equation (7):

$$\omega = 0.622\,\frac{\phi\ \mathrm{p_g}}{\mathrm{p} - \phi\,\mathrm{p_g}} \tag{7}$$

In this way, the supply air specific humidity can be related to latent cooling load by:

$$q_l = \dot{m}_{Ar}\left(\omega_s - \omega_i\right) i_{fg} \tag{8}$$

where:

$\dot{m}_{Ar}$ = supply air mass flow rate (*kg/s*);

ω_s = supply air specific humidity (*kg water/kg-dry air*);

ω_i = initial specific humidity of air in the room (*kg water/kg-dry air*);

i_{fg} = water vaporization enthalpy at supply air temperature.

The reader is referred to Mcquiston *et al*. (2000) for a more detailed discussion of Eq. (5) to (8).

The effect of the cold air distribution on people were investigated by the introduction of three rectangular prisms of 0.25 × 0.25 × 1.70 meters. The prisms are a first approximation to a 70 kilogram human body resulting in a superficial area of 1.8 m^2 As pointed by Nielsen (2002) such simplified models can be used when the bodies act as boundary conditions for global distributions of temperature and air speed in a room.

By its turn, the bodies were defined as surfaces with heat fluxes given by the Brazilian code NBR 16401. According to the code for low level of physical activity an adult person would release 75 W of sensible heat and 55 W of latent heat. It must be pointed that in this study we neglect the contribution of the latent heat in the total cooling load.

The turbulence model employed was the Reynolds Averaged Navier-Stokes Stress (RANS). From preliminary studies it was the best fitted for the simulations.

The simulation was initially set to 10 minutes. However, as steady state was reached before 7 minutes, all the subsequent runs were set to 7 minutes. The cross flow air distribution with inlet and outlet at the same wall (Figure 1a) reached the steady state in 6.45 minutes; the cross flow air distribution with inlet and outlet in opposite walls (Figure 1b) in 6.27 minutes; the bottom distribution in 6.32 minutes and the mixed distribution in 6.18 minutes.

The mesh used in the simulations was hybrid, with tetra and hexahedral cells. The number of cells for distributions (a), (b), (c) and (d) was 127320, 128033, 130613 and 123720, respectively. In Figure 2 is presented a typical grid for the bottom type of air distribution.

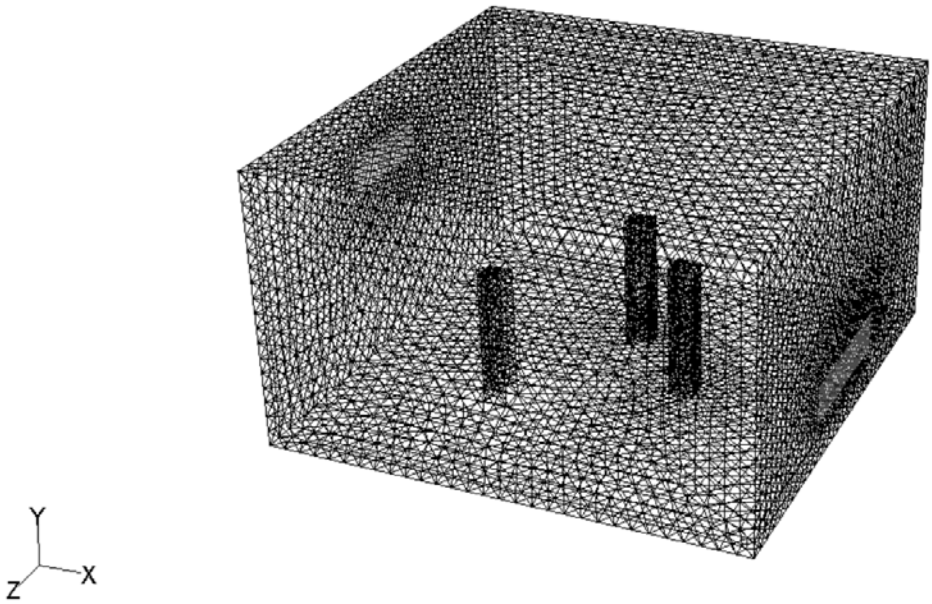

Figure 2 a typical grid distribution

4. RESULTS AND DISCUSSION

The four possible inlet and outlet placement for air were simulated and the results (presented in Figures 3 to 6) clearly highlight the contours of temperature (a and b letters), velocity magnitude (c and d letters) and relative humidity (e and f letters), at two different z planes. The results were compared against the Brazilian standards mentioned previously.

For crossed distribution with air inlet and air outlet at the same wall temperature profiles near the prismatic bodies were outside the range established by the standards, as we can see in Figure 3a and 3b. In this situation, the average temperature was 21 °C with minimum at 20 °C. The velocity profiles were also over the range established by the standard with values up to 1.0 m/s (Figure 3c and 3d). The relative humidity near all the bodies was within 50 to 60% as the standard range (Figure 3e and 3f).

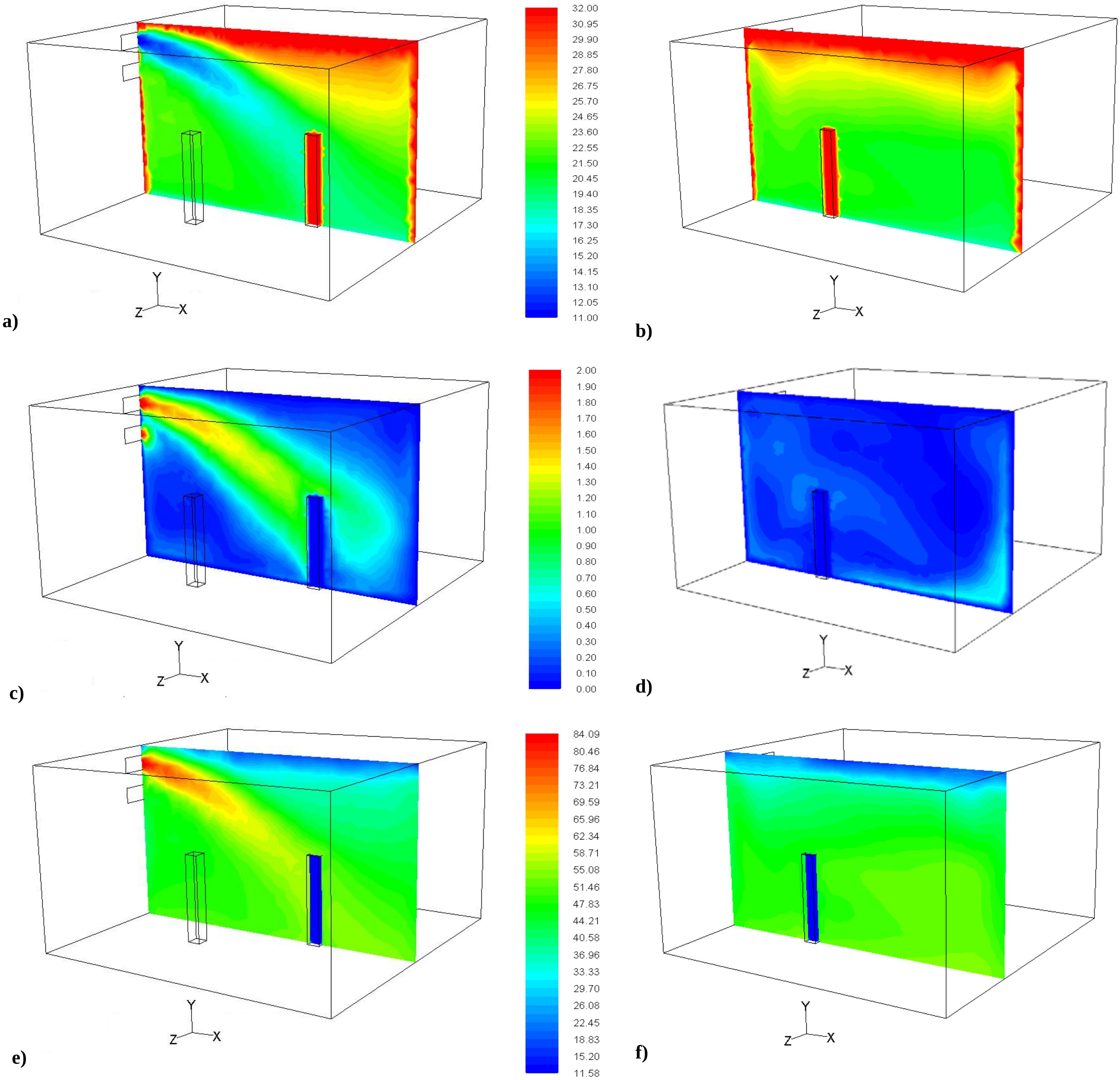

Figure 3. Cross flow with inlet and outlet at the same wall: (a) and (b), contours of temperature (°C) at two different z planes; (c) and (d), contours of velocity magnitude (m/s) at two different z planes; (e) and (f) contours of relative humidity at two different z planes.

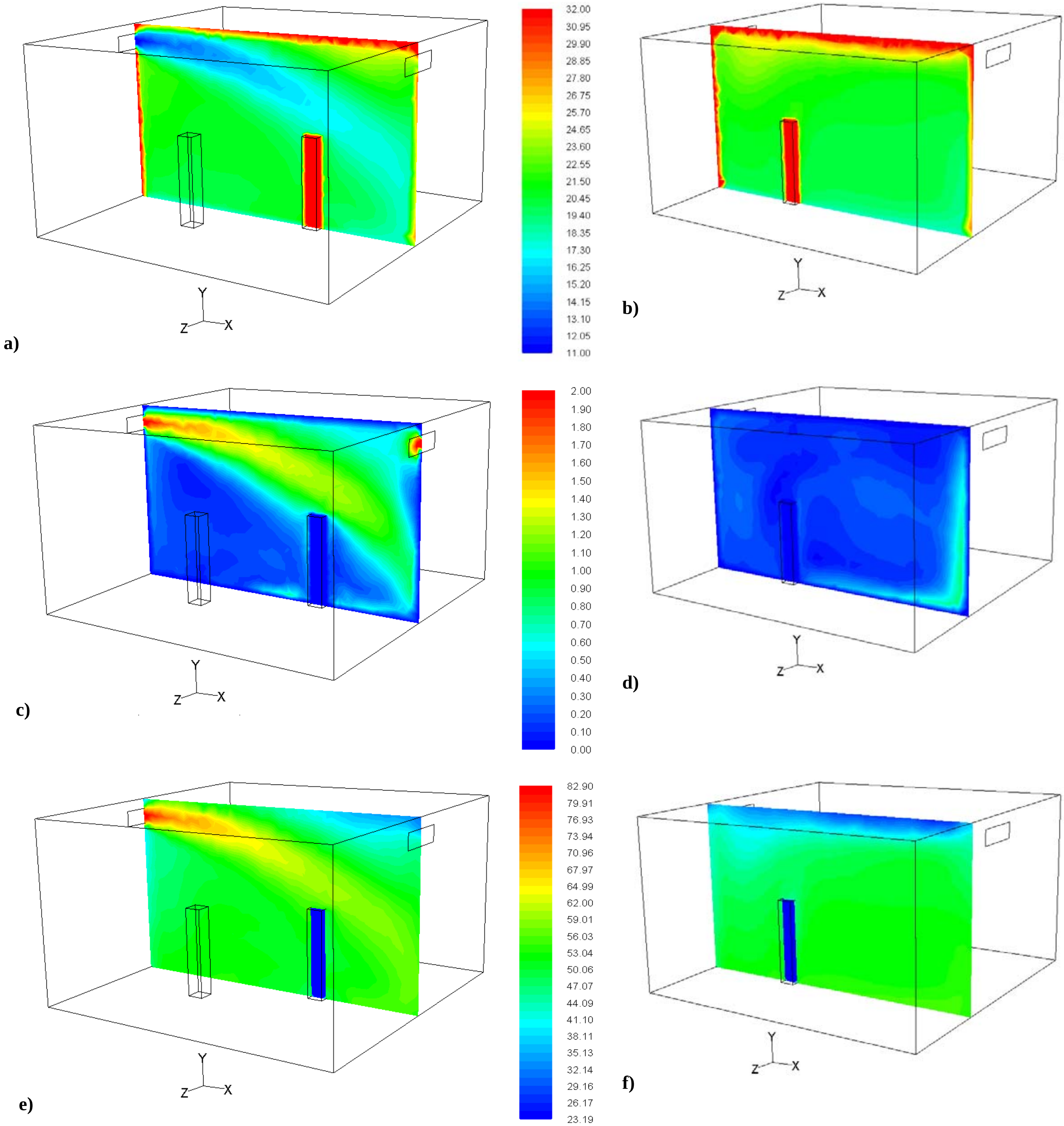

Figure 4 – Cross flow with inlet and outlet in opposite walls : (a) and (b), contours of temperature (°C) at two different z planes; (c) and (d), contours of velocity magnitude (m/s) at two different z planes; (e) and (f) contours of relative humidity at two different z planes.

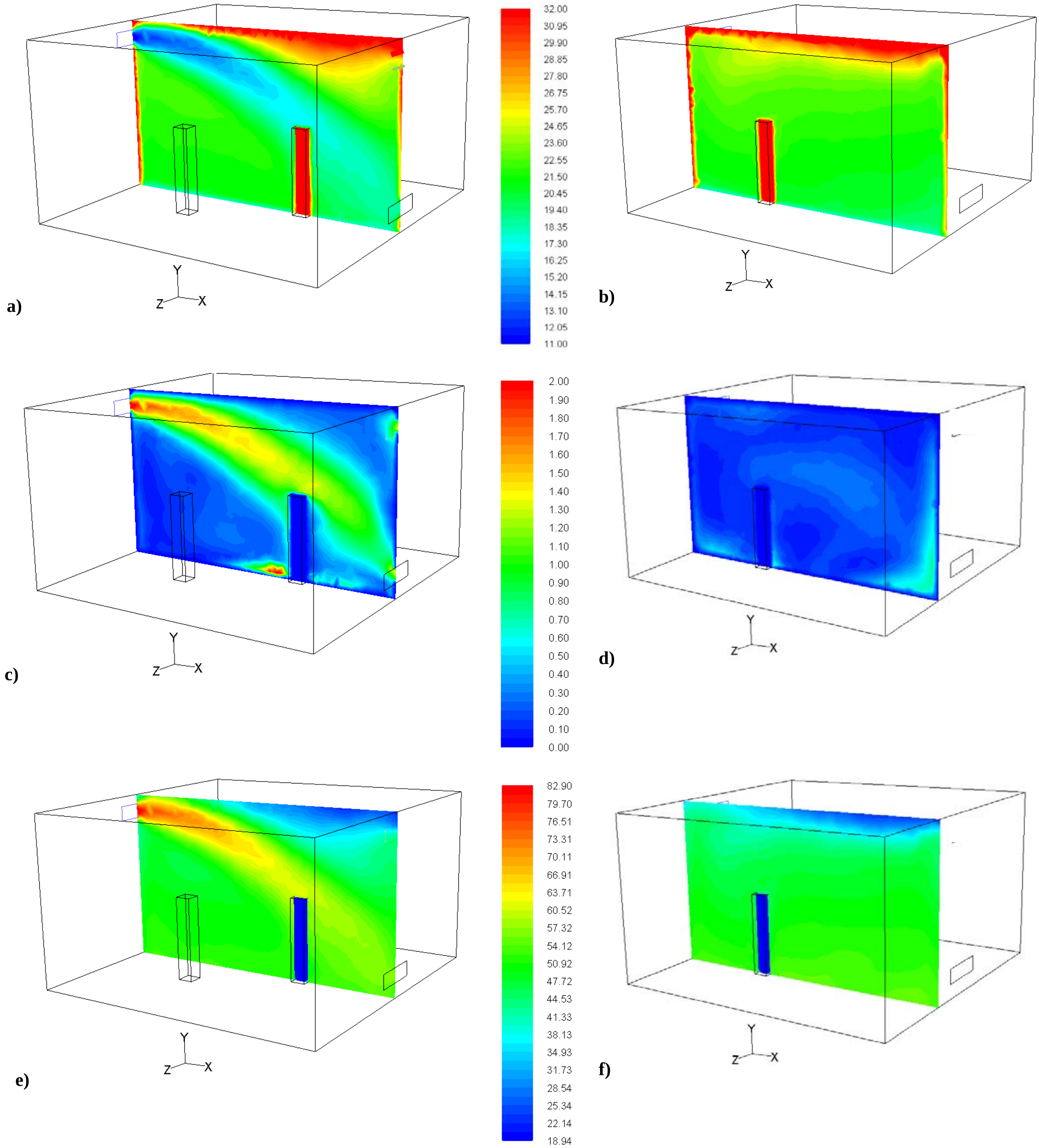

Figure 5 – Bottom: (a) and (b), contours of temperature (°C) at two different z planes; (c) and (d), contours of velocity magnitude (m/s) at two different z planes; (e) and (f) contours of relative humidity at two different z planes.

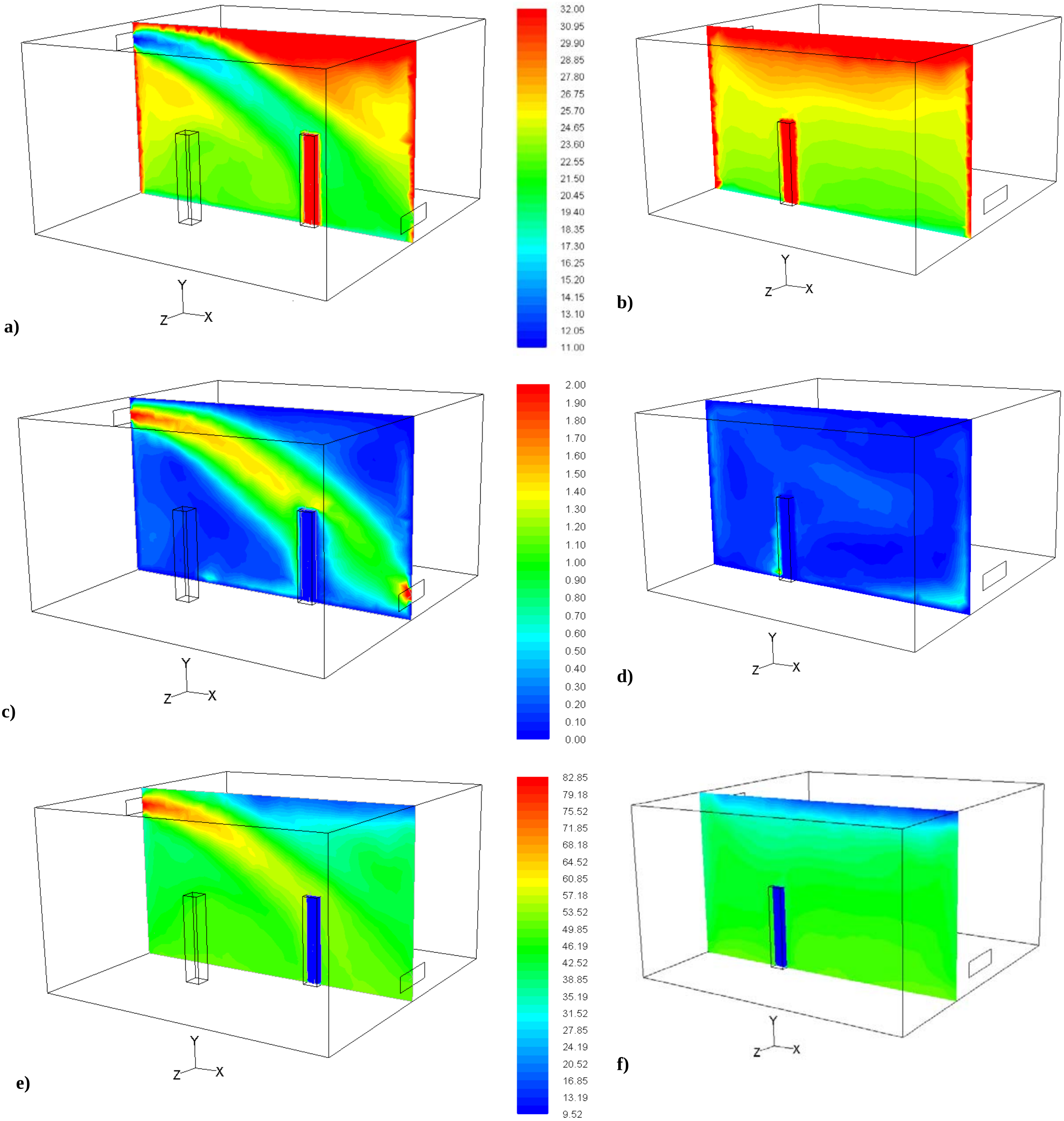

Figure 6. Mixed: (a) and (b), contours of temperature (°C) at two different z planes; (c) and (d), contours of velocity magnitude (m/s) at two different z planes; (e) and (f) contours of relative humidity at two different z planes.

The crossed distribution with air inlet and air outlet in opposite walls presented at Figure 4, also offered non acceptable values for temperature since it was below 20 °C near all the bodies (Figure 4a and b). Furthermore, the velocities near the outlet wall reached 0.6 m/s and above the velocity limit (Figure 4c and d). The relative humidity profile for the room central plane was in the adequate range (Figure 4e). However, for planes crossing the other two bodies and equidistant from the central plane the relative humidity reached 46 % below the limit (Figure 4f).

The third type of inlet and outlet arrangement for our simulation study was the distribution by below (Figure 5). In this case, the temperatures situated underneath the acceptable range near all bodies (Figures 5a and 5b). The velocities also reached higher than the acceptable values near the bodies, mainly in the central plane (Figure 5c and 5d). The relative humidity for the body in the central plane was adequate (Figure 5e). However, for the other two planes; values below 50 % were reached (Figure 5f).

For the mixed distribution showed at Figure 6, the temperature profiles were within the acceptable range for bodies equidistant from the central plane (Figure 6a). However, in the central plane, values below 17 °C were reached (Figure 5b). In addition, the velocities for this distribution of inlet and outlet were above the allowable limits (Figure 6c and 6d). The relative humidity in this case was within the 50 to 60% admissable range in all z planes (Figure 6e and 6f).

It must be noted that in the case of air outlet located at the wall bottom, there is an increase in air velocity in comparison with the other cases. This air outlet location causes a poor air mixing resulting in higher gradients of temperature and relative humidity.

5. CONCLUSION

In the present study was analyzed numerically the effect on some thermal comfort parameters of different inlet/outlet and bodies placement in an example room. The following inlet/outlet distribution was investigated: Cross flow with inlet and outlet at the same wall, cross flow with inlet and outlet in opposite walls, bottom and mixed of air inlet and outlet placement were investigated. The bodies were modeled as prismatic surfaces placed in a central plane and two equidistant planes, respectively. CFD simulations confirm that all the cases did not fully attend the Brazilian standards (ANVISA and ABNT), however, for the mixed distribution, the regions staggered from the central plane are in accordance with the standards. This results concern only the specific cases examined.

As further investigations are in progress, the following effects would be considered: air inlet velocity profile, other internal sources of latent and sensible heat, solar radiation and radiation exchange between the surfaces, ventilator suction, control device, return air profiles.

6. REFERENCES

ABNT. 2008 *NBR 16401 1* Central and unitary air conditioning systems Part 1: Design of instalations 1st edition (In Portuguese). Rio de Janeiro, 91 pp.

Chang, T-J., Y-F. Hsieh, H-M Kao. 2006. Numerical investigation of airflow pattern and particulate matter transport in naturally ventilated multi-room buildings. *Indoor Air*, 16, 136-152.

Chen, Q. 2007. Computer Simulations and Experimental Measurements of Air Distribution in Buildings : Past, Present and Future. *HVAC&R Research; 13, 6, 849-851*.

de Dear, R. J., G. S. Brager, J. Reardon, F. Nicol, 1998. Developing na adaptive model of thermal comfort and preference. *ASHRAE Transactions*, 104, 145-167.

Fanger, P.O. 1972, *Thermal Comfort*, 1st ed., pp. 256. New York: McGraw-Hill Book Co.

Gagge, A.P., A. P. Fobelets, L.G. Berglund. 1986. Standard predictive index of human response to the thermal environment. *ASHRAE Transactions* 92: 709-731.

Gang Li; J. E Braun; S. Frankel; E. A. Groll. 2005. Application of CFD models to two-phase flow in refrigerant distributors. *HVAC&R Research*, 11, 1, 45-62.

Herrlin, M. K. 2008. Airflow and cooling performance of data centers: two performance metrics. *ASHRAE Transactions*, 114, part 2, 182-187.

ASHRAE. 1994. Selecting and preparing a thermal sensation model for use by the profession — 781-RP.

Kessler, M. R. B.; Nicol, J. F. 1998. Perception of comfort in relation to weather and indoor adaptive opportunities. ASHRAE Transactions, v. 104, 1005-1014.

Kowalski, W. J.; W. P. Bahnfleth; Rosenberger, J. L. 2003. Dimensional analysis of UVGI air disinfection systems. *HVAC&R Research*, 9, 3, 347-362.

Lamberts, R.; Xavier, A. A. P. 2000. Indices of thermal comfort developed from field survey in Brazil. *ASHRAE Transactions*, 106, 45-58.

MINISTÉRIO DA SAÚDE. Agência Nacional de Vigilância Sanitária. Brasília, DF, Brasil. RE nº176, 24 de outubro de 2000. Available at: < http://www.anvisa.gov.br/legis/resol/176_00re.htm>. Acessed in 20 mai. 2009.

Mcquiston, F. C., Parker, J. D., Spitler, J., 2000, *Heating, ventilating, and air conditioning: analysis and design*, 5th edition, pp. 223, John Wiley & Sons, USA.

Murakami, S. 2004. Analysis and design of micro-climate around the human body with respiration by CFD. *Indoor Air*, v. 14, n.7, pp. 144-156.

Murakami, S.; S. Kato, T. Kim. 2001.Coupled simulations of convection, radiation and HVAC control for attaining a given PMV value. *Building and Environment*, v. 36, pp. 701-709.

Nielsen, P. V., F.V. Winther, M. Buus, M. Thilageswaran. 2008. Contaminant flow in the microenvironment between people under different ventilation conditions. *ASHRAE Transactions*, v. 114, part 2. 632-638

Nielsen, P. V.; C. Topp, D. Sorensen. 2002. Application of computer simulated persons in indoor environmental modeling. *ASHRAE Transactions*, v. 108, 1084-1089.

Nilsson, H. O.; I. Holmér. 2003. Comfort climate evaluation with thermal manikin methods and computer simulation models. *Indoor Air*, v. 13, 28-37.

Nishihara, A. ; K. Sasao; H. Go; K. Kinoshita; S. Zushi . 1998. Numerical simulation of air conditioning environment in a computer room. *ASHRAE Transactions*, v. 104, 218-225.

Ooka, R. S. Z., T. Sakoi, K. Tsuzuki, S. Kato. 2008. Development of a computational thermal manikin applicable in a non-uniform thermal environment – Part 2: coupled simulation using Sakoi's human thermal physiological model. *HVAC&R Research*, v. 14, n. 4., 545-564

OpenCFD (2008a). OpenFOAM Programmers Guide. OpenCFD Limited.

OpenCFD (2008b). OpenFOAM User Guide. OpenCFD Limited.

Sorensen, D. N., P. V. Nielsen. 2003. Quality control of computational fluid dynamics in indoor environments. *Indoor Air*, 13, p. 2-17

Tung, Y. C., Y-C. Tung, Y-C Shih, S-C Hu. 2009. Numerical study of the dispersion of airborne contaminants from an isolation room in the case of door opening. *Applied Thermal Engineering*, v. 29, pp. 1544-1551.

Umeno, T.; Hokoi, S.; Takada, S. 2001, Prediction of skin and clothing temperatures under thermal transient considering moisture accumulation in clothing. ASHRAE Transactions, v. 107, 71-81.

Yigit, A. 1999. Combining thermal comfort models. *ASHRAE Transactions*, v. 105, 149-155.

Zhu, Q. ; S. Kato, S. Murakami, K. Ito. 2007. 3D-CFD analysis of diffusion and emission of VOCs in a FLEC. *Indoor Air*,