23rd ABCM International Congress of Mechanical Engineering
December 6-11, 2015, Rio de Janeiro, RJ, Brazil

CFD SIMULATION OF THERMAL COMFORT USING INDIVIDUALIZED AIR CONDITIONING SYSTEMS

Rafael Prata Ritzinger
Nathane Ventura de Miranda
Álvaro M. Bigonha Tibiriçá
Alexandre Navarro da Silva
Henrique M. Pereira Rosa
Júlio C. Costa Campos
Universidade Federal de Viçosa , Av. P.H.Rolfs, s/n, Viçosa – MG , Brazil, 36570-900
rafael.ritzinger@gmail.com
nathane.miranda@ufv.br
alvaro.tibirica@ufv.br
alexandre.navarro@ufv.br
henrique.rosa@ufv.br
julio.campos@ufv.br

Abstract. *This study examines the ability to separate a room maintained at a temperature of 25 ºC in zones of different temperatures. CFD simulations were carried out for three regimens to achieve results of the velocity flow lines, the temperature and the Predicted Mean Vote (PMV). To ensure reliable results a mesh convergence test was done, the result stipulated that the elements with a maximum of 2 cm were accepted. All proposed schemes have deployment area located above the occupant's head with a circular opening of 360 mm in diameter. The first scheme consists of a simple opening with vertical insufflation, the second scheme consists of a circular diffuser with a 10 cm high and 1 meter radius bulkhead around the diffuser, the third scheme has both vertically and horizontal insufflations (due to the Coanda effect). Only the air deployment velocity was varied in order to understand its influence in the results. The results showed that it is possible to generate a thermal comfort zone around the occupant using schemes 2 and 3. However, the increased speed to 0,75 m/s causes asymmetry in the zone.*

Keywords: *HVAC, Personalized ventilation, Thermal comfort, Ceiling diffusers.*

1. INTRODUCTION

The custom ventilation aims to provide the occupant with fresh air in the breathing zone, and also allow the user microclimate temperature control (Melikov et al, 2002). The use of individual temperature control in HVAC systems (Heating, Ventilation and Air Conditioning) is a possible solution to improving air quality conditions and thermal comfort (LO et al, 2010). These systems allow fresh air to be filled with a suitable temperature and quality for each occupied zone (Yang et al, 2010).

Makhoul et al (2012) studied the performance of a system that integrates ceiling diffuser and coaxially personalized ventilation. The coaxial nozzle minimizes drag between fresh air and room air, causing the fresh air to reach effectively the occupant. Lo et al (2010) studies the use of localized air flow in order to divide an open office into different zones. Melikov et al (2010) compared the energy savings between systems using custom ventilation with mixed ventilation and desk fan systems. Yang et al (2010) suggest the use of personalized ventilation located in the ceiling for offices in tropical climates. It avoids the use of pipeline systems in the occupied zone.

One way of separating the open environment in different regions with microclimate is changing the geometry of the diffuser as done in the work Makhoul et al (2012) and Sun et al (2004). Creating shields in the roof can help direct the air jet in common ventilation systems that use the Coanda effect to distribute the air in the room. This article compares three custom ventilation schemes with inflating the ceiling to create microclimates around the occupant. The first geometry consists of simply blowing air through an opening located above the occupant. The second uses a common geometry diffuser and a border height of 10cm and 1m radius around the nozzle to keep the air in the area of the occupant. The third case is a mixture of the first two; there is a vertical and horizontal supply air. The objective is using custom ventilation located on the ceiling to create microclimate in order to meet the occupant's satisfaction individually.

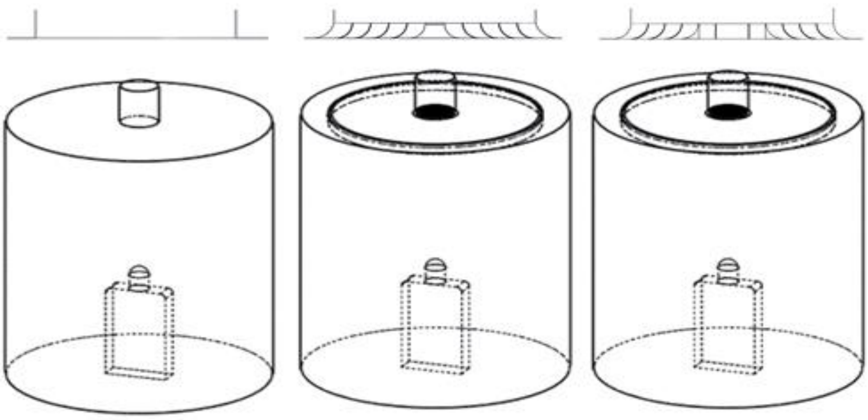
2. METHODOLOGY

This work used CFD modeling to create mesh, change boundary conditions and to simulate the speed, temperature and PMV around the person. The details for the three cases considered are discussed below.

2.1 Region studied and Proposed schemes

The area studied is a cylindrical volume with a radius of 1.25 m and height of 2.8 m. The purpose of this geometry is to analyze the air around an occupant which is located in an environment where the average temperature remains constant. In the three cases cold air is blown through the ceiling by a pipeline with 360 mm in diameter. This standardization aims to reduce the number of variables to be analyzed.

Three simplified air diffusers schemes capable of implementation were proposed in order to create a microclimate around the occupant. Fig 1 shows the schemes.



(a) (b) (c)

Figure 1. Proposed schemes: (a) open insufflation, (b) circular diffuser and (c) diffuser with circular opening in the center.

The first scheme is an opening of 360 mm in diameter, 400 mm length and located 2.8 m above the floor (1.6m above the occupant). A similar experiment was conducted by Yang et al (2010) but with 95 mm nozzle diameter, length of 140 mm and distance between the nozzle and person of 1.3 m. According to Yang et al (2010), the increase in diameter of the outlet will reduce the vertical speed at Breathing Zone thus decreases the cooling effect. However the decrease in diameter will reduce the area of air into the breathing zone and will not cover the entire person. Since the objective of this work is to create a microclimate around the occupant, it is more interesting to use a larger diameter insufflation.

The second scheme is a circular diffuser similar to that used in the work of M. Aziz (2012), but with a larger diameter. A circular edge with a radius of 1 m was placed around the diffuser to direct air down. This edge is 100 mm high and 1 mm thick.

The third scheme has the same values of diameter, length and height of the above. However, part of the air jet descends vertically through the center of the diffuser, while the other part exits at the end of the diffuser in a horizontal direction and is subject to the Coanda effect. In order to increase the air range, a 100 mm barrier was made at a distance of 1 m from the nozzle to direct air downwards. Thus creating a comfort zone around the occupant. A similar technique using different insufflation angles in the diffuser was suggested by Makhoul et al (2012). This technique allows having a desired low temperature in the occupant's microclimate compared to the temperature reached in overall room.

2.2 Mesh generated

The meshes were generated in ANSYS Meshing software. At first, a mesh refinement test was made in order to generate the mesh independence. Fig. 2 shows the points chosen for the test. The points are located in a vertical line at 180mm from the center of the space analyzed. The reason for this choice is because it has a greater temperature gradient and speed. Therefore obtains different values at each point.

Symmetry using ¼ of the geometry has been done in order to reduce computational time. The number of elements in each test was 100538 (100k), 146,002 (146k), 168,042 (268k), 732,337 (732k). The size of the elements in each case was 5 cm, 4 cm, 3 cm and 2 cm, respectively. According to Figs. 3 and 4, the results obtained with 263k and 732k were close to the maximum percentage error of 4.3% for Fig. 3. Therefore, the results with 732k can be considered independent.

The error was calculated according to Equation 1: $Error = \frac{\Delta T_{local}}{\Delta T} \times 100\%$ (1)

where:

ΔT_{local}=*Temperature variation between points with different mesh sizes*

ΔT=*Maximum temperature variation in the environment*

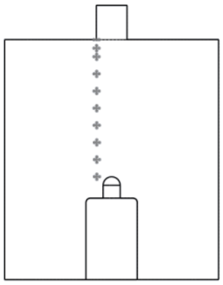


Figure 2. Selected points for the mesh convergence analysis.

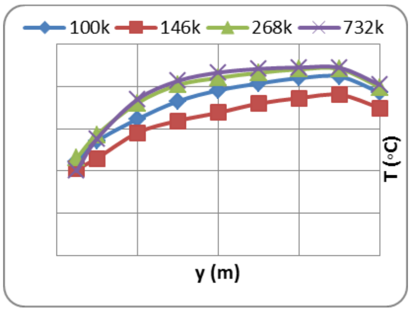


Figure 3. Temperature distribution in the vertical line located 180mm from the center of geometry - Open-inflating.

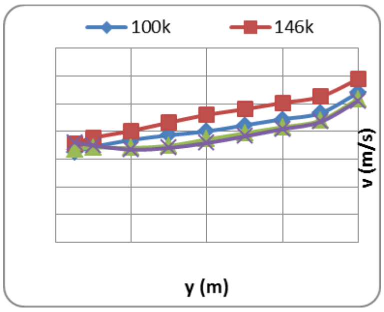


Figure 4. Distribution of the velocity on the vertical line located 180mm from the center of geometry - Open-inflating.

Figure 5 shows the end of the meshes generated simulation. An influence body was used for local refinement to the scheme 1 shown in Fig. 5.a. The refinement zone was generated with tetrahedral elements with dimensions of 0.02 m and grain growth ratio of 1.2. Inflation was used in the ceiling, floor and the inlet duct. Following the parameter A. Makhoul et al (2012), the inflation has a total of 4 layers, the first thickness being 1.5mm and growth ratio of 1.2. All simulations were made using symmetry conditions.

Figure 5. Mesh generated in the three geometries. Maximum element size of 2 cm in the regions of interest

2.3 CFD models

2.3.1 Governing equations

The basic CFD indoor airflow equations are shown in table 1.

Table 1 – Mathematical model employed in the simulations.

Conservation equation of continuity	$\frac{\partial u_i}{\partial t} = 0$
Conservation equation of momentum	$\frac{\partial u_i}{\partial t} + \frac{\partial u_i u_j}{\partial x_j} = \frac{1}{\rho} \cdot \frac{\partial \left(p + {}^{2\rho k}/_{3}\right)}{\partial x_i} + \frac{\partial u_i}{\partial x_j}\left\{(\nu + \nu_t)\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right\} - \beta g_i \theta$
Conservation equation of energy for a turbulent buoyancy incompressible fluid	$\frac{\partial \theta}{\partial t} + \frac{\partial \theta u_j}{\partial x_j} = \frac{\partial}{\partial x_j}\left\{\left(\kappa + \frac{\nu_t}{\sigma_\theta}\right)\frac{\partial}{\partial x_j}\right\} + h(x_i, t)$
Transport equations for turbulent kinetic energy and its dispersion rate	$\frac{Dk}{Dt} = \frac{\partial}{\partial x_j}\left\{\left(\nu + \frac{\nu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right\} + \nu_t\left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j}\right)\frac{\partial u_i}{\partial x_j} + \beta g_i \frac{\nu_t}{\sigma_\theta}\frac{\partial \theta}{\partial x_i} - \varepsilon$
	$\frac{D\varepsilon}{Dt} = \frac{\partial}{\partial x_j}\left\{\left(\nu + \frac{\nu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right\} + \frac{\epsilon}{k}\left\{C_1 \nu_t\left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j}\right)\frac{\partial u_i}{\partial x_j} - C_2\varepsilon + C_3 \beta g_i \frac{\nu_t}{\sigma_\theta}\frac{\partial \theta}{\partial x_i}\right\}$

where:

u_i = Mean velocity componentes in the x_i direction.
x_i = Cartesian coordiantes.

t = Time.
p = Mean static pressure difference.

ρ = Fluid density.
k = Turbulence kinetic energy.
v = Fluid kinematic viscosity.
v_t = Fluid turbulent viscosity.
β = Fluid volumetric coefficient of expansion.
g_i = Gravitational acceleration in the x_i direction.
θ = Difference between average indoor and local temperatures
κ = Thermal diffusivity.
h = Mean volumetric heat generation rate.
ε = Dissipatin rate of turbulence energy.
$C_D, C_1, C_2, C_3, \sigma_k, \sigma_\varepsilon$ *and* σ_θ = Empirical constants
$C_D = 0.09$
$C_1 = 1.44$
$C_2 = 1.92$
$C_3 = 1.0$
$\sigma_k = 1.0$
$\sigma_\varepsilon = 1.3$
$\sigma_\theta = 0.9$

2.3.2 Contour conditions

Tables 2 and 3 show the insufflation boundary conditions and general boundary conditions for all three schemes, respectively.

Table 2. Conditions of insufflated air of the three cases. Temperature at the inlet = 18 °C.

Case	Inlet air velocity (m/s)	Air flow (Kg/s)
1	0.25	0.008
	0.5	0,015
2	0.5	0,015
	0.75	0,023
3	0.5	0,015
	0.75	0,023

Table 3. General boundary conditions for the three cases.

Air opening	
Static pressure (Pa)	0
Total area (m^2)	21,98
Ceiling and floor	Adiabatic
Person	34 °C
Air temperature in the environment	25 °C
Fluid properties	Ar a 25 °C
Buoyancy	Yes*
Turbulent model	k –ε
Energy model	Thermal Energy
Simulation regime	Steady state
Stop criteria	Error RMS ≤ 10^{-4}

(1) Bouyancy considered by Boussinesq model.

The most important parameters in obtaining induced comfort are the air flow rate and the temperature (Makhoul A., 2012). With this in mind, three simulations were made for each case. Modifying the intake air speed at 0.25 m/s, 0.5 m/s and 0.75 m/s. And maintaining the inlet temperature at 18 ° C. The environment around the area of interest was maintained at 25 ° C. The turbulence model k –ε was chosen to solve the Navier-Stokes equation. It provides a better prediction for the spreading rate of jets as well as a more reliable solution for flows that are related to separation and recirculation (Makhoul A., 2012).

2.4 Thermal comfort index

Through data tabulated by Fanger (1982), an iterative equation was made by curve adjustment to calculate the PMV. Equation 2 shows minimal correlation coefficient of 0.9941 and it has been implemented in the software to generate contours of that index over the plane.

$$PMV = (0{,}3259V^{0{,}1193})T - 1{,}171\ln(V) - 8{,}32 \quad (2)$$

V = Air velocity [m/s]
T = Air temperature [°C]
The relative humidity was considered 50%, the thermal resistance of clothing as 0.75 clo and the metabolic rate as 60 $Kcal/m^2hr$.

2.5 Result analysis

The analysis was done with thermal comfort indicators used by Makhoul et al (2012). It consists of taking the average of the speed and temperature values at a distance of 5 cm from the surface of the head.

There was also an analysis of PMV between -1 and 1 (region considered good for thermal comfort) located in the plane at 1 m horizontal from the floor.

3. RESULTS AND DISCUSSION

This section will detail the simulation results obtained in the three schemes described in the methods section. The schemes were separated into sections to better compare the results. Each of the schemes generated two different insufflation speeds, totaling six simulations. Using the software it was possible to generate contours of air flow velocity, temperature and thermal comfort index (PMV). Figs. 5, 6, 7, 8, 9 and 10 represent the vertical and horizontal planes generated. The horizontal planes are located at 0.5m and 1m from the floor and vertical planes are located in the symmetry region. The results were collected and analyzed.

3.1 Scheme 1

Figs. 6 and 7 represent the two cases for this scheme with insufflation speeds of 0.25 and 0.5 m/s, respectively. To facilitate the approach Fig. 6 will be considered as case 1 and Fig. 7 as case 2. The values of the average speed achieved in the region near the head (5 cm) were 0.63 and 0.85 m / s for cases 1 and 2, respectively. The air velocity is greater than the value inflated due to the buoyancy effect. The average temperature values obtained in the region near the head (5 cm) were 21.1 and 19.7 ° C for the inflating speed of cases 1 and 2, respectively. It is possible to observe a decrease of temperature with increasing speed insufflating

The respective PMV values for cases 1 and 2 are -1.22 and -1.77 which makes them considered to be cold areas. The horizontal plane area at 1 m height with PMV between -1 and 1 (comfort) are 0.33 m² and 0.36 m² for cases 1 and 2 respectively. These values are low compared to schemes 2 and 3 (about 7 times lower), therefore it will be disregarded when analyzing the results.

Fig. 6 (a) and 6 (b) show the outline of speed and temperature, respectively. It may be noted that regions with greater speed and lower temperature generate a PMV of -2 to -3 (FIG. 5 (c)), and is considered to be a cold zone. While the regions with low speed and high temperature generates a PMV between 2 and 3, which is considered to be hot. In general, the contours of Fig.5 are similar to those of Fig. 6.

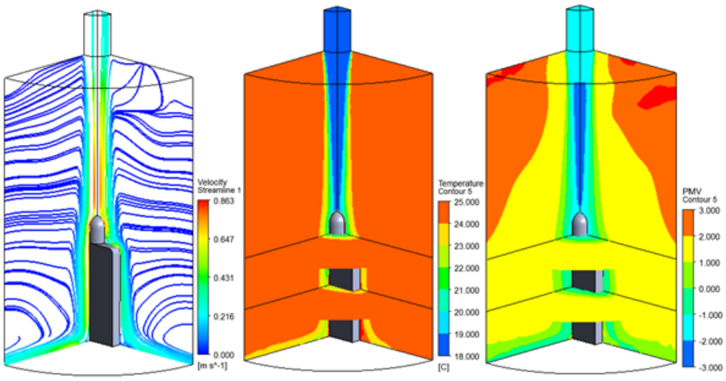


Figure 6. Scheme 1 (insufflation speed of 0.25 m/s):
(a) Velocity flow line, (b) temperature and (c) thermal comfort (PMV).

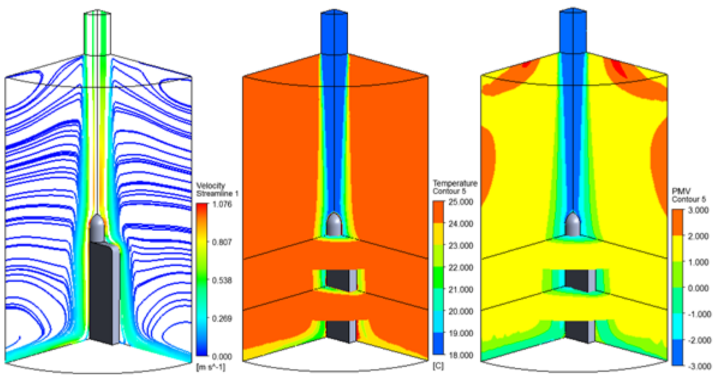


Figure 7. Scheme 1 (insufflation speed of 0.5 m/s):
(a) Velocity flow line, (b) temperature and (c) thermal comfort (PMV).

3.2 Scheme 2

Figures 8 and 9 represent the two cases for this scheme with insufflation speeds of 0.75 and 0.5 m/s, respectively. To facilitate the approach Fig. 8 will be considered as case 3 and Fig. 9 as case 4. The values of the average speed achieved in the region near the head (5 cm) were 0.08 and 0.23 m/s for cases 3 and 4, respectively. The average temperature values obtained in the region near the head (5 cm) were 24.8 and 24.3 ° C for cases 1 and 2, respectively. These values showed a variation of 0.5 ° C, and is considered small compared to the total variation in temperature (7 ° C). The respective PMV values for cases 3 and 4 were 0.68 and 0.08 which are considered to be comfort zones. The horizontal plane area at 1 m height with PMV between -1 and 1 (comfort) are 2,585 m² and 2,612 m² for cases 3 and 4 respectively.

It is possible to see the contour of the flow velocity shown in Fig. 9 (a), the geometry of the occupant generates greater influence on the air direction compared to the first case. This directly affects the temperature values and PMV. The contours are not symmetrical and have 1 to 2 °C difference between the temperature of the air in the front and the longitudinal axis of the occupant. The PMV between -1 and 1 is greater than 1 m away from the dummy in the front axle and about 55 cm away from the dummy in the longitudinal axis, measured at the shoulder.

In Fig. 9 (a), the air flow velocity helps to explain the asymmetry generated in Figure 9 (b), which refers to temperature. The increase in supply air along with the geometry and distance from the ceiling of the dummy causes the speed contour lines to suffer deviation. Thus altering the thermal comfort zone of Fig. 9 (c). There were visually and numerically significant changes between the cases shown in Figs. 7 and 8.

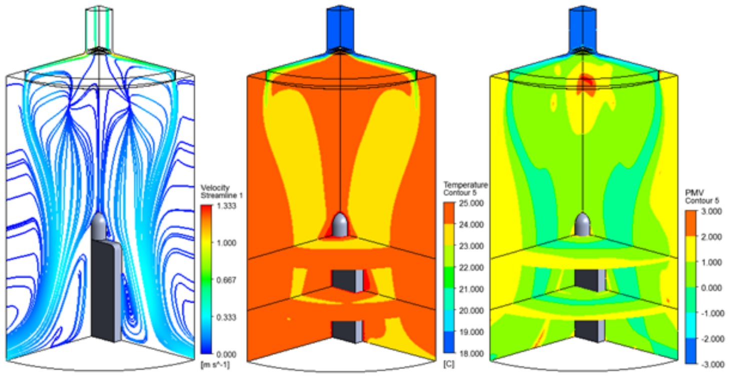


Figure 8. Scheme 2 (insufflation speed of 0.5 m/s):
(a) Velocity flow line, (b) temperature and (c) thermal comfort (PMV).

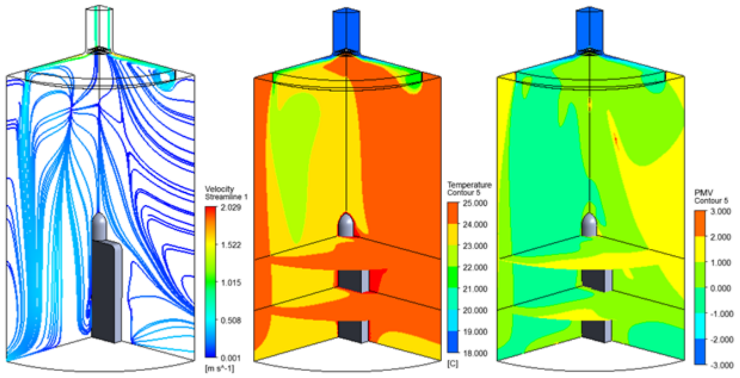


Figure 9. Scheme 2 (insufflation speed of 0.75 m/s):
(a) Velocity flow line, (b) temperature and (c) thermal comfort (PMV).

3.3 Scheme 3

Figures 10 and 11 represent the two cases for this scheme with insufflation speeds of 0.75 and 0.5 m/s, respectively. To facilitate the approach Fig. 10 will be considered as case 5 and Fig. 6 as case 11. The values of the average speed achieved in the region near the head (5 cm) were 0.54 and 0.64 m/s in cases 5 and 6, respectively. The average temperature values obtained in the region near the head (5 cm) were 22.7 ° C for the two cases. It is possible to observe that despite the increase in insufflation velocity there was no variation in temperature. The PMV values for cases 5 and 6 were -0.67 and -0.71 and are considered to be within the comfort zone. The horizontal plane area at 1 m height with PMV between -1 and 1 (comfort) are 1.813 and 1.855 in cases 5 and 6, respectively. Although this value is close, case 5 showed greater symmetry in the velocity contours, temperature and PMV when compared to case 6.

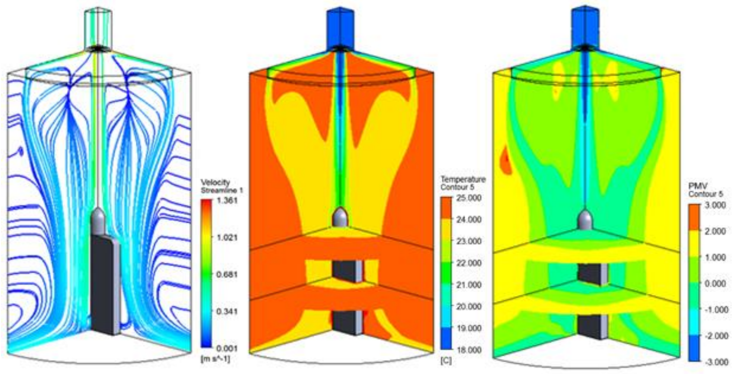


Figure 10. Scheme 3 (insufflation speed of 0.5 m/s):
(a) Velocity flow line, (b) temperature and (c) thermal comfort (PMV).

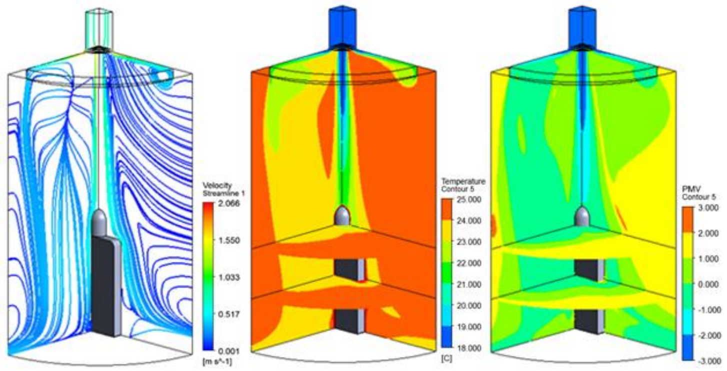


Figure 11. Scheme 3 (insufflation speed of 0.75 m/s):
(a) Velocity flow line, (b) temperature and (c) thermal comfort (PMV).

3.4 Discussion

The area considered to have thermal comfort (PMV between -1 and 1) in the horizontal region of 1 m from the floor is described in Table 4 for each of the proposed schemes. Table 4 also shows that the increase in the inflating velocity caused an increase of the area of thermal comfort at that zone. By comparing the three schemes with inflating velocity of 0.5 m/s and insufflation temperature of 18 ° C, scheme 2 obtained 2.585 m², promoting a higher thermal comfort area around the user. Scheme 1 showed an area about 7 times smaller than schemes 2 and 3. Since the region concerned for scheme 1 undergoes variations of 7 ° C at a distance of 30 cm, this area was not considered as comfort, therefore it was not shown in Tab. 4. The largest area of thermal comfort in the horizontal plane at 1 m height from the floor was in scheme 2 with 0.75 m/s insufflation velocity. Table 3 shows that using scheme 2 with 0.75 m/s of insufflation velocity it was possible to obtain the best thermal comfort index (0.08) and velocity values of 0.23 m/s at the region close to the head (5 cm away). When the insufflated air velocity is low, schemes 2 and 3 show better symmetric contours of temperature and PMV around the occupant.

Table 4. Average temperature, speed and PMV in the region near the head for the three schemes and comparison of thermal comfort zone (PMV) between schemes.

Schemes	Insufflated air velocity [m/s]	Average temperature close to head [°C]	Average air velocity close to head [m/s]	Average PMV close to head	Comfort area (-1 a 1) on the horizontal plane at 1 m height. (m^2)
1	0.25	21,1	0,63	-1,22	-
	0.5	19,7	0,85	-1,77	-
2	0.5	24,8	0,08	0,68	2,585
	0.75	24,3	0,23	0,08	2,612
3	0.5	22,7	0,54	-0,67	1,813
	0.75	22,7	0,64	-0,71	1,855

4. CONCLUSION

Three custom ventilation schemes have been proposed. All of them with air inflation from the ceiling. In the first scheme there was a variation of 7 ° C in temperature and -3 to 0 of thermal comfort levels in the region at a distance of 20 cm from the occupant, for the two simulated cases. However the other two schemes verified the formation of a comfort zone (PMV between -1 and 1) around the occupant (radius less than 1 m from the symmetry axis of the occupant). In all schemes, the inflating speed of the air was varied. It was noted that the increase in speed caused asymmetry in the comfort zone for schemes 2 and 3. Overall, the results showed that both scheme 2 and scheme 3 may be used to provide thermal comfort using personalized ventilation.

It is expected that the variation of the air insufflation temperature will keep the boundary region similar to the results obtained, having a slight variations due to buoyancy effect and with different values of PMV. The height and the distance between the screen and the center axis of the diffuser are factors that can be analyzed in future studies in order to verify the effects they cause.

5. ACKNOWLEDGMENTS

To Fapemig, CAPES, CNPq and Funarbe for the financial help for the research group.

6. REFERENCES

AZIZ M. A.; GAD I. A. M.; MOHAMMED E. S. F. A.; MOHAMMED R. H.“Experimental and numerical study of influence of air ceiling diffusers on room air flow characteristics”. Energy and buildings (2012).

CAO G.; AWBI H.; YAO R.; FAN Y.; SIRÉN K.; KOSONEN R.; ZHANG J. “A review of the performance of different ventilation and air flow distribution systems in buildings”. Building and environment (2013).

CHEN Q.; SREBRIC J.“A Procedure for Verification, Validation, and Reporting of Indoor Environment CFD Analyses”. HVAC&R Research (2011).

FANGER P. O.“Human requirements in future air-conditioned environments”. International Journal of Refrigeration (2000).

FANGER P. O. “Thermal Comfort”. Robert E Kreiger Publishing Company, Malabar, FL (1982).

GAO N.; NIU J.“CFD study on micro-environment around human body and personalized ventilation”. Building and environment (2004).

LAM J. C.; CHAN A. L. S. “CFD analysis and energy simulation of a gymnasium”. Building and Environment (1999).

LO L. J.; NOVOSELAC A.“Localized air-conditioning with occupancy control in an open office”. Energy and Buildings (2010).

MAKHOUL, A.; GHALI, K.; GHADDAR, N.“Thermal comfort and energy performance of a low-mixing ceiling-mounted personalized ventilator system”. Building and Environment (2012).

MAKHOUL A.; GHALI K.; GHADDAR N.“Desk fans for the control of the convection flow around occupants using ceiling mounted personalized ventilation”. Building and Environment (2012).

MELIKOV A. K.; CERMAK R.; MAJER M.“Personalized ventilation: evaluation of different air terminal devices”. Energy and Buildings (2002).

SMITH T. F.; SUN Y.“Air flow characteristics of a room with square cone diffusers”. Building and Environment (2004).

SUN Y.; THEODORE F.; SMITH. “Air flow characteristics of a room with square cone diffusers”. Building and Environment (2004).

YANG B. “Thermal comfort and indoor air quality evaluation of a ceiling mounted personalized ventilation system integrated with an ambient mixing ventilation system.” National University of Singapore (2009).

YANG B.; SEKHARA C.; MELIKOVB A. K.“Ceiling mounted personalized ventilation system in hot and humid climate—An energy analysis.” Energy and Buildings (2010).

7. RESPONSIBILITY NOTICE

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