DEVELOPMENT AND APPLICATION OF A VENTILATION SYSTEM BASED ON VERTICAL DESCENDENT CONFLUENT JETS

Eusébio CONCEIÇÃO*[1], João GOMES[2], Mª. Inês CONCEIÇÃO[3], Mª. Manuela LÚCIO[1] and Hazim AWBI[4]

[1] FCT – Universidade do Algarve, Faro, Portugal
[2] CINTAL, Faro, Portugal
[3] IST – Universidade de Lisboa, Lisboa, Portugal
[4] School of Built Environment – University of Reading, Reading, United Kingdom
* *Corresponding author: econcei@ualg.pt*

ABSTRACT

This paper presents the development and application of a ventilation system based on vertical confluent jets. The thermal comfort and indoor air quality levels, Air Distribution Index and energy consumption are evaluated and discussed. The numerical study is carried out in a virtual chamber with dimensions of $4.50\times2.55\times2.50$ m^3. This chamber is equipped with six tables, twelve chairs, one outlet system and one confluents jets system, and is occupied with twelve virtual occupants. The inlet system has two horizontal 0.15 m diameter ducts, installed at a height of 1.8 m from the floor, which have consecutive holes in order to promote downward jets close to the side walls. The outlet system has six air ducts, located above the head of the occupants, connected to the ceiling area. The study was developed for three different airflow rates, considering winter conditions. When the airflow rate increases, indoor air quality improves, thermal comfort remains within an acceptable level and ADI improves slightly.

INTRODUCTION

The confluent jets are associated to a system of multiple jets that, after being inflated, converge in a single airflow to the exhaust ventilation system. In its design different techniques and methodologies are used (Arghand et al., 2015; Karimipanah et al., 2000; Cho et al., 2008). In general, in this type of ventilation system, the inlet is made through lines made up of consecutive nozzles and the outlet is made through an exhaust system located on the ceiling of the compartment.

The evaluation of occupant comfort is made by the levels of thermal comfort and indoor air quality, the local discomfort of the occupant is made by the Draught Risk (DR) and the performance of the Heating Ventilating and Air-Conditioning (HVAC) system is made by the Air Distribution Index (ADI).

The Predicted Mean Vote (PMV) and Predicted Percentage of Dissatisfied (PPD) indexes, developed by Fanger (1970), are used to evaluate the thermal comfort level (ISO 7730:2005). ISO 7730:2005 defines three categories to characterize thermal comfort: A (PPD $\leq$ 6%); B (PPD $\leq$ 10%); C (PPD $\leq$ 15%). The carbon dioxide (CO_2) concentration can be used to evaluate the indoor air quality (Conceição et al., 2008a). In this numerical work, CO_2 concentration release by the occupants is used as evidential of the indoor air quality in occupied spaces (ASHRAE 62.1:2016). ASHRAE 62.1:2016 refers 1800 mg/m^3 as the acceptable limit for CO_2 concentration. DR is an index developed by Fanger et al. (1988). It depends on the air temperature, air velocity and air turbulence intensity. ISO 7730:2005 defines three categories to characterize DR: A (DR $\leq$ 10%); B (DR $\leq$ 20%); C (DR $\leq$ 30%). ADI depends on the air quality level, thermal comfort level, contaminants removal efficiency and heat removal efficiency. It was presented and detailed in the works of Awbi (2003), for uniform environments, and Conceição et al. (2013), for non-uniform environments.

The present numerical study is based on a coupling between Computer Fluid Dynamics (CFD) and the Human Thermal Response (HTR) numerical models, whose an application of the coupling methodology can be seen, as example, in Conceição & Lúcio (2016). This methodology can be seen in the works of Conceição (2000), Conceição & Lúcio (2001), and Conceição et al. (2007, 2010a).

Some of the input data of the numerical models coupling is obtained from the output data of a Building Dynamics Response (BDR) software. The works of Conceição et al. (2000, 2008b) and Conceição & Lúcio (2009, 2010a, 2010b) present applications of the BDR software in the evaluation of air temperature distribution, surfaces temperature distribution and energy consumption. This software takes into account the evaluation of thermal comfort using the PMV/PPD indexes (Conceição et al., 2018), the adaptive thermal comfort (Conceição et al., 2010b), and the temperature preferred control model (Conceição et al., 2009). The evaluation of the air temperature and air velocity around the occupants is required for the assessment of the thermal comfort, whose methodology applied to buildings using numerical techniques can be seen in Conceição & Lúcio (2016).

This numerical work uses a coupling of a differential (CFD) and an integral (HTC) software, in conjunction with a third integral (BDR) software, to numerically evaluate a ventilation system of vertical confluent jets with the novelty of using ducts with a long line of nozzles. The objective is to achieve an efficient distribution of the air blown in the room in order to improve the levels of thermal comfort and indoor air quality for the occupants, with a low DR values, while improving the performance of the HVAC system. The coupling of CFD and HTR software is used to simulate the airflow around the occupants and the human body and clothing temperatures distribution. The BDR is used to calculate the temperature of the surrounding surfaces of the virtual chamber. This study was developed for three airflow rates, considering winter conditions

NUMERICAL MODEL

In this work, a numerical model consisting of two other models was applied: one, a coupling of a differential numerical model, CFD (Figure 1), and an integral numerical model, HTR (Figure 2); two, an integral numerical model, BDR.

The differential CFD model evaluates the air velocity, air temperature, air turbulence intensity and CO_2 concentration. The numerical model, which simulates high occupancy levels, simulates the three-dimensional airflow in Cartesian coordinates. These equations are of mass continuity, moment, energy, turbulence kinetic energy, turbulence energy dissipation rate and contaminants concentration. The human body is divided into 25 boxes by this numerical model. More details can be seen in the works of Conceição & Lúcio (2001) and Conceição et al. (2013). In the present work, DR around the occupants, thermal comfort, air quality and ADI are assessed.

The CFD numerical model, which works in steady-state conditions, considers:

- The non-isothermal thermal conditions;
- The RNG, for high Reynolds number, turbulence model;
- The partial differential equations solved by the finite volume method;
- The hybrid scheme used in the convective and diffusive fluxes;
- The SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithm used in the velocity and pressure equations;
- The non-uniform methodology used in the grid generation;
- The grid refined near the surfaces and in the airflow inlet and outlet;
- The density effect negligible in the temperature equation;
- The vertical air velocity equation considering the impulsion term;
- The carbon dioxide equation considering the source term in the breathing zone;
- The iterative TDMA (Tri-Diagonal Matrix Algorithm) method used in the equations system resolution;
- The surface proximity considers the wall boundary.

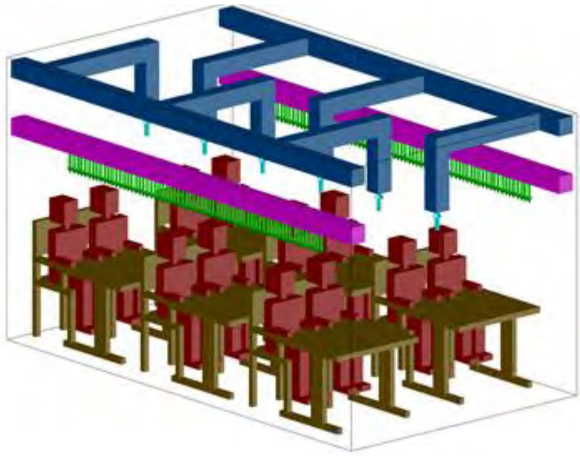


Figure 1. Scheme of the virtual chamber, equipped with confluent jets ventilation system, used in the CFD: inlet, green arrows; outlet, blue light arrows

The integral HTR model, which simulates the human thermoregulatory response and the thermal response of the human body and clothing systems, evaluates the temperature distribution in the tissues, blood and clothing. The human body is divided into 24 cylindrical elements and 1 spherical element by HTR model. Each element is constituted by several concentric layers and it can also be protected by several layers of clothing. More details can be seen in the works of Conceição (2000), Conceição & Lúcio (2001) and Conceição et al. (2007, 2010a, 2013).

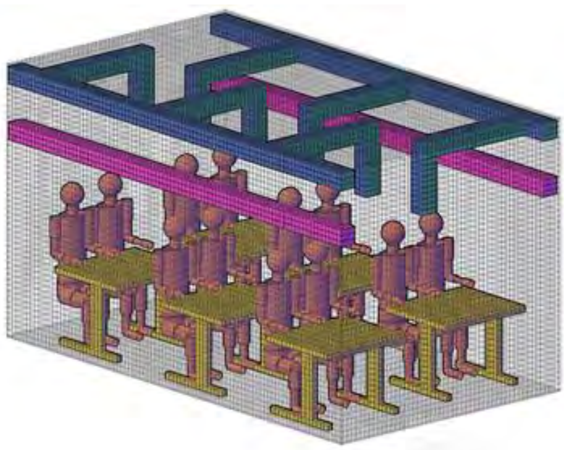


Figure 2. Scheme of the virtual chamber, equipped with confluent jets ventilation system, used in the HTR

The integral BDR model, which is used to simulate the virtual chamber thermal response, evaluates the indoor air temperature distribution, interior surrounding surfaces temperature distribution and thermal energy consumption. This numerical model separates the building into opaque (ceiling, walls, floor and interior bodies) and transparent (windows) surfaces.

The three numerical models presented in this work, are based on a coupling between CFD and HTR numerical models, with data input from the BDR. A validation that uses the three numerical models simultaneously can be seen in Conceição & Lúcio (2016). These validation was done using experimental and numerical values of the chamber surface temperature, air temperature, air velocity, air turbulence intensity and Draught Risk around the occupants.

However, the three numerical simulation was validated, also, individually. The CFD was validated in a study of airflow inside office compartments with moderate environments in the work of Conceição et al. (2008c). The HTR, as example, in a numerical and subjective responses of human thermal sensation, was validated in Conceição & Lúcio (2001). Finally, the BDR, as example in summer conditions, for complex topology school buildings, was validated in Conceição & Lúcio (2006).

This numerical work considers three numerical simulations: two integral, as HTR and BDR, and one differential, as CFD. The methodology considered is as follows:

1. The BDR calculates the temperature of the surrounding surfaces;
2. The CFD uses the output data of the BDR and HTR as input data;
3. The CFD calculates the environmental variables around the occupants;
4. The HTR uses the output data of the BDR and CFD as input data;
5. The HTR calculates the body and clothing temperature;
6. The iterative method uses sequentially the steps 2 to 5 and stops when the convergence is acceptable.

The HVAC system performance is assessed by the ADI. This index considers the levels of thermal comfort and air quality of each occupant, as well as the effectiveness of the ventilation system in removing heat and contaminants from the interior space.

NUMERICAL METHODOLOGY

The numerical study is carried out in a virtual chamber (Figures 1 and 2). This virtual chamber, which simulates a real experimental chamber, is 4.50 m long, 2.55 m wide and 2.5 m high. The simulated scenario corresponds to a classroom equipped with six tables and twelve chairs, and occupied by twelve virtual people seated. The location of the virtual occupants and their identification number are shown in Figure 3.

The inlet ventilation system installed in the virtual chamber is founded on a confluent jets system constituted by two horizontal ducts (represented by purple colour in Figure 1) equipped with a row of consecutive nozzles (outlet of the air jets represented by the green arrows in Figure 1). These ducts are placed next to the side walls in order to promote vertical jets close to them. The ducts are 3.3 m long, 0.125 m in diameter and are placed 1.8 m high from the floor. The exhaust ventilation system is installed in the central zone of the chamber, consisting of six ducts (shown in blue in Figure 1), 0.125 m in diameter, located above the level of the occupants' head (Figure 1).

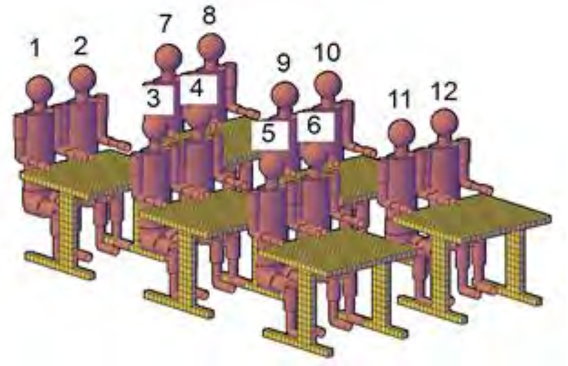


Figure 3. Location of the virtual occupants and their identification number

The numerical simulations were done for a typical winter day characterized by an outdoor air temperature of 0°C, an average indoor air relative humidity of 50% and an average indoor air temperature of 20°C. In these conditions, a clothing level of 1.0 clo and an activity level of 1.2 met were considered (ISO 7730: 2015).

The numerical simulation was developed in order to study the effect of the airflow rate variation on the levels of thermal comfort and indoor air quality of the occupants, as well as on the performance of the proposed ventilation system. Therefore, three airflow rate values were defined according to the recommended value for the number of occupants: 0.1167 m^3/s for 12 occupants (Case A); 0.2333 m^3/s for 24 occupants (Case B); 0.3500 m^3/s for 36 occupants (Case C). Other input data from the numerical simulation, for each of the three Cases defined above, are presented in Table 1.

Table 1. Input data from the numerical simulation for each Case studied

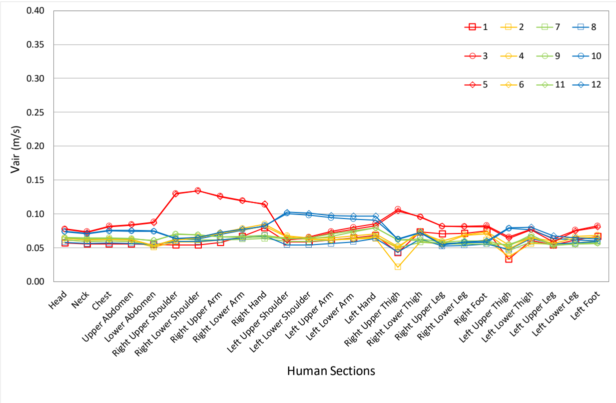
Case	A	B	C
Inlet air velocity (m/s)	0.7	1.4	2.1
Inlet air temperature (°C)	11.6	15.8	17.2
Thermal power (W)	1447.3	3942.6	6438.2

RESULTS AND DISCUSSION

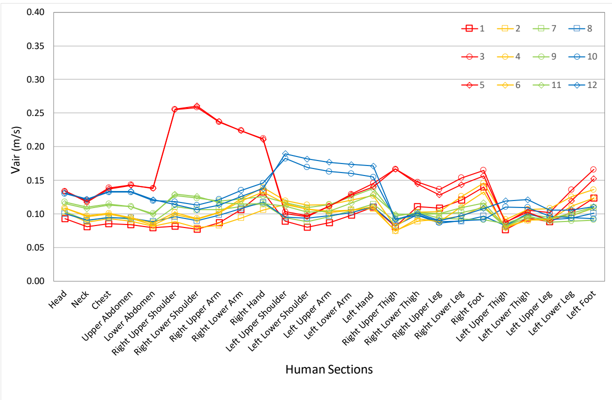
In this section, the results obtained from the distribution of environmental variables (air velocity, air temperature and DR) around the occupants, as well as those from the ADI, are presented and discussed for the three values of airflow rate used.

Environmental variables

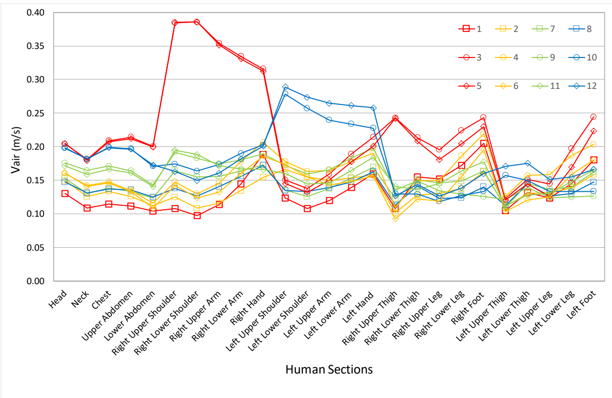
Figures 4-6 show the results obtained from the distribution of air velocity, air temperature and DR around the occupants' body sections. Points a), b) and c) are, respectively, related to Case A, Case B and Case C.



a)

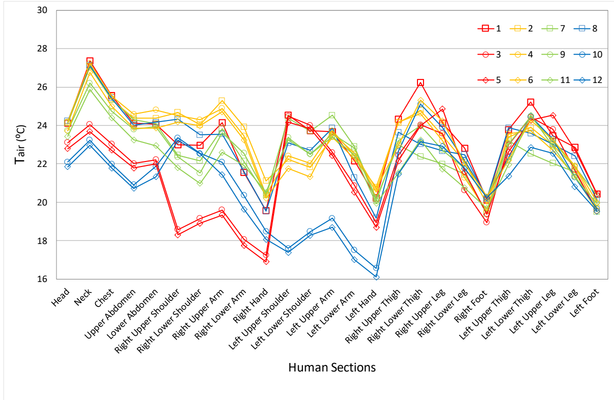


b)

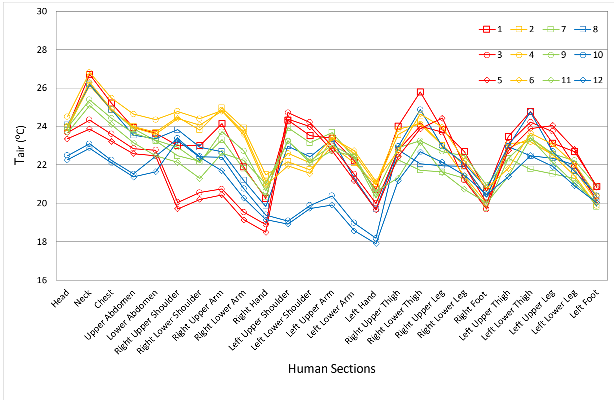


c)

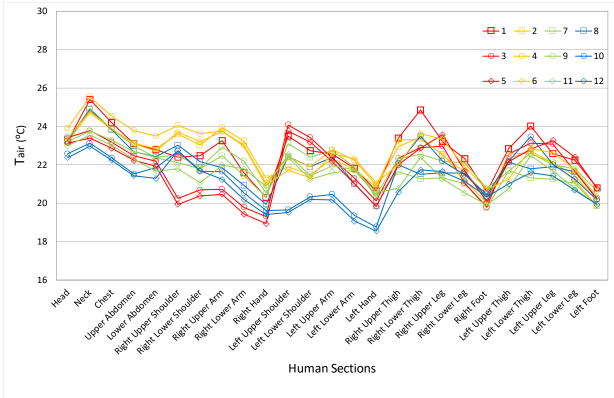
Figure 4. Air velocity (V_{air}) distribution around the occupants: a) Case A; b) Case B; c) Case C



a)



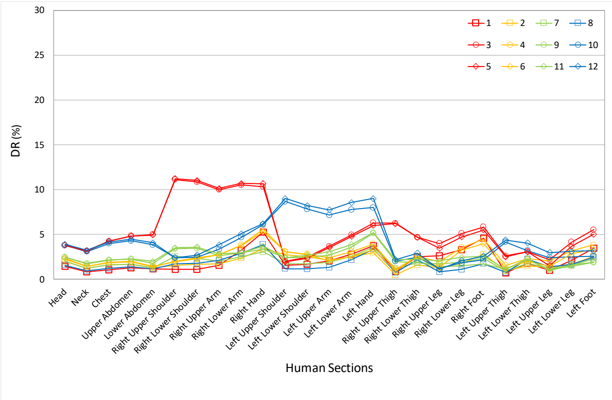
b)



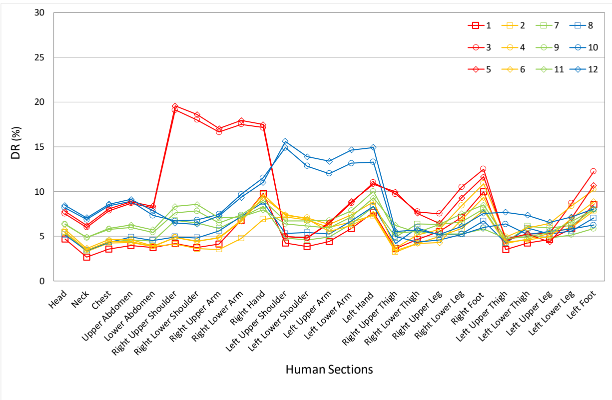
c)

Figure 5. Air temperature (T_{air}) distribution around the occupants: a) Case A; b) Case B; c) Case C

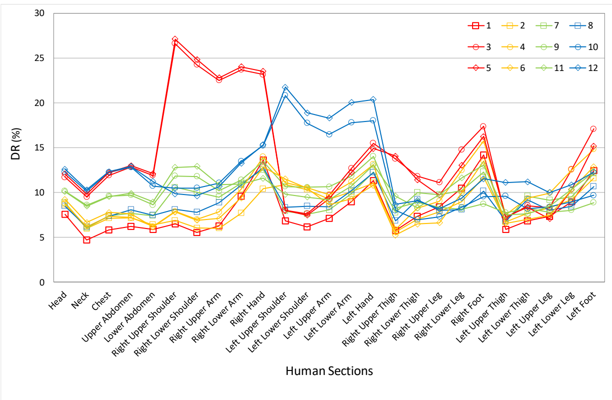
According to the results obtained, it appears that, in general, the air velocity around the lower sections of the occupants remains relatively uniform between 0.05 m/s and 0.15 m/s for the airflow rates of Cases A and B, rising slightly to the range between 0.10 m/s and 0.20 m/s for the airflow rate of Case C. The air velocity around the upper sections of the occupants located in the central area of the compartment also remains relatively uniform, within the values mentioned above. The highest differences are seen in occupants 3, 5, 10 and 12 located next to the side walls, directly under the effect of the vertical jets. In these occupants, the air velocity around their upper sections is greater than in their lower sections. The air velocity around the sections of these occupants increases when the airflow rate also increases. In occupants 3 and 5, air velocity is higher in the right arm and shoulder sections. In occupants 10 and 12, air velocity is higher in the left arm and shoulder sections.



a)



b)



c)

Figure 6. Draught Risk (DR) distribution around the occupants: a) Case A; b) Case B; c) Case C

In general, for the airflow rates considered in this study, the distribution of the air temperature around the occupants' body sections is relatively uniform, noting the greatest differences in occupants 3, 5, 8 and 10 located next to the side walls. The amplitude of the fluctuations in the air temperature around the occupants' body sections decreases slightly with the increase in the airflow rate. It is also noted that the maximum value of the air temperature around the occupants' body sections decreases slightly and that its minimum value increases slightly when the airflow rate is increased. It is highlighted that the occupants who are positioned in the center of the room will feel the same thermal sensation of the occupants occupying the sides near the ducts.

When the airflow rate increases, the values of the distribution of the DR around the occupants' body sections also increase, although all of them remain within the acceptable limit according to category C (ISO 7730: 2005). For the airflow rate of Case A, almost all values of the DR distribution around the occupants' body sections are within category A, while for the airflow rate of Case B, some body sections of the occupants 3, 5, 10 and 12 already present DR values within category B. In general, the distribution of the DR is greater among the occupants' body sections located next to the side walls than in the occupants' body sections located in the central zone of the compartment. The highest DR values are obtained in the upper sections of the occupants' body located next to the side walls, in particular the right arm, shoulder and hand of the occupants 3 and 5, and the left arm, shoulder and hand of the occupants 10 and 12.

Air Distribution Index

Figure 7 shows the values obtained for the PPD and the CO_2 concentration in the breathing area to which the occupants are subject regarding to the airflow rates of Case A, Case B and Case C.

The results show that the average PPD values are acceptable within category B (ISO 7730: 2005) for all airflow rates used. Therefore, it can be considered that an acceptable level of thermal comfort for the occupants is ensured. The CO_2 concentration decreases with the increase in the airflow rate. For the case C airflow rate, its value is close to the acceptable limit recommended by the ASHRAE 62.1:2016 standard, so the air quality can be considered close to the acceptable.

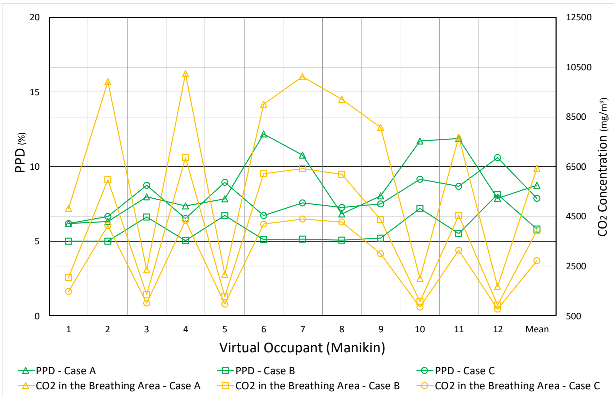


Figure 7. PPD and CO_2 concentration in the breathing area to which occupants are subject regarding to the airflow rates of Case A, Case B and Case C

Figure 8 shows the values obtained for the Effectiveness for Heat Removal and Effectiveness for Contaminant Removal to which the occupants are subject regarding to the airflow rates of Case A, Case B and Case C.

The results show that, for each of airflow rates used, the Effectiveness of Heat Removal is higher than the Effectiveness for Contaminant Removal. When the airflow rate increases, the average value of Effectiveness for Heat Removal decreases from 68.1% (Case A) to 53.8% (Case C), with a decrease of 10.9% between Case A and Case B and 3.4% between Case B and C. The Effectiveness of Heat Removal from each of the occupants it is relatively uniform. When the airflow rate increases, the average Effectiveness for Contaminant Removal increases from 22.3% (Case A) to 28.4% (Case C), this increase being 3.8% between Case A and Case B and 2.3% between Case B and C. The Effectiveness for Contaminant Removal of the occupants (3, 5 10 and 12) located next to the side walls is much greater than that of the other occupants. For example, for the airflow rate of the Case C, occupant number 12 has an Effectiveness for Contaminant Removal of 91.6%, the highest value obtained among all occupants.

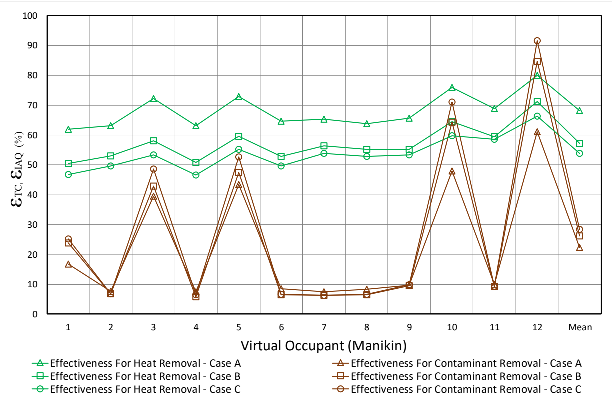


Figure 8. Effectiveness for Heat Removal (ε_{TC}) and Effectiveness for Contaminant Removal (ε_{IAQ}) to which the occupants are subject regarding to the airflow rates of Case A, Case B and Case C

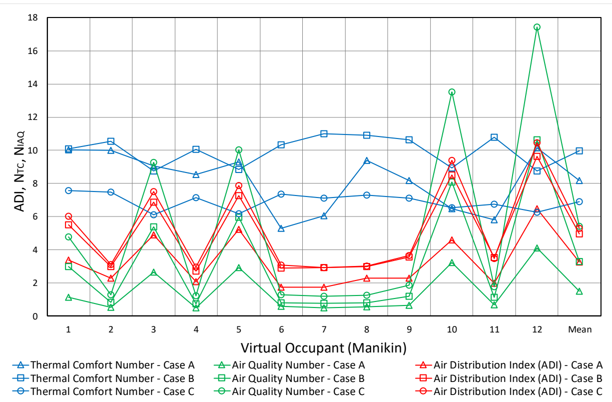


Figure 9. ADI, Number of Thermal Comfort (N_{TC}) and Number of Indoor Air Quality (N_{IAQ}) to which the occupants are subject regarding to the airflow rates of Case A, Case B and Case C

Figure 9 shows the values obtained for the ADI, Number of Thermal Comfort and Number of Indoor Air Quality to which the occupants are subject regarding to the airflow rates of Case A, Case B and Case C.

The results show that the Number of Thermal Comfort increases 22.0% from Case A to Case B and decreases 31.0% from Case B to Case C. The Number of Indoor Air Quality and ADI increase when the airflow rate increases. The increase in the average Number of Indoor Air Quality from Case A to B is 120.0% and from Case B to C is 63.6%. The increase in the average ADI value from Case A to B is 53.1% and from Case B to C is 7.6%. It can be seen that the increase in the airflow rate improves the indoor air quality but, above a certain value, it begins to penalize the thermal comfort level of the occupants. It can be seen that the increase in the airflow rate improves the quality of indoor air but, above a certain value, it begins to penalize the thermal comfort level of the occupants. Nevertheless, the performance of the HVAC system improves, that is, it continues to guarantee acceptable conditions of thermal comfort, at least within category C (ISO 7730: 2005), and to improve the level of indoor air quality to a value close to acceptable. The ADI value is much higher for occupants located next to the side walls than for occupants located in the central area of the compartment, which shows a better performance of the HVAC system in the zone next to the side walls than in the central zone.

CONCLUSIONS

In this article, an HVAC system was developed based on a system of vertical confluent jets located next to the side walls, at a height of 1.8 m from the floor, and on an exhaust system consisting of ducts aligned in the central area of the compartment, and located above the level of the head. The aim is to improve the level of thermal comfort and indoor air quality for occupants while improving the performance of the HVAC system. In this sense, the performance of the proposed system was studied, assuming winter conditions, for three values of the airflow rate (Case A, Case B and Case C).

The main conclusions drawn from this work with the increase in the airflow rate are as follows:

- The distribution of air temperature around the occupants' body sections become more uniform;
- The values of the distribution of the DR around the sections of the occupants' body increase, although remaining within the acceptable limit of category C (ISO 7730: 2005);
- The CO_2 concentration decreases for values close to acceptable and the average value of PPD increases remaining within the acceptable limit of category B (ISO 7730: 2005);
- The average Effectiveness for Heat Removal decreases and the average Effectiveness for Contaminant Removal increases;

- The average value of ADI increases.

Concerning to occupants, those who have the best compromise between the level of thermal comfort, air quality, the DR and the value of the ADI are those located close to the side walls, regardless of the value of the airflow rate.

The best performance of the HVAC system is obtained for the case C airflow rate (0.3500 m^3/s). In this case, it is possible to obtain, on average, the best compromise between the values of the PPD, the concentration of CO_2 and the DR, in general, within the acceptable values recommended by the standards (ISO 7730: 2005, ASHRAE 62.1: 2016).

ACKNOWLEDGMENTS

The authors would like to acknowledge to the project (SAICT-ALG/39586/2018) from Algarve Regional Operational Program (CRESC Algarve 2020), under the PORTUGAL 2020 Partnership Agreement, through the European Regional Development Fund (ERDF) and the National Science and Technology Foundation (FCT).

REFERENCES

American Society of Heating, Refrigerating and Air-Conditioning Engineers (2016). *Ventilation for Acceptable Indoor Air Quality* (ASHRAE 62.1).

Arghand, T., Karimipanah, T., Awbi, H., Cehlin, M., Larsson, U. and Linden, E. (2015). "An experimental investigation of the flow and comfort parameters for under-floor, confluent jets and mixing ventilation systems in an open-plan office," *Building and Environment*, 92, 48–60.

Awbi, H. (2003). *Ventilation of buildings*. Taylor & Francis, London (UK).

Cho, Y., Awbi, H. and Karimipanah, T. (2008). "Theoretical and experimental investigation of wall confluent jets ventilation and comparison with wall displacement ventilation". *Building and Environment*, 43(6), 1091–1100.

Conceição, E. (2000). "Evaluation of thermal comfort and local discomfort conditions using the numerical modelling of the human and clothing thermal system". *Proceedings of 7th International Conference on Air Distribution in Room, ROOMVENT 2000,* Reading (UK), 9-12 July, 2000.

Conceição, E., Silva, M., André, J. and Viegas, D. (2000). "Thermal behaviour simulation of the passenger compartment of vehicles". *International Journal of Vehicle Design*, 24, 372-387.

Conceição, E. and Lúcio, M. (2001). "Numerical and subjective responses of human thermal sensation". *Proceedings of the 6th Portuguese Conference on Biomedical Engineering*, Faro (Portugal), 11-12 June, 2001.

Conceição, E. and Lúcio, M. (2006). "Numerical study of thermal response of school buildings in summer conditions". *Proceedings of the 8th International Conference and Exhibition on Healthy Buildings (HB 2006)*. Lisbon (Portugal), 4-8 June 2006.

Conceição, E., Lúcio, M., and Farinho, J. (2007). "Experimental and numerical study of personalized of ventilation in classrooms desks". *Proceedings of the 10th International Conference on Air Distribution in Rooms*, Helsinki (Finland), 13-15 June, 2007.

Conceição, E., Lúcio, M., Vicente, V. and Rosão, V. (2008a). "Evaluation of local thermal discomfort in a classroom equipped with crossed ventilation". *International Journal of Ventilation*, 7(3), 267–277.

Conceição, E., Lúcio, M. and Lopes, M. (2008b). "Application of an indoor greenhouse in the energy and thermal comfort performance in a kindergarten school building in the south of Portugal in winter conditions". *WSEAS Transactions on Environment and Development*, 4, 644-654.

Conceição, E., Vicente V. and Lúcio, M. (2008c). "Airflow inside school building office compartments with moderate environments". *HVAC&R Research*, 14(2), 195-207.

Conceição, E. and Lúcio, M. (2009). "Numerical study of the thermal efficiency of a school building with complex topology for different orientations". *Indoor and Built Environment*, 18(1), 41-51.

Conceição, E., Lúcio, M., Ruano, A. and Crispim, E. (2009). "Development of a temperature control model used in HVAC systems in school spaces in Mediterranean climate". *Building and Environment*, 44, 871-877.

Conceição, E. and Lúcio, M. (2010a). "Numerical simulation of passive and active solar strategies in building with complex topology". *Building Simulation*, 3, 245-261.

Conceição, E. and Lúcio, M. (2010b). "Numerical study of the influence of opaque external trees with pyramidal shape on the thermal behaviour of a school building in summer conditions". *Indoor and Built Environment*, 19(6), 657-667.

Conceição, E., Rosa, S., Custódio, A., Andrade, R., Meira, M. and Lúcio, M. (2010a). "Study of airflow around occupants seated in desks equipped with upper and lower air terminal devices for slightly warm environments". *HVAC&R Research, 16(4)*, 401-412.

Conceição, E., Nunes, A., Gomes, J. and Lúcio, M. (2010b). “Application of a school building thermal response numerical model in the evolution of the adaptive thermal comfort level in the Mediterranean environment”. *International Journal of Ventilation*, 9(3), 287-304.

Conceição, E., Lúcio, M. and Awbi, H. (2013). “Comfort and airflow evaluation in spaces equipped with mixing ventilation and cold radiant floor”. *Building Simulation*, 6(1), 51-67.

Conceição, E. and Lúcio, M. (2016). “Numerical simulation of the application of solar radiant systems, internal airflow and occupants’ presence in the improvement of comfort in winter conditions”. *Buildings*, 6(3), 38.

Conceição, E., Gomes, J. and Ruano, A. (2018). “Application of HVAC systems with control based on PMV index in university buildings with complex topology”. *IFAC PapersOnLine.*, 51, 20–25.

Fanger, P. (1970). *Thermal comfort*. Danish Technical Press, Copenhagen (Denmark).

Fanger, P., Melikov, A., Hanzawa, H. and Ring, J. (1988). “Air turbulence and sensation of draught”. *Energy and Buildings*, 12(1), 21-39.

International Organisation for Standardisation (2005). *Ergonomics of the thermal environments – analytical determination and interpretation of thermal comfort using calculation of the PMV and PPD indices and local thermal comfort criteria* (ISO 7730).

Karimipanah, T., Sandberg, M. and Awbi, H. (2000). “A comparative study of different air distribution systems in a classroom”. *Proceedings of 7th International Conference on Air Distribution in Rooms, ROOMVENT 2000*, Reading (UK), 9-12 July 2000.