CFD model of thermal and velocity conditions in a particular indoor environment

Miguel Mora Pérez[1], Gonzalo López Patiño[1], Ignacio Guillén Guillamón[2], P. Amparo López Jiménez[1]

[1] Hydraulic and Environmental Engineering Department. Universitat Politècnica de Valencia. Spain.
[2] Applies Physics Department. Universitat Politècnica de Valencia. Spain.

Abstract
The demand for maintaining high indoor environmental quality (IEQ) with the minimum energy consumption is rapidly increasing. In the recent years, several studies have been completed to investigate the impact of indoor environment factors on human comfort, health and energy efficiency. Therefore, the design of the thermal environment in any sort of room, specially offices, has huge economic consequences. In this paper, a particular analysis on the air temperature in a multi-task room environment is modeled, in order to represent the velocities and temperatures inside the room by using Computational Fluid Dynamics (CFD) techniques. This model will help to designers to analyze the thermal comfort regions inside the studied air volume and to visualize the whole temperatures inside the room, determining the effect of the fresh external incoming air in the internal air temperature.

Keywords: Indoor environmental quality (IEQ); Computational Fluid Dynamics (CFD); Ventilation; Under floor heating.

1. Introduction
The simulation of indoor climates is being used to improve comfort and energy consumption of the buildings. The owners are recently taking pro-active steps to address actions in such buildings to improve indoor environmental conditions with minimal energy use. Despites the movement of air is completely determined by the fluid mechanics laws [1], several studies have been completed focused in the research of the impact of indoor environment factors on human comfort, health and productivity in the recent years. As a result, the general performance of office work is affected by the indoor air quality.
To provide a healthy indoor environment a good indoor air quality is required [2-4]. The IAQ (Indoor Air Quality) or the IEQ (Indoor Environmental Quality) [5] are used to determine people's comfort in buildings. Both take into account the combination of numerous factors. Indoor environments are highly complex and building occupants may be exposed to a variety of contaminants and hydrothermal conditions. The quality of an indoor environment comprises the interaction of the current ventilation and heating systems and its operation, the weather conditions which affects the building, the indoor and outdoor pollutant sources: building furnishings, chemicals, biological or physical pollutants; the building occupants and its activity and acoustic and visual parameters. Therefore it is almost impossible to describe with one indicator the indoor environment in a building.

There are many standards and guidelines which quantify the requirements of a comfortable indoor environment like CR 1752 (1998) [6], EN ISO 7730 (2007) [7], EN 15251 (2007) [8], ASHRAE 55 (2007) [9], etc. In these documents several criteria for acceptable thermal and pollution conditions are specified. Understanding the sources of indoor environmental discomfort and controlling them helps to improve occupants productivity and the buildings energy consumption. On the one hand, the way to reduce indoor air pollution is by filtering air in the ventilation system and renovating the indoor air [10]. The second way needs to be controlled to avoid energy wasting. On the other hand computational models are required to simulate the indoor environment and how the heating and ventilation systems work to try to improve the thermal comfort. In this sense, thermal comfort includes draught, huge vertical air temperature gradients, air velocity, radiant temperature asymmetry, floor temperature etc.

The comfort conditions should be considered in the occupied zone defined by the region within an occupied space between 75 and 1800 mm above the floor and more than 600 mm from the walls or fixed air-conditioning equipment [11].

One of the most used heating systems is under floor heating. This is considered one of the best systems to reach a huge thermal comfort and energy efficiency rate. The thermal comfort is influenced by floor surface temperature and the radiant exchange by thermally conditioning the interior surfaces with low temperature long wave radiation and through convection by the surface's influence on air density. The energy efficiency is good, as it operates with low temperatures in heating and powerless source of energy are required [12]. This temperature ranges determine that under floor heating can work with solar thermal systems or geothermal, both renewable energy sources. For this reasons, under heating floor is considered a good energy efficiency system and uses in the present research, focused in a particular control volume simulation as real part of a building.

The present research focuses on the thermal conditions to get global and local thermal comfort: vertical air temperature gradients, surface temperature of the floor and draught are studied in a multitask room with under floor heating and ventilation. The indoor environment was design to maintain 21ºC in winter.

2. Methodology and general objective

2.1 General objective
An indoor computational model of air velocities and temperatures has been done to develop the initial design of the heating and ventilation system in a control volume defined by the internal faces of a multitask room placed in Valencia (Spain). The aim of the present study was to demonstrate that Computational Fluid Dynamics (CFD) could be used to determine comfort in indoor environments. A case of study is presented in which the quantification of a first approximation to comfort conditions in a general multitask room with under floor heating was done. The modelling aims to determine which indoor volumes were exposed to higher levels of discomfort due to different renewable air temperature and velocity settings and the energy implications of supplying cold renewable air in a room with an under heating floor system.

2.2 CFD solver applied to indoor ventilation
The here depicted methodology is a investigation with CFD and its application research in building systems and energy behaviour. The literature is profuse in research based on numerical applications of CFD: Lo et al. [13]; Ramponi and Blocken [14]; Heiselberg et al. [15]; Kato et al. [16] and Evola and Popov [17]. These models also include experimental validations applied to analyze ventilation in buildings: Chu et al. [18]; Tablada et al. [19], Murakami et al. [20]; Jiang et al. [21] and Ji et al. [22]. Even analytical approximations to the modeling of air movement in different elements of buildings can be found in the references: Karava et al. [23]; Etheridge and Sandberg [24] and Costola et al. [25].

3. Mathematical model
Computational fluid dynamics (CFD) research uses computational and mathematical models of flowing fluids to describe and predict fluid response in problems of interest. In the building scope can be applied to model the outdoor environment, such as the flow of air around a building and the indoor environment. CFD is presented as an efficient, costless-effective tool for predicting systems response under a broad range of operating conditions. Furthermore, they can visualize hydrodynamic aspects impossible to measure or represent in a real case (i.e. velocity stream lines) that have great importance in the comprehension of the studied phenomena.

The mathematical model is composed by a geometry where mass and momentum conservation equations are solved by the code. The geometry model is designed to work on three-dimensional meshes. The volume mesh in a simulation is the mathematical description of the space (or geometry) of the problem being solved.

The computational model solves numerically the governing laws of Fluid Dynamics in a geometrical domain taking into account the turbulent phenomena. Velocity, pressure and temperature fields are calculated in a discrete manner at the nodes of a certain mesh or grid and they are represented along the mesh. The continuity or mass conservation equation solved by the software used is expression (1).

$$\frac{\partial \rho}{\partial t} + \nabla \rho \vec{v} = S_m \tag{1}$$

where ρ is the fluid density, $\vec{v}$ is velocity and S_m represents the mass source contained in the control volume. Also, the momentum equation is considered by equation (2).

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

where p is the static pressure, τ the stress tensor defined in expression (3) and the gravitational (g) and outer forces (F) defined on the control volume, respectively. In (3) μ is the eddy viscosity and I is the unit tensor. The third term accounts for the effect of the expansion of volume.

$$\overline{\tau} = \mu \left[\left(\nabla \vec{v} + \nabla \vec{v}^T \right) - \frac{2}{3} \nabla \vec{v} I \right] \tag{3}$$

All conditions and properties are defined via STAR-CCM+ and solved using the coupled solver. The results are displayed via available post-processing tools.

3.1 Geometry
The case study is a multitask office located in Valencia (Spain). The office has a complex shape. Figure 1 shows the indoor environment modeled to obtain velocities profiles and the temperature distributions. The total volume is mainly occupied by a table and sits (not modeled), a kitchen with an air extractor in the middle of the room and several furniture placed in one of the lateral wall (Figure 1a). The geometry consists of a linear supply diffuser 120 mm wide and 140mm long installed on the inner wall of the room (Figure 1a); a linear exhaust 110 mm. wide and 1.8m long, installed in the ceiling (Figure 1b); a principal door 2.46m. high and 3.8 m. wide (Figure 1c); a lateral window 2m. high and 0.5m. wide (Figure 1d); a rear window 0.5m. high and 3.98m. wide (Figure 1e) and a secondary door 2.6 m. high and 1.53 m. wide (Figure 1f). The room is 2.8m. high, 6.430m. wide and 4.3m. deep. The geometry includes an extractor (which function is not modeled), a small kitchen in the middle and the kitchen furniture closed to the indoor wall where the inlet is placed.

3.2 Boundary conditions and physics
The CFD analysis performed includes steady state. Segregated flow for model is used. The gravity model is used as it permits the inclusion of the buoyancy source terms in the momentum equations when using the segregated flow model. K-Epsilon turbulence model is used for representing turbulence.
The entire domain is defined as a single fluid region (air). A region is a volume domain in space defined by boundaries. A boundary is each surface that surrounds and defines a region in the model. Each boundary has its own properties, defined in Table 1.
According to [6] and [10], renewable air velocity module was set 1.54 m/s, which is the velocity to achieve 2.5 air removals per hour, at the supply diffuser. Temperatures of the renewable air were set according to air temperature in winter in Valencia (Spain) 8ºC, 10ºC, 12ºC and 14ºC. The under floor heating system was set 27ºC in all the floor surface, the internal face of the outdoor walls were set 20.5ºC and the internal face of the windows were set 19ºC. The heat balance in the room reach 21ºC indoor average temperature [6] in winter conditions.
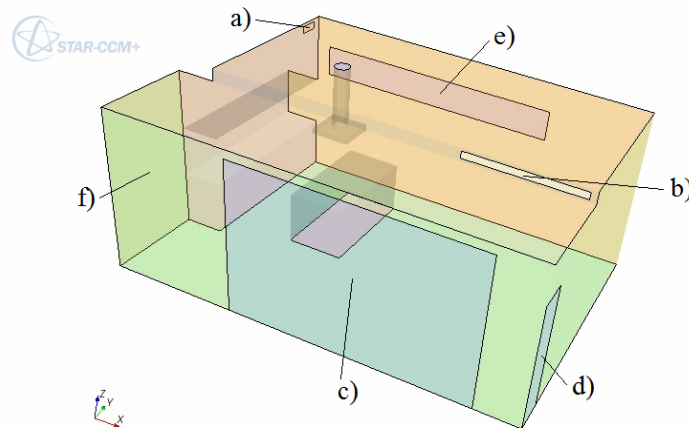
Figure 1. Modeled room

Table 1. Boundary conditions specifications

Type	Surface (Wind tunnel)	Properties
Velocity Inlet	Linear supply diffuser (Figure 1a)	Velocity module and direction
Pressure Outlet	Linear exhaust (Figure 1b)	By default
Wall	Windows (Figure 1c, d, e)	Temperature condition
Wall	Outdoor walls (Figure 1c, d, e)	Temperature condition
Wall	Indoor wall (Figure 1f)	Adiabatic
Wall	Under floor heating system	Temperature condition

3.3 CFD mesh and convergence
The numerical method is solved by the finite volume technique. The solution to a flow problem is solved by calculating the flow-equations on the nodes within the cells. The accuracy of the result depends on the definition of the nodes. The definition of a good mesh is crucial to find the optimum between the smallest number of nodes and the accuracy of the results. Finally, the mesh for the volume control used has the following characteristics: 114,401 items; 675,846 internal faces and 544,867 vertices. When the mesh has been completed, a grid-independence study, including the number of nodes and the size of the enlarged domain was performed in order to ensure the validity of the numerical computational procedure. Wall treatment was necessary for modeling up proper boundary conditions for turbulence. In this case the wall treatment used is the high-y+. The high-y+ wall treatment implies the wall-function-type approach in which it is assumed that the near-wall cell lies within the logarithmic region of the boundary layer. It is suitable for use with models that do not explicitly damp the turbulence in the near-wall region. While a good rule of thumb is that the wall-cell centroid should be located in the logarithmic region of the boundary layer (y+>30) [26], as in the present cases. Correct values of y+ allows a proper assessment of the mesh. In this case, this requirement was accomplished for all walls.

4. Analysis of results
4.1 Room temperature model
A first set of four simulations has been carried out. The simulations have been used to analyze the indoor air flow performance and the thermal effect in the medium average temperature in the room. Only the temperature of the inlet air flow was modified. A sample of temperatures was chosen to represent inlet air temperature in Valencia in winter. The temperatures were $T_{in}=8$ºC, 10ºC, 12ºC and 14ºC.
Figure 2 shows a vertical virtual plane to visualize the room's temperature in the occupied zone. It is shown that the temperature of the room is 21ºC approximately in any case. It meant that the cool flow did not modify the average temperature of the room and had no appreciable energy effect in general terms. Nevertheless, the volume placed in front of the inlet diffuser had lower air temperatures than the rest of the office. While the air temperature of the air supplied was increased, the volume with low temperatures was reduced.
Although no big temperature changes were observed, temperature streamlines from the inlet diffuser and velocity vectors should be considered to notice if the occupants could feel uncomfortable i.e. draught.

(a) Tin = 8ºC

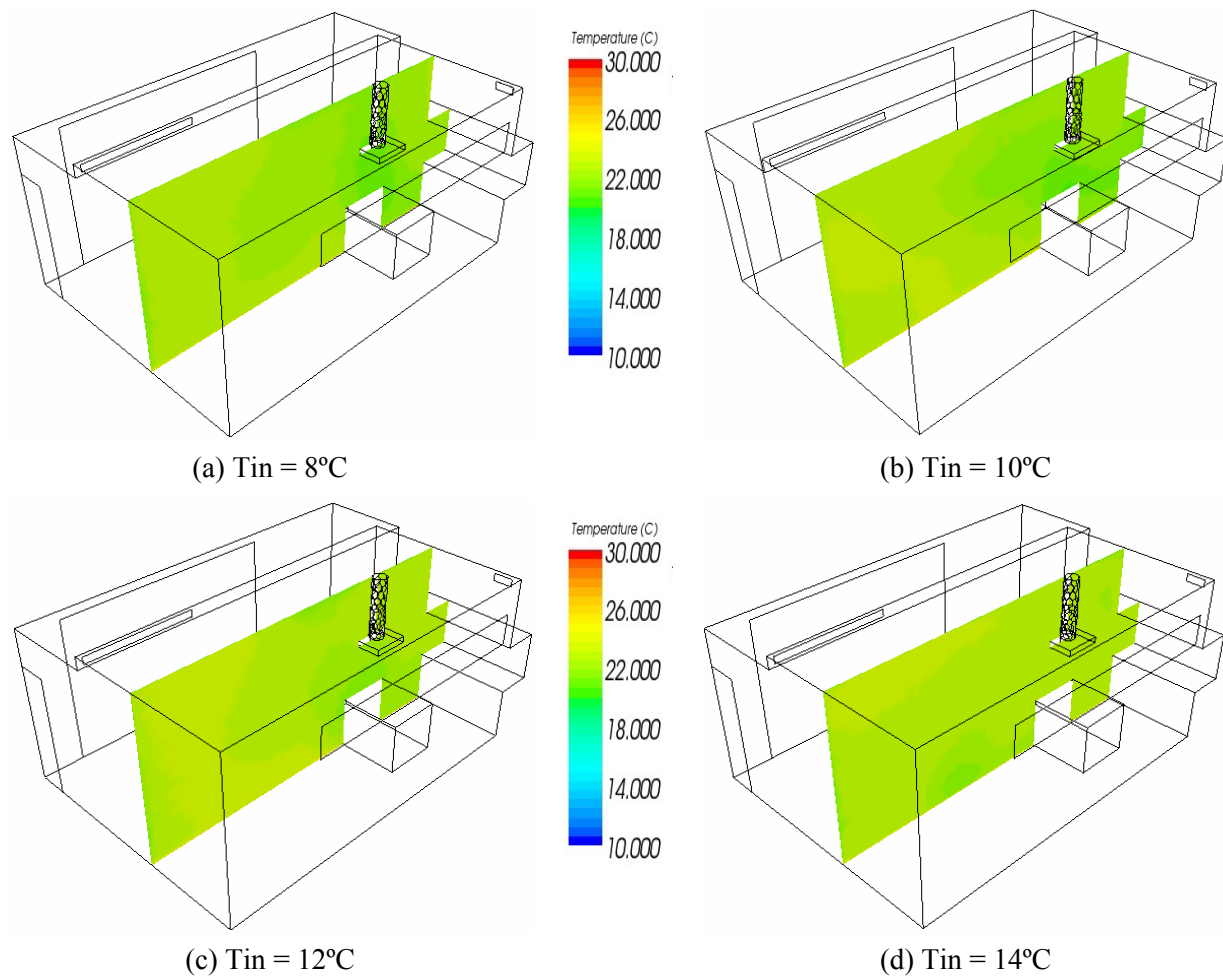(b) Tin = 10ºC

(c) Tin = 12ºC

(d) Tin = 14ºC

Figure 2. Air temperature in the office room

Figure 3 shows how the temperature streamlines of the air supplied were spread into the office. It can be verified visually that there is a volume placed in front of the supply diffuser where the temperature gradient is high. This temperature gradient is reduced while the temperature of the new air introduced in the room is increased. Nevertheless, the occupants will feel uncomfortable in this part of the room due to velocity feeling. In addition, velocity streamlines also follows this streamlines and draught could be felt by occupants.

Figure 3 shows that the inlet air was spread in similar way in all cases, so the inlet air temperature does not influence the total air mixture in the room. The temperature distribution is uniform and there is only a volume with high temperature gradients placed near the ceiling in front of the air supply diffuser. As high inlet air temperature later the velocity streamlines falls down to the occupied zone. Furthermore, the position of the air kitchen extractor is not the most recommended one because in the cases Tin= 8, 10 and 12 ºC it could be noticed that most of the new air flow comes under the extractor and can be removed from the room before mixing with the rest of the air.

From the energy point of view, there is no effect due to the huge power of the under floor heating in the whole room: the temperature considered constant for the model, and maintained by the under floor heating system has very little affection with the considered number of air removals. The discomfort sensation will have only local considerations, very near to the entrance of fresh air.

4.2 Analysis of critical zones in the room in which comfort concerns
The analysis of the air velocity profiles allows detecting volumes with draught. According to [6], draught occurs when air velocity exceeds 0.2 m/s. Figure 4 shows the velocity vectors above 0.2 m/s in the office room in cases (a) and (d). The cases (b) and (c) are not represented because the velocity profiles are between the air velocity profile in (a) and (d). It can be observed that low air temperatures increase the air velocity in the central volume of the room. The figure shows the inlet air velocity vectors above 0.2 m/s in which draught is felt.
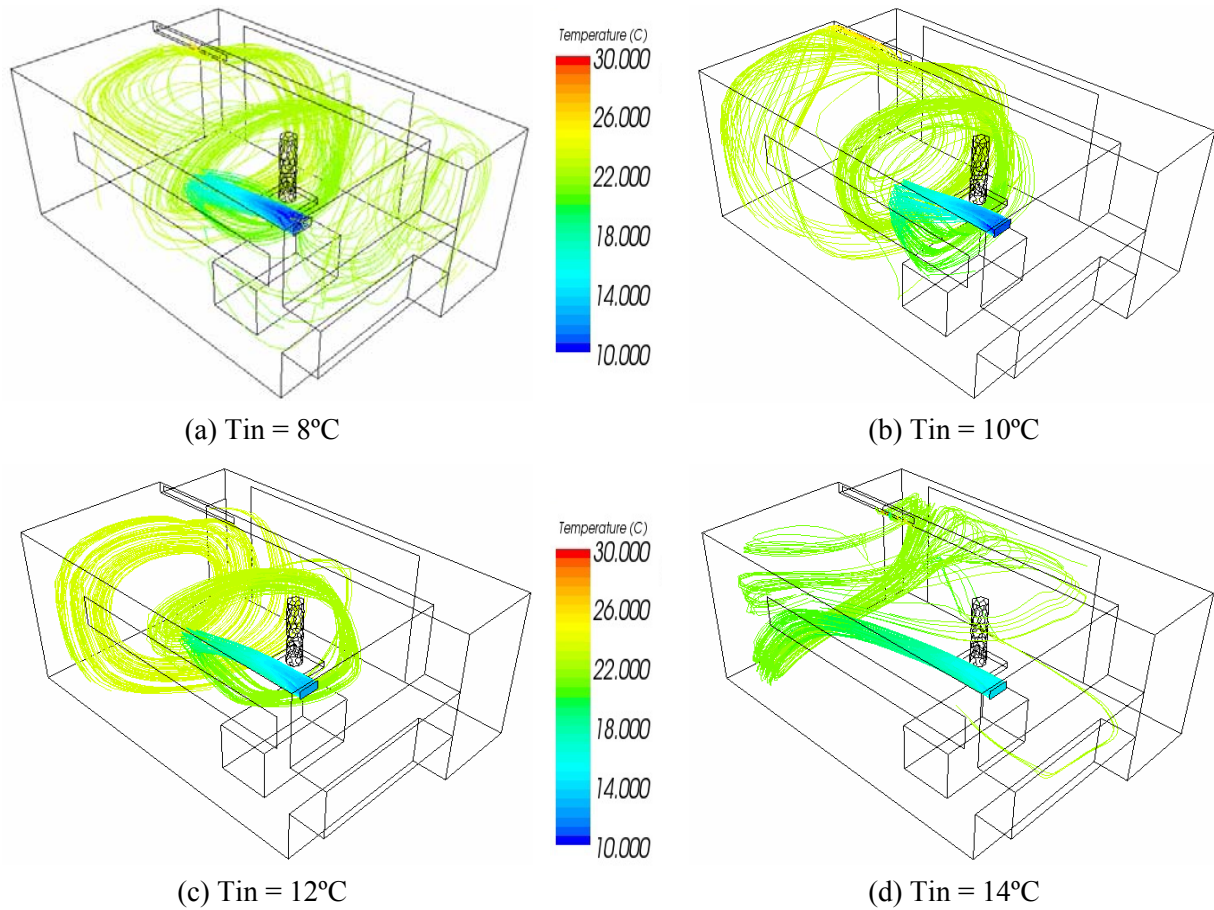
(a) Tin = 8ºC   (b) Tin = 10ºC

(c) Tin = 12ºC   (d) Tin = 14ºC

Figure 3. Flow streamlines representing temperature

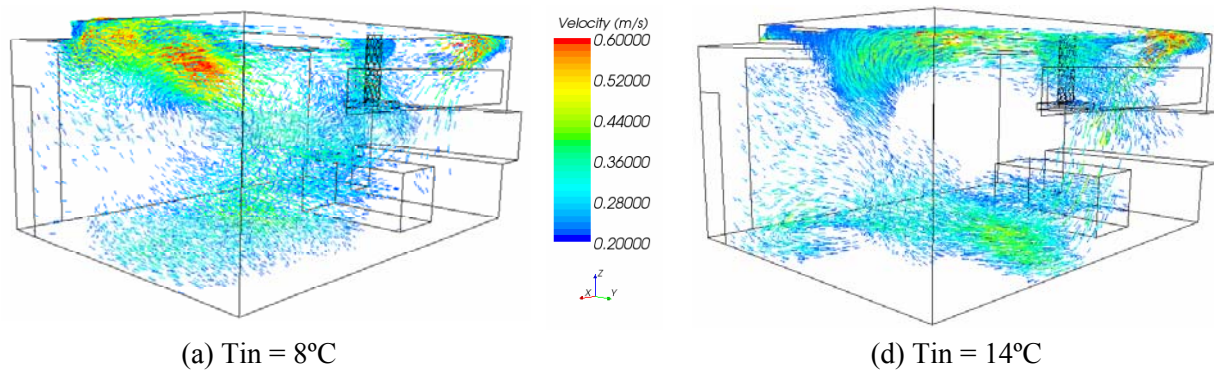(a) Tin = 8ºC   (d) Tin = 14ºC

Figure 4. Air velocity vectors above 0.2 m/s (draught)

Figure 4 shows that while the temperature of the air supplied is increased, the central draught volume is reduced.

Nevertheless, to completely avoid draught in this volume, the increase the inlet air temperature is not enough. The external temperature of air should then be pre-heated in order to totally equalize the temperatures in the room, which implies other additional energy consumption sources and strategies in the building.

5. Conclusion

The present study focuses on the energy implications and thermal comfort effect of a cold air introduced in a multitask room to renovate the indoor air as International Standards require to assure a good IEQ factor. A CFD model of the room temperature and velocity is proposed by considering suitable boundary conditions. The room was heated by under floor heating and which indoor air is renovated by a ventilation system placed in the roof. This ventilation system supplied cold air (considering external air

temperatures in Valencia, Spain, with typical Mediterranean climate). The external air was mixed with the indoor air with almost no thermal changes in the total air considered, due to the power of the under floor heating and the relative small volume of renewal external air. Nevertheless. the conditions of air renovation were sufficient for a normal occupation of the room.

With this thermal study, a local indoor volume was noticed with high temperature and velocity contrasts. This volume is placed in the central area of the room. Different temperature of the inlet air were tested, concluding that even with high temperatures of the new air, the discomfort source was the air velocity and not the temperature gradients in the room. Total energy consumption was increased almost with no global effect in the room. The inlet velocity should be carefully design to avoid velocities above 0.2 m/s in the occupied zone. Further studies should be done analyzing the air velocity through the inlet diffuser to improve comfort in which air velocity concerns, for example, modifying the air supply diffuser geometry or position. Furthermore, additional comfort parameters including most of the parameters which influence comfort as the predicted mean vote will be considered. In any way, the CFD is a capable tool for defining these local aspects that cannot be considered by any other techniques, and will help very much designers to define the comfortable zones in any sort of building.

Acknowledgements

References

[1] P.F. Linden. The fluid mechanics of natural ventilation. Annual Review of Fluid Mechanics, 1999, Vol 31, pp. 201–238.
[2] G. Carrilho da Graça, Q. Chen, L.R. Glicksman, L.K. Norford. Simulation of wind-driven ventilative cooling systems for an apartment building in Beijing and Shanghai. 2002 Energy and Buildings. Vol.34, pp. 1–11.
[3] Q. Chen. Ventilation performance prediction for buildings: a method overview and recent applications.2009 Building and Environment. Vol.44, pp. 848–858.
[4] P. Heiselberg, M. Perino. Short-term airing by natural ventilation – implication on IAQ and thermal comfort. 2010 Indoor Air. Vol.20, pp. 126–140.
[5] Dowd Richard. Indoor air quality. Environmental Science & Technology, 1984, Vol.18(6), pp.187.
[6] CEN. 1998. Standard CR 1752. Ventilation for Buildings: Design Criteria for the Indoor Environment. Brussels.
[7] CEN. 2005. Standard ISO EN 7730. Analytical determination and interpretation of thermal comfort using calculation of the PMV and PPD indices and local thermal comfort. Brussels.
[8] CEN. 2007. Standard EN 15251. Indoor environmental input parameters for design and assessment of energy performance of buildings- addressing indoor air quality, thermal environment, lighting and acoustics. Brussels.
[9] ASHRAE. 2007. ASHRAE Standard 55-2007. Thermal environment conditions for human occupancy. Atlanta.
[10] ASHRAE 2001. ASHRAE Standard 62-2001. Ventilation for acceptable indoor air quality. Atlanta.
[11] Ventilation for acceptable indoor air quality. American Society of Heating, Refrigerating and Air-Conditioning Engineers. Semiannual meeting in Dallas 1976. Atlanta: ASHRAE 1989.
[12] Babiak J., Olesen, B.W., Petráš, D., Low temperature heating and high temperature cooling – Embedded water based surface systems, 2007 REHVA Guidebook no. 7, Forssan Kirjapaino Oy-Forssan, Finland.
[13] Lo L-J, Banks, D., Novoselac, A. Combined wind tunnel and CFD analysis for indoor airflow prediction of wind-driven cross ventilation. 2013 Building and Environment. Vol.60, pp12–23.
[14] Ramponi, R. Blocken, B. CFD simulation of cross-ventilation for a generic isolated building: Impact of computational parameters. 2012 Building and Environment. Vol.53, Pp 34–48.
[15] P. Heiselberg, Y. Li, A. Andersen, M. Bjerre, Z. Chen. Experimental and CFD evidence of multiple solutions in a naturally ventilated building. 2004 Indoor Air. Vol.14, pp. 43–54.
[16] S. Kato, S. Murakami, T. Takahashi, T. Gyobu. Chained analysis of wind tunnel test and CFD on cross ventilation of large-scale market building. 1997 Journal of Wind Energy Aerodynamics. Vol.67–68, pp. 573–587.

[17]   G. Evola, V. Popov. Computational analysis of wind driven natural ventilation in buildings. 2006 Energy and Buildings. Vol.38, pp. 491–501.

[18]   C.R. Chu, Y.-H. Chiu, Y.-W. Wang. An experimental study of wind-driven cross ventilation in partitioned buildings. 2010 Energy and Buildings. Vol.42, pp. 667–673.

[19]   A. Tablada, F. De Troyer, B. Blocken, J. Carmeliet, H. Verschure. On natural ventilation and thermal comfort in compact urban environments-the Old Havana case. 2009 Building and Environment. Vol.44, pp. 1943–1958.

[20]   S. Murakami, S. Kato, S. Akabayashi, K. Mizutani, Y. Kim. Wind tunnel test on velocity–pressure field of cross-ventilation with open windows. 1991 ASHRAE Trans, Vol.97, pp. 525–538.

[21]   Y. Jiang, D. Alexander, H. Jenkins, R. Arthur, Q. Chen. Natural ventilation in buildings: measurement in a wind tunnel and numerical simulation with large-eddy simulation. 2003 Journal of Wind Energy Aerodynamics. Vol.91, pp. 331–353.

[22]   L. Ji, H. Tan, S. Kato, Z. Bu. Wind tunnel investigation on influence of fluctuating wind direction on cross natural ventilation. 2011 Building and Environment. Vol.46, pp. 2490–2499.

[23]   P. Karava, T. Stathopoulos, A.K. Athienitis. Wind driven flow through openings – a review of discharge coefficients. 2004 Ventilation Journal Vol.3, pp. 255–266.

[24]   D.W. Etheridge, M. Sandberg. A simple parametric study of ventilation. 1984 Building and Environment. Vol.19, pp. 163–173.

[25]   D. Costola, B. Blocken, M. Ohba, J. Hensen. Uncertainty in airflow rate calculations due to the use of surface-averaged pressure coefficients. 2010 Energy and Buildings. Vol.42, pp. 881–888.

[26]   CD-Adapco Star CCM+ User's Manual.

Miguel Mora Pérez. M.Sc. in Industrial Engineering, Master in Energy Technology for Sustainable Development and Research Engineer in the Department of Hydraulic Engineering (Universitat Politècnica de Valencia). He is currently researcher at Universitat Politècnica de València in subjects related to sustainability in buildings.

Gonzalo López Patiño is Industrial Engineer, Assistant Professor in the Hydraulic and Environmental Engineering Department at the Universidad Politécnica de Valencia. He is currently Director of the Master Programme in Construction and Industrial Facilities at Universitat Politècnica de València. He has more than a decade of experience in modelling building and urban facilities, especially in fluids applications (air, water and other industrial flows) and actually he has focused his researches in sustainable applications on building constructions.

Ignacio Guillen Guillamon is Ph.D. Architect on Architectural Acoustics from the University Polytechnic of Valencia on 1999. Assistant Professor of Applied Physics on the Architectural School since 1998 and currently Researcher at the Physical Technologies Center. Director of several Master Thesis on LCA for buildings, Energy Building Simulation and Energy and Sustainable Housing Renovation. He is currently IP of an R+D+i project founded from CDTi together with a pool of Enterprises. The E3 Project "EdificaciónEcoEficiente" search through the development and construction of high efficient buildings prototypes new ways to improve the efficiency and industrialization on housing, both for news o renovation constructions. And also has been working as technical consultant on projects on topics like: Acoustic Desing of Concert and Opera Halls and theaters, Acoustic Characterization of listening rooms, Sound insulation, Passive solar architecture and natural ventilation, Energy Efficiency and Building Simulation.

P. Amparo López Jiménez is M.Sc. and PhD in Industrial Engineering, Associate Professor in the Hydraulic and Environmental Engineering Department at the Universitat Politècnica de València. She is currently the Associate Director of the Hydraulic and Environmental Engineering Department of Universitat Politècnica de València. She has more than a decade of experience in research and teaching in Engineering fields, always related to hydraulic topics. She is author and editor of several publications about Hydraulic an Environmental Engineering and Flow Dynamics. She has participated in national and international R&D projects and co-organized International Seminars and Networks. She is an experienced University Teacher, an active researcher and a former practicing engineer.