Computational simulation of natural ventilation in a laboratory model of a service building

João Pedro Monteiro Pereira

joao.pedro.monteiro.pereira@tecnico.ulisboa.pt

Mechanical Engineering Department, Instituto Superior Técnico, Technical University of Lisbon, Avenida Rovisco Pais, 1049-001 Lisboa, Portugal

November 2017

Abstract

The goal of this study is to numerically investigate the flow and temperature fields in a laboratory model of a compartment in an office building. The motivation lies in a project concerned with the study of an innovative solution that optimizes the use of suspended ceiling (generally used for decorative purposes, mainly useless), passive and mechanical ventilation in the cooling process of an office, with the aim of reducing the overall energy consumption. Nigh cooling represents a way to accomplish that goal because we can take advantage of the relative cold outdoor temperature and cool the building thermal mass, reducing the air average temperature on the next working day. The problem is explored with a computational simulation using a CFD software, reproducing the scaled model. In a first stage the study examines the temperature distribution and the flow field of the model with a localized floor heat source. Subsequently, the flow field with forced ventilation without heat source. And finally, the influence of the gaps between the suspended ceiling and walls are analysed.

The results indicate that the increased amount of exposed slab area contributes to lower the day peak indoor temperatures. Night cooling decreases the slab core temperature by 4.5 °C in about 6 hours. And the configuration for the gap of 90×125mm yields the best thermal comfort.

Keywords: Thermal Comfort, CFD, Buoyancy driven Ventilation, Enclosure, Suspended Ceiling, Night Cooling, Performance analysis.

1. Introduction

During the last decades, buildings represent the major consumers of energy, accounting for about 40% of the final energy consumed in Europe [1]. The main source of the problem is due to the constant rise, over the years, of the thermal loads in commercial buildings, mainly due to the arrival of office computers and lighting requirements; Heating, ventilating and air conditioning (HVAC) systems are therefore designed to neutralize these loads and to create a good comfortable indoor environment [2] [3]. However, these mechanical systems consume large quantities of energy having high maintenance costs, for environmental reasons the constant rising cost of energy, a suitable energy efficiency measure can contribute to a drastic decrease of the overall building energy consumption [4]. So, taking advantage of the relatively low ambient air during the summer nights (when ambient temperature is lower than indoor temperature), there is a potential of using nocturnal ventilation (fan) in order to cool down the structural elements and thereby increase the energy efficiency of mechanical systems of a building.

This work presents an innovative solution that uses the suspended ceiling and the plenum (space between the slab and the suspended ceiling) for cooling purposes. Different configurations of the length and width of the suspended ceiling will be studied to optimize the phenomenon of night cooling and the best way to analyze these several design scenarios is to study computationally.

This numerical study is an additional work that aims to simulate the natural and mechanical ventilation of an experimental project in which an office in reduced scale (1:7) is reproduced.

2. Experimental Setup

The experimental project that we aim to simulate consists in model with dimensions of: $0.75 \times 1.25 \times 0.45\ m$ and is assumed to be "occupied" from 8:00 to 20:00. A heat source (electric resistance) is used to simulate the heat release due to the occupants and equipment. This heating period is followed by 12 h (20:00 to 8:00) of night cooling, without thermal load, using a fan in one of the openings (the closest to the load) operating with a velocity magnitude of $6/8/10\ m/s$.

The reduced scale model allows to change the size of the peripheral gap between the suspended ceiling (made of cardboard) and the surrounding walls, and the tests done to date have the following gaps:

Table 1 - Dimensions of the gaps between the suspended ceiling and the surrounding walls for the different design scenarios

	Scenario 1	**Scenario 2**	**Scenario 3**	**Scenario 4**	**Scenario 5**	**Scenario 6**
Length (mm)	32	64	90	240	0	*without Suspended Ceiling*
Width (mm)	32	64	125	120	0	

The numerical study will be modeled according to the experimental arrangement using 50 Watts of power, an insufflation velocity of 8m/s during night cooling, and the ambient temperature simulated as a sinusoidal. The scenario 2 (64*64) was chosen as a starting point. All the scenarios are studied (for natural ventilation) except for the Scenario 5 (no gap).

3. Problem formulation

The calculations were performed using ANSYS Fluent 14.5. The CFD code solves the appropriate governing equations (RANS equations and turbulence models as well as the radiative transfer model) at each mesh point by advancing the previous solution through time and/or space to obtain a possible description of the whole flow field. The turbulence model chosen was the RNG k-ε model and for the radiation, the Surface to Surface (S2S) model. The main advantage of this latter model is the simplicity and convenience of not having a participant media, by ignoring any absorption, emission, or scattering of radiation, therefore, only "surface-to-surface" radiation need to be considered.

The Boussinesq approximation was used in all simulations because, in this type of problem, the air temperature differences are usually small, and the density variation can be ignored.

4. Numerical Modelling

First, a preprocessing application (*Ansys DesignModeler*) is used to establish the geometry of the model. In Figure 1 a schematic of CFD geometry is presented, the model is composed by two solid domains (Solid-Slab and Suspended Ceiling) and one fluid domain (Air). The dimensions are $0.75 \times 1.25 \times 0.45\ m$ (not including the rectangular boxes of the inlet and outlet) and it was constructed according to [5][6]. The cylindrical heat source is represented as a heat flux in the surface of the fluid, it's misleading to recognize as a body.

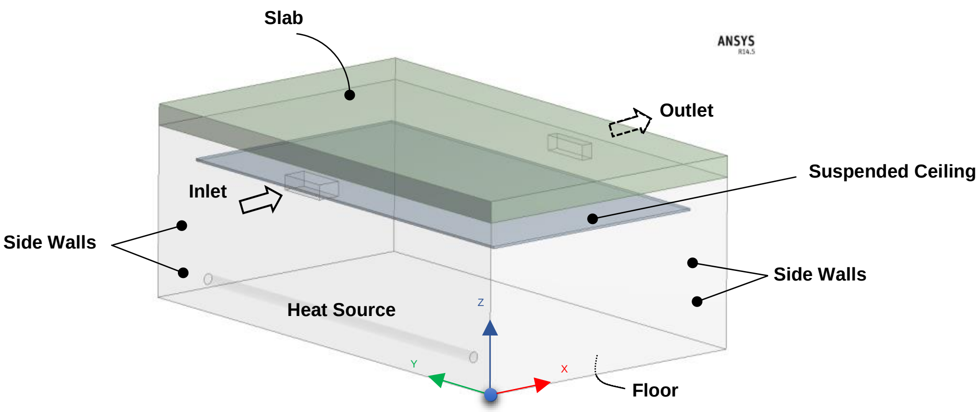

Figure 1 - Geometry of CFD model

Second, the *Ansys Mesh* generates the three-dimensional mesh. The mesh was refined in regions of high temperature gradients and its mainly assembled by unstructured tetrahedral elements (Fluid) and by a small fraction of hexagonal elements (Slab and Suspended Ceiling). For the mesh independence study, it was concluded that a mesh with 1 100 000 cells gave a robust and grid-independent solution.

Third, the solver settings are defined. All the governing equations were solved using a Second Order Upwind discretization scheme, a first order implicit time formulation, and a coupled solver. The stepping method used in all the simulations for the heating period, is: $3 \times 100s + 5 \times 300s + 69 \times 600s$. The convergence criterion demanded that the sum of the residuals of the discretized equations dropped below 10^{-3} for all the governing equations except for energy, 10^{-6} .

1.1. Boundary Conditions

The boundary conditions of the virtual model under investigation were defined as close as possible to the experimental set-up.

At the inlet slot, the boundary condition is defined as a velocity inlet boundary type during night cooling (20:00h-8:00h) and a pressure outlet (with gauge pressure taken equal to zero) during the heating period (8:00h-20:00h). At the outlet slot, the boundary was defined as a static pressure outlet during both the heating and cooling periods. The supply ambient temperature varies according to the hour of the day ranging from $23\ °C$ to $25.4\ °C$. It is important to define this temperature not only for the air supply during night (forced) ventilation but also during the day, since the supply and extraction slots remain open. The heat load will be emitting $629\ W/m^2$.

The exterior walls (side walls, floor and slab), the boundary condition chosen was heat loss by convection, with a theoretical value of $3\ W/m^2K$ which was estimated analytically using correlations for a horizontal and vertical hot plate.

5. Results and Discussion

In an attempt to obtain the computational results as closest as possible to the experimental values (normal losses due to infiltrations and other irregularities), computational heat losses through the boundaries were adjusted according to the experimental data. It was concluded that the use of theoretical convection coefficients was not an adequate solution to match the experimental results because it seems that there is a constant loss independent of the working conditions. For that reason, a constant heat flux loss of $-7.9\ W/m^2$ (found by trial and error) through the floor and the side walls was used during the heating period. As observed from the Figure 2 the experimental and the numerical simulation results show evidence of a good agreement for this latter value.

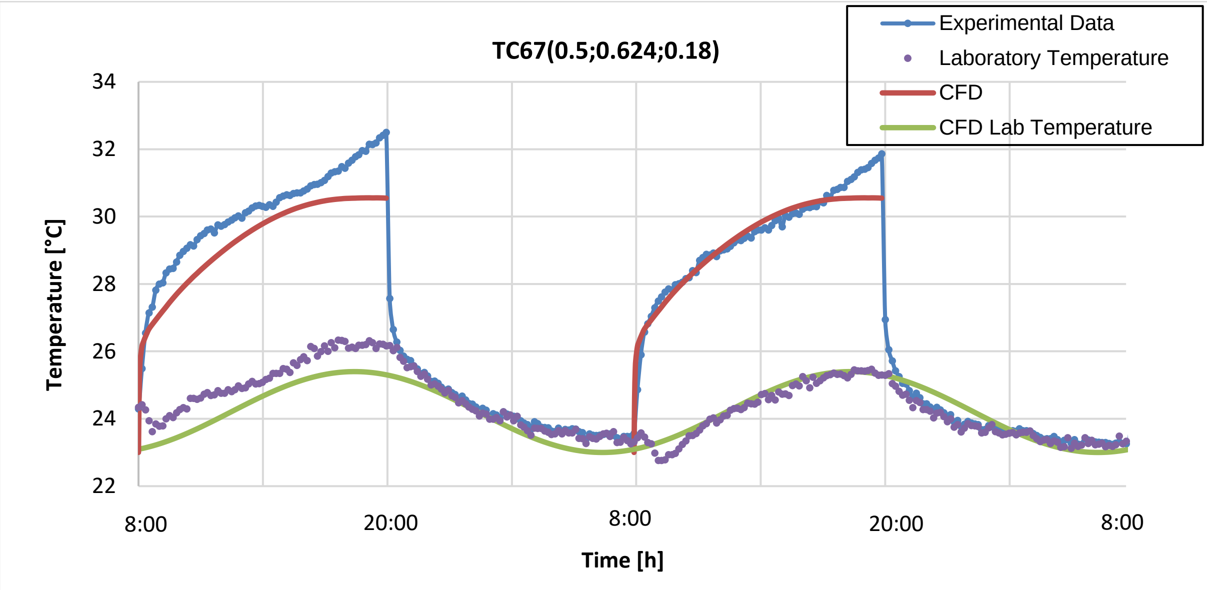

Figure 2 - Difference between measured and calculated temperature in the heating period at thermocouple 35 (plenum) for two days.

In Figure 3 and Figure 4 results of temperature and velocity (modelled with radiation), through the heating period can be seen. The results, show the general behavior of hot air. It rises very hot from the heat source creating an uprising thermal plume that attaches to the ceiling and spreads to the opposite side of the model from where it finally falls down attached to the wall into the bottom part of the indoor space, setting a peripheral rotational motion and some vortexes that enhances heat transfer through the enclosure. Nevertheless, as the indoor air is mixed the values of temperature show rising progression both at the upper and the bottom part of the space, and the stratification is shown clearly in the shaded contour plots being the upper buoyant layer above the suspended ceiling the warmest (desired for thermal comfort). In this region the slab is acting as a heat accumulator, with its overall temperature increasing with time and, as it can be seen through the temperature gradient, the hottest region is on the lower surface of the slab above the heat load. Also in this region, some of the warm air of the plume exits through the left opening (closest) and cold air enters through the opposite opening with the same flow rate.

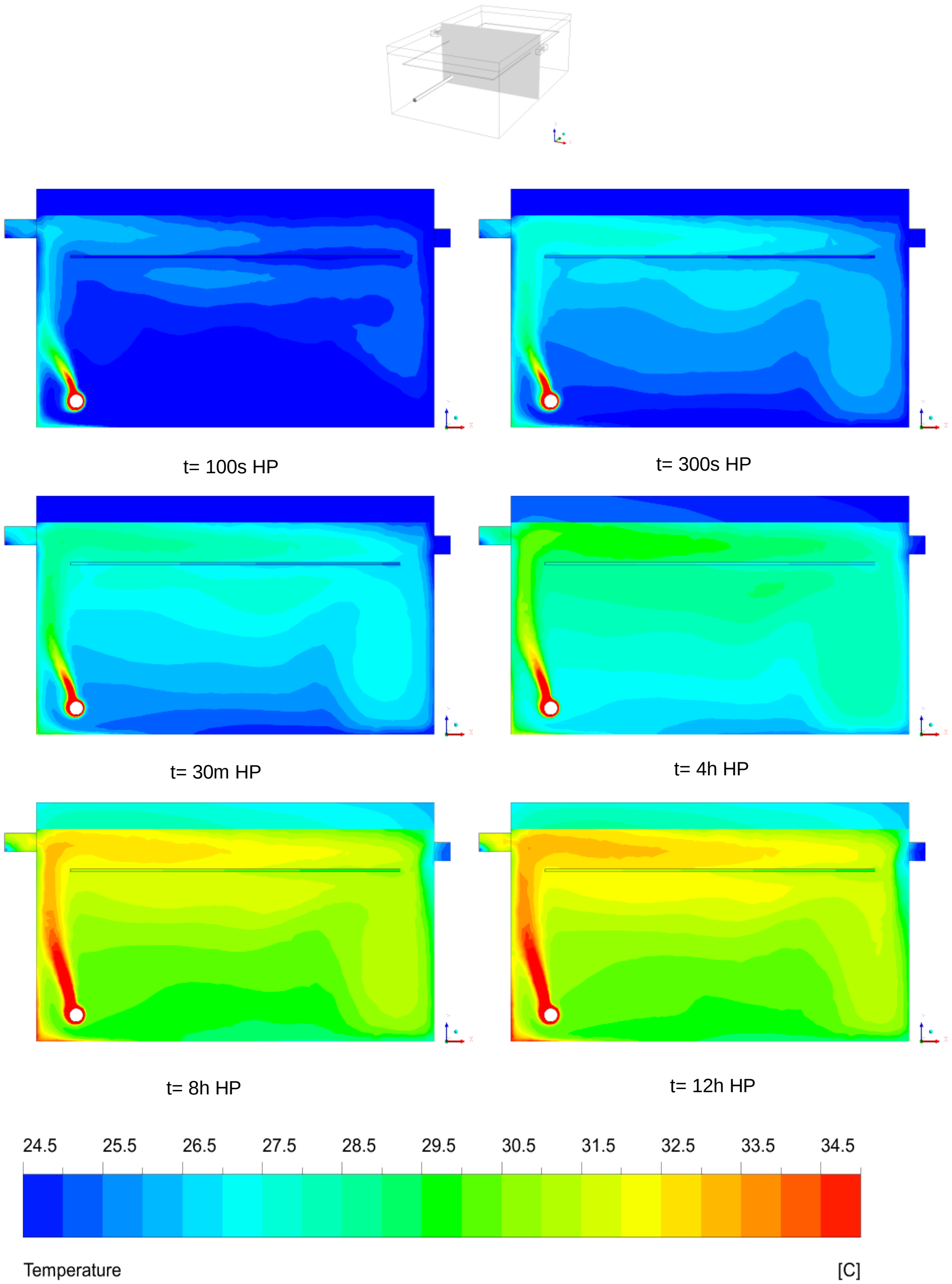

Figure 3 - Temperature contour profiles with radiation for six heating periods. Plane Y=0.625m. HP - Heating Period

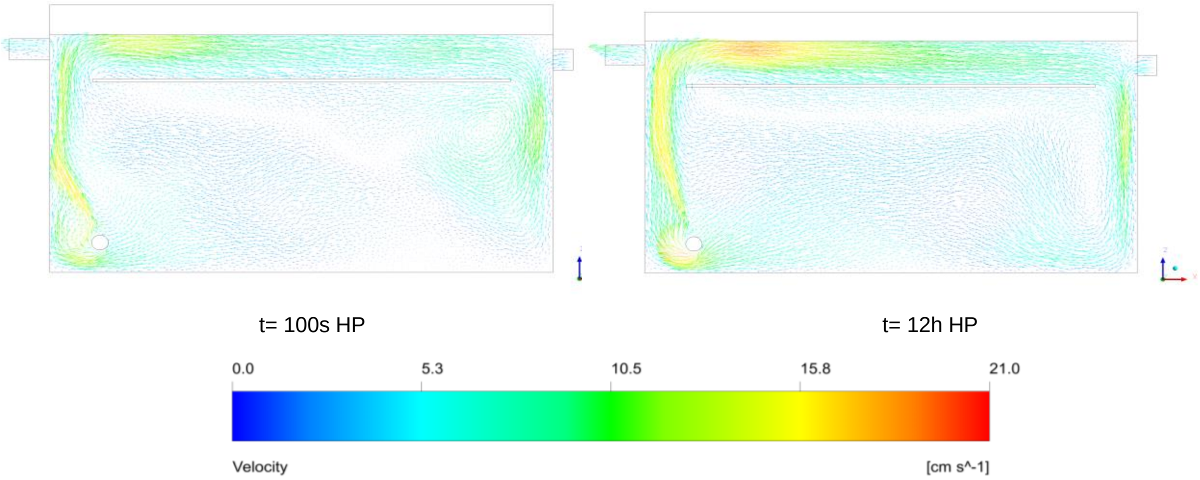

Figure 4 - Vector distribution with radiation for two heating periods. Plane Y=0.625m.

The following analysis is night ventilation. As previously described, the operation is to apply a constant air velocity in the inlet slot for 12 hours. And, by advancing the previous heating state This process is done by advancing the previous heating state with radiation involved. The results of this technique at 12 cooling periods are presented in Figure 5 and Figure 6.

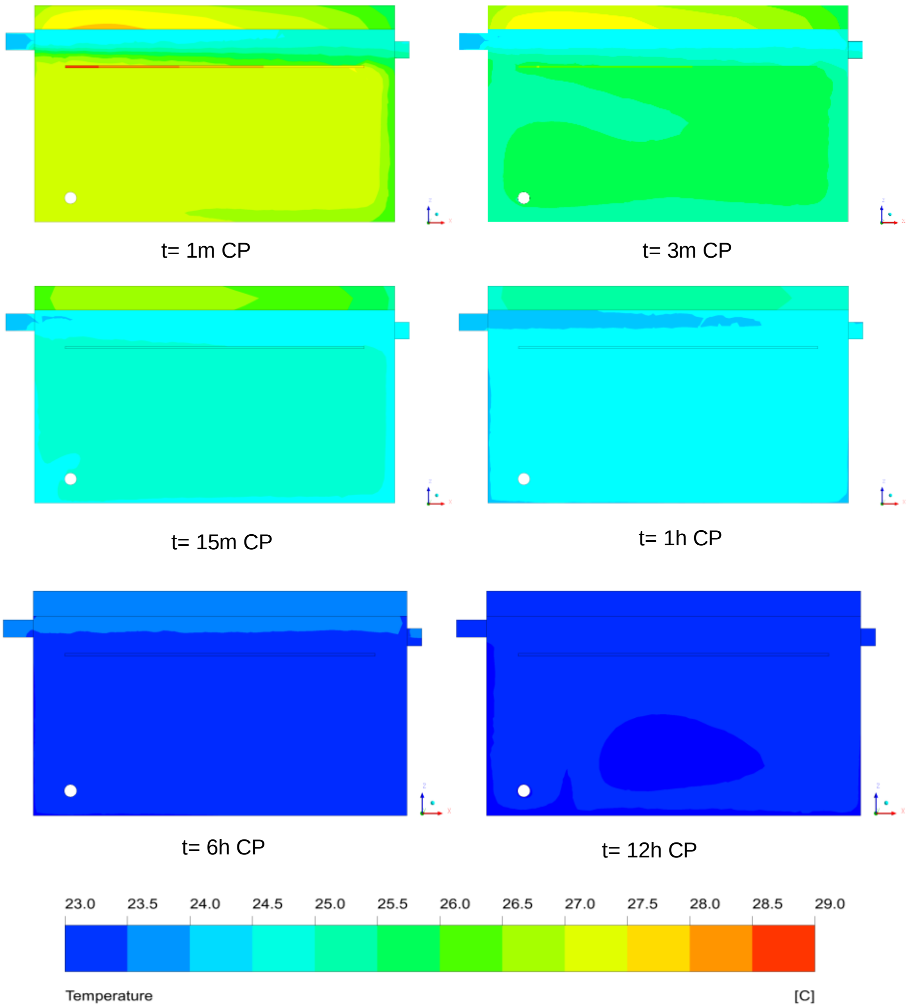

Figure 5 - Temperature contour profiles with radiation for six cooling periods. Plane Y=0.625m. CP - Cooling Period

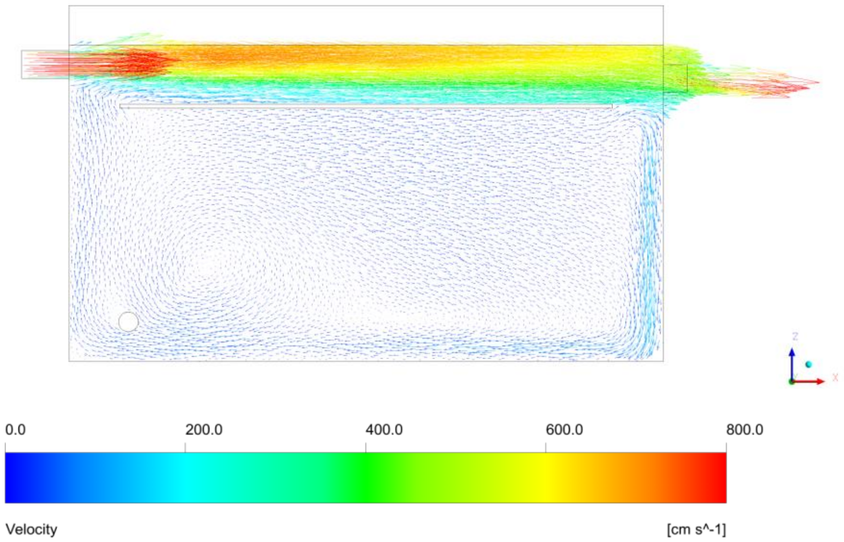

Figure 6 - Velocity vector for 60s of operation time (the average velocity below the suspended ceiling is 50cm/s). Plane Y=0.625m.

As it is possible to observe by analysing the temperature contours. It takes less than 9 hours, after the start of process, to uniformize the air temperature. This effect can be explained by the highly transient process of replacing the air within the space with cool ambient air. This exchange it's done through the gaps of the suspended ceiling, Figure 6. Globally, in the end of the cooling period, the results illustrate that the model is completely cooled, ready for another cycle.

Suspended Ceiling Influence, HP

The results corresponding to the several design scenarios are presented. This was done with a goal of observing the influence of the gaps on the temperature and mass flow rate through the slots. Four different vertical temperature profiles retrieved from the simulations are drawn in Figure 8 and Figure 9 for all the desired scenarios $(32 \times 32\, mm, 64 \times 64\, mm, 90 \times 125\, mm, 240 \times 120\, mm, W/o)$ and for two heating periods – 5 minutes and 12 hours. For the sake of better visualization, the locations of the suspended ceiling and the slab are represented by black dashed lines. The location of the vertical lines (from the floor to the outer wall of the slab) is shown Figure 7. Each line has 100 measuring points. It is important to highlight that all the simulations were performed with exactly the same boundary conditions.

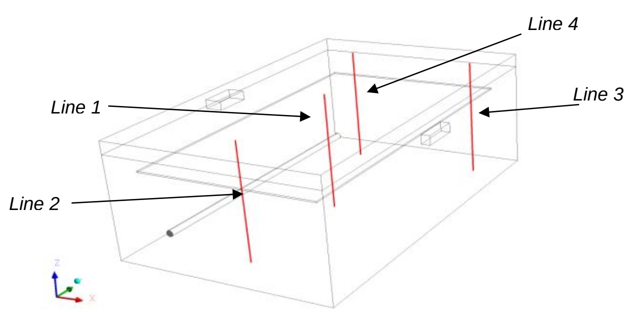

Figure 7 - Location of the vertical measurement lines. Line 1 (0.375; 0.625; 0 : 0.45) Line 2 (0.375; 0.15; 0 : 0.45) Line 3 (0.65; 1.1; 0 : 0.45) Line 4 (0.2; 1.1; 0 : 0.45)

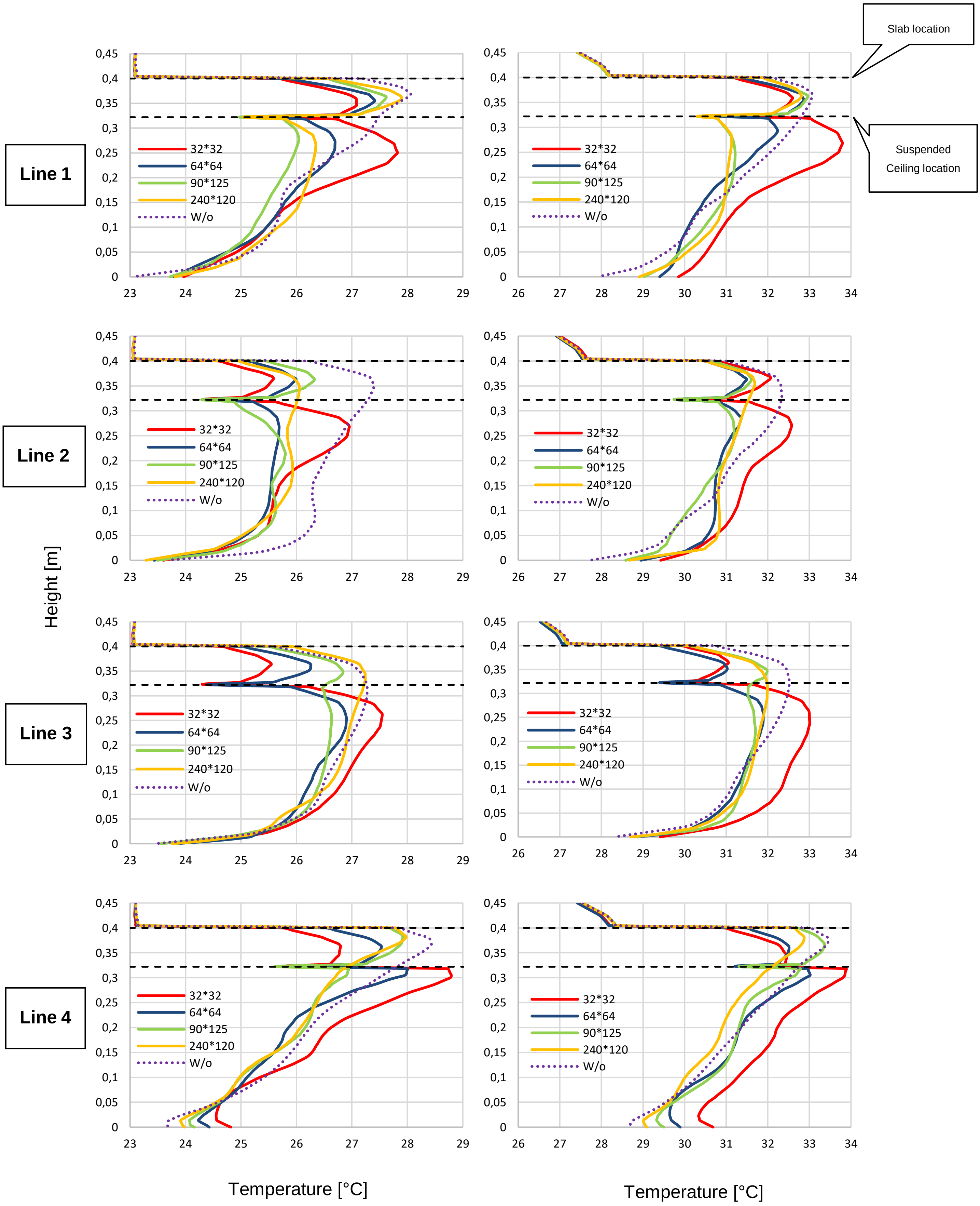

Figure 8- Vertical temperature profiles along the four measurement lines for 5 minutes of operation time (HP), with the 5 different design scenarios.

Figure 9 - Vertical temperature profiles along the four measurement lines for 12 hours of operation time (HP), with the 5 different design scenarios.

In Figure 8 and Figure 9 we can see that the temperatures calculated below the suspended ceiling (representing the occupation zone), are higher in the scenario with smaller peripheral gap - Red line. This is easily explained by analysing the mass flow rate through the inlet/outlet slot,Table 2. Here we see that the values for the two periods are relatively low compared to the other scenarios, which means that the suspended ceiling is restricting the mass flow rate through the peripheral gaps and working as an insulator between the two spaces keeping away the desirable heat exchange between the slab and thermal plume. For the case without suspended ceiling relatively high temperatures are registered, as well, in the occupation zone. Surprisingly, these two configurations are frequently encountered in buildings. Finally, for the remaining scenarios, it is possible to observe, by analysing the graphics that good results are provided in the occupation zone. This shows that, specially for the wider gaps $(90 \times 125\ mm, 240 \times 120\ mm)$, there is a temperature homogenization in this zone due to the fact the higher temperature stratification level is now above the suspended ceiling (the opposite occurs in the scenario with a gap of $32 \times 32\ mm$) and greater heat exchanges may occur with the slab. It is important to highlight that the property value of conduction of the suspended ceiling plays an important role in this homogenization.

Table 2 - Results of the fluid mean temperature and the mass flow rate at the slots in two heating periods for the different scenarios.

	Sc1 $\mathbf{32 \times 32\ (mm)}$		**Sc2** $\mathbf{64 \times 64\ (mm)}$		**Sc3** $\mathbf{90 \times 125\ (mm)}$		**Sc4** $\mathbf{240 \times 120\ (mm)}$		**Sc6** $\boldsymbol{W/o}$	
$\boldsymbol{Time}$	5 m	12 h	5 m	12 h	5 m	12 h	5 m	12 h	5 m	12 h
$\bar{\boldsymbol{T}}\ [\boldsymbol{K}]$	28.15	33.838	27.937	33.235	28.028	33.219	28.203	33.188	28.369	33.211
$\dot{\boldsymbol{m}}\ [\boldsymbol{g/s}]$	0.2243	0.2221	0.2597	0.3295	0.319757	0.35607	0.285919	0.353442	0.3628	0.3451

By analysing all the previous results, the performance for the gap configuration of $90 \times 125mm$ shows better agreement (lower values for the vertical lines as well for the average fluid temperature) in thermal comfort on occupation zone.

6. Conclusions

The purpose of this study was to investigate the accuracy of CFD modelling of building ventilation flows driven by natural forces, and mechanical forces; and by evaluating the performance of multiple design scenarios for the suspended ceiling in a reduced scale model of an office with the aim of improving thermal comfort.

Comparisons were made between the theoretical CFD calculations (using RNG k-epsilon model) and the experimental measurements for a heating period of 12 hours. The temperature in the overall model for the CFD calculations were slightly higher $(\sim 2°C)$ and exhibited a different overall behaviour at the start of the heating period because we defined the external wall boundary condition as convection, and it seemed that the experimental data was losing the same amount of energy regardless of working conditions. For that reason, it is reasonable to assume a constant heat flux loss to simulate experimental data. The efficiency of night cooling was tested, and depends directly on power of the heat source during the day period, the air flow rate during the night period, and the thermal capacity of the slab. It was found that the slab core temperature drops 4.5 degrees in only 6 hours night cooling operation exhibiting how this technique is effective. The size of the suspended ceiling gaps controls the air flow pattern of the thermal plume and the mass flow rate in the inlet/outlet slots. Wider gaps between the suspended ceiling and the walls promote better heat exchange ratios and contribute to involve the slab in the passive technique of cooling. Also, it was concluded that this latter plays an important role in the attenuation of indoor peak temperatures. The configuration with the best performance in $90 \times 125mm$ shows yields the best thermal comfort.

References

[1] P. G. F. Branquinho, "Modelos de Previsão do Consumo Energético no Sector Residencial," *PhD Thesis, IST Lisbon*, p. 93, 2014.

[2] A. S. of H. R. and A. Engineers, "Thermal Environmental Conditions for Human Occupancy," pp. 127–520, 2003.

[3] A. Kayne, "Computational Fluid Dynamics (CFD) Modeling of Mixed Convection," *PhD Thesis, Washingt. Univ.*, p. 70, 2012.

[4] C. E. Walker, "Methodology for the evaluation of natural ventilation in buildings using a reduced-scale air model," *PhD Thesis, Massachusetts Inst. Technol.*, p. 211, 2006.

[5] M. Lança, P. J. Coelho, and J. G. Viegas, "Enhancement of heat transfer in commercial buildings during night cooling - Reduced scale experimentation.," *9th World Conf. Exp. Heat Transf. Fluid Mech. Thermodyn. Brazil*, p. 3, 2017.

[6] M. Lança, P. J. Coelho, and J. G. Viegas, "Enhancement of heat transfer in commercial buildings during night cooling - CFD study and reduced scale experimentation," *Congr. M. Numéricos en Ing. España*, p. 15, 2017.

[7] F. Durrani, M. J. Cook, J. J. Mcguirk, N. B. Kaye, and B. Engineering, "CFD Modelling of plume interaction in natural ventilation," *12th Conf. Int. Build. Perform. Simul. Assoc. Sydney*, pp. 14–16, 2011.

[8] Q. Chen, "Comparison of different k -ε models for indoor air flow computations," *Numer. Heat Transf. Part B Fundam.*, vol. 28, no. 3, pp. 353–369, 1995.

[9] C. Zhang, M. H. Kristensen, J. S. Jensen, P. K. Heiselberg, R. L. Jensen, and M. Pomianowski, "Parametrical analysis on the diffuse ceiling ventilation by experimental and numerical studies," *Energy Build.*, vol. 111, pp. 87–97, 2016.