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

**Amrin Banu Taboleiros Ahmed Satar**

amrin.satar@tecnico.ulisboa.pt

Instituto Superior Técnico, Universidade de Lisboa, Portugal

June 2017

**Abstract**

A numerical investigation of the convective flow in an enclosure was performed. The model is a two-dimensional simplification of a reduced scale model used to investigate the efficiency of natural ventilation and cooling techniques on reducing energy consumption in a building. Night ventilation is used to accomplish that goal since it allows the cold air to cool the building thermal mass reducing and delaying air and wall temperature peaks. As such an innovative solution focus on the use of the plenum, space between the slab and the suspending ceiling, is studied with the purpose of increasing the heat exchange between structural elements of the building and the zone that needs to be cooled.

A commercial code was used to predict the temperatures distribution as well as the flow field inside the enclosure. In a first stage a sensitivity study was performed to investigate the influence of mesh, time step and modeling choices on the accuracy of the results. Subsequently, the use of the surface-to-surface radiation model and real model walls was tested to analyze the influence on the overall problem.

The results show the need of using very small-time steps to obtain convergence of the initial natural convection flow. It was also found that the modeling of the walls in the numerical model reduces the temperatures in the domain. The results also show the effective use of night ventilation on reducing the next day temperature pick.

## 1. Introduction

As Lança et al. suggested in [1] and [2], one way of reducing the energy consumption in buildings, is to make the best use of natural ventilation and cooling techniques. They proposed the use of night ventilation on a real building to address the {Bibliography}efficiency of this strategy on reducing the day peak indoor temperatures. Building thermal mass is essential to accomplish that goal because of is capacity to retain and dissipate heat. However, in modern constructions the amount of exposed thermal mass is kept to a minimum and elements like suspending ceilings represent an insulation between the thermal elements and the occupied zone. As such the focus of their study is the use of the plenum (space between the slab and the suspending ceiling) for an increase of heat exchange between the structural elements of the building and the zone that needs to be cooled. In order to investigate this, an experimental reduced scale model was

constructed for experimental data evaluation [1] and numerical simulations were conducted on CFD program Fluent 14.0 [2]. To test this solution on decreasing the temperature in a room during the day experiments with the duration of 3 days were performed. They include two periods: the occupied room during the day, where powers of 30W or 50W are being emitted, and the night ventilation time where a fan is turned on, injecting into the room a specific air flow rate.

With this in mind, the initial goal was to go further on the numerical modeling of the problem, more specifically, in the investigation of numerical parameters like time step and radiation influence on the results when compared with experimental data. However, when it comes to natural convection problems, difficulties in achieving convergence have long been reported, Linden et al. [3] and Cook et al. [4], and this problem was witnessed when starting the simulation process with the 3D CFD model. It was possible to see that the results are highly sensitive to modeling choices like boundary conditions and solver settings.

As such, a 2D model simplification of the problem was made to reduce the calculation cost and simplify the parametrical study on the flow field and heat transfer analysis.

## 2. Method

The enclosure was constructed based on the 3D simplification of the reduced sale experiment presented on the work of Lança et al [1]. The problem will be focused in the middle plane of the enclosure ($y = 0.625\ m$), and in a first stage of the problem the extruded polystyrene (XPS) insulation panels, as well as the concrete slab, will be omitted. The two openings slots of the experimental setup were simplified into a projection which leads to two lines with the same width. The peripheral gaps between the suspended ceiling and the surrounding walls were assumed $60\ mm$ in all simulations. A practical visualization of the plane studied as well as the main dimensions is represented in the Figure 2.1.

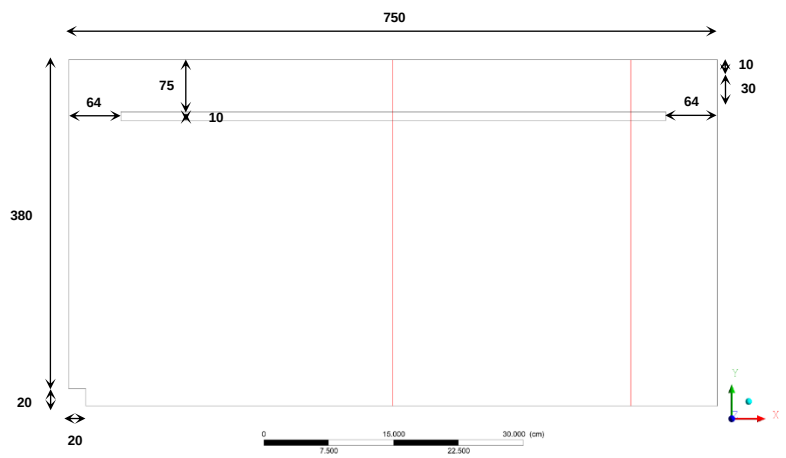


*Figure 2.1 – 2D Model*

With the aim of making a cartesian mesh, the cylindrical heat load of the experimental setup was approximated into a rectangular shape ($2 \times 2\ cm$) in the corner of the 2-dimensional plane.

Therefore, three types of cartesian meshes modelled by ANSYS Meshing software (ICEM), were generate. Taking into account that the height of the suspended ceiling constrains the maximum element size of the cartesian mesh, $1cm$, the first mesh has 2 996 cell elements (coarse, $element\ size\ of\ 1\ cm$), the second one has 47 936 cell elements (fine, $element\ size\ of\ 1/4\ cm$) and finally the third one has 191 744 cell elements (very fine, $element\ size\ of\ 1/8\ cm$).

In this first stage of the study only the heating period will be evaluated and for simplicity the inlet air takes the value of $23ºC$ in all simulations. And as the levels of convergence of the residuals are very sensitive to the large amount of settings in Fluent, the selection of the general setup and the main boundary conditions, as a first guess, were taken from the 3D simulations. Therefore, the pressure-velocity coupling is achieved using the Coupled algorithm, the second-order upwind differencing scheme is used to evaluate the convection terms for the Navier-Stokes equations, the energy equation and the turbulent transport equation and finally under-relaxation factors were taken as default.

Concerning the boundary conditions, a power load of 50 W, emitted in the heating period, is equivalent in our problem to a constant heat flux given by:

$$\dot{q} = \frac{P}{A} = \frac{50W}{0.04m} = 1250\ W/m$$

Where $0.04m$ is the length of the heat source, Figure (2.1)

The external wall boundary conditions have a fixed convective heat transfer coefficient of 4 $W/m^2K$, which was estimated also analytically using a correlation for a hot horizontal/vertical plate facing upwards and loosing heat by natural convection. To define static pressure, “pressure outlet” is used in the two opening slots.

Concerning the density of the main fluid, air, the Boussinesq assumption in our problem only gives converged results below timesteps of 0.01s (trial and error), and for this setup, 2 seconds of the heating period is simulated in 3minutes of computational time (or 15 minutes of heating lasts approximately 23 hours of computational time) which leads to excessive simulation time. Therefore, Ideal gas approximation was chosen for density specification in all the remaining simulations.

Since the absolute velocity was almost the same for each point of the domain is value in the Courant number was neglected and its only relevant the value of $\frac{\Delta t}{\Delta x}$.

In the choice of mesh/timestep size, it was considered, that to avoid the numerical irregularity and have consistent results without being too time-consuming (comparing with the simulation of mesh 1/8cm, timestep=0.01) a mesh of 1 cm with a 0.01s timestep (smallest courant number), is a reasonable choice.

.

## 3. Results

### Heating Period

A representative study with a sinusoidal ambient temperature, through all the heating period (12h), was made. The computational ambient temperature was defined as a sinusoidal temperature function), that simulate the typical temperature of the day inside the laboratory where is located the experimental setup. In Figure (3.1), temperature contours are presented for the 12h of the heating period.

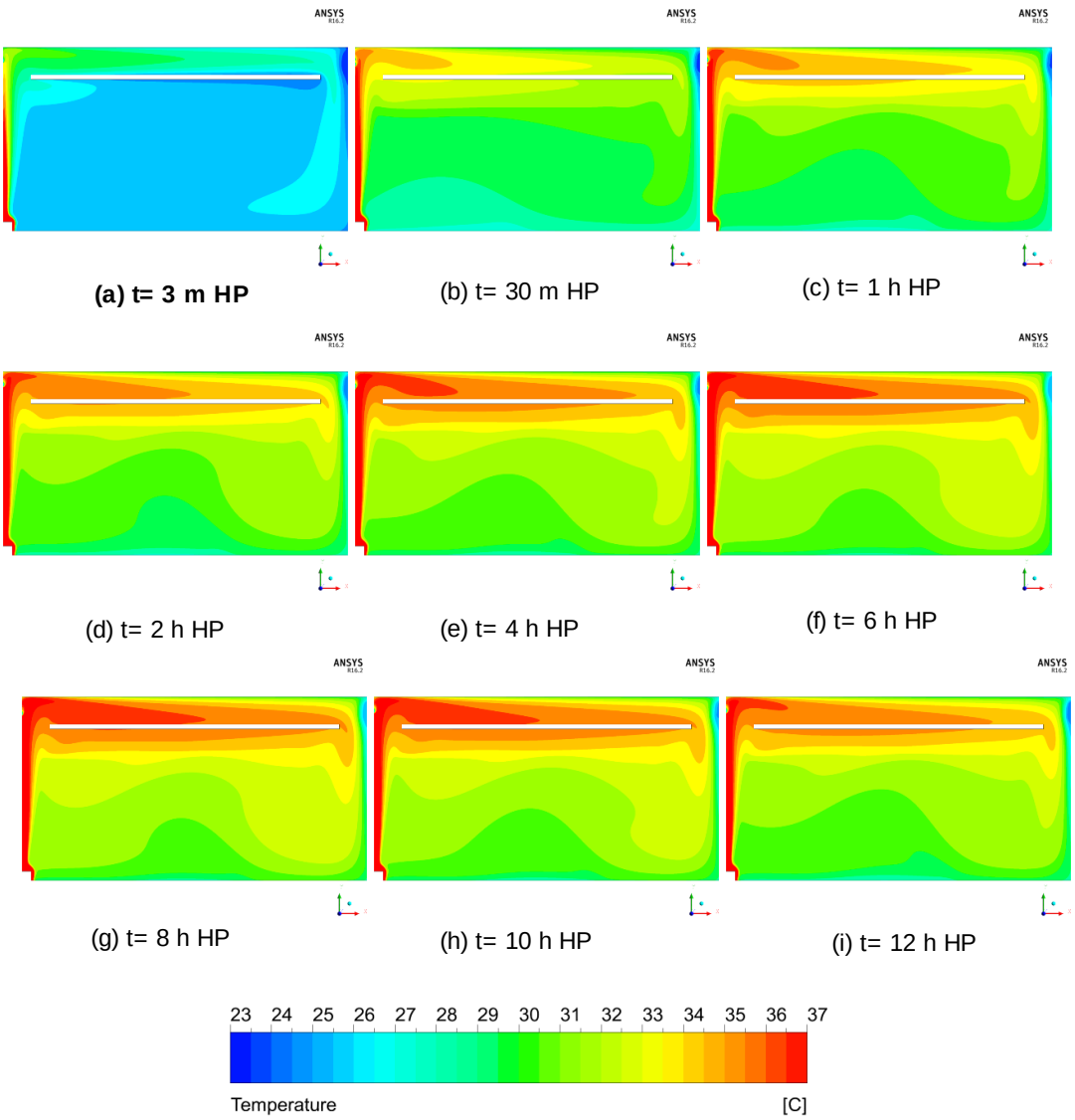


*Figure 3.1 - Temperature contour profiles for twelve heating periods. HP - Heating Period*

It is possible to see, by analyzing the temperature contours, that the suspended ceiling plays an important role in the behavior of the flow, mainly changing the temperature distribution in the space. More specifically, working as an insulator between the two spaces keeping away the undesired thermal inertia of the plume, leading to a “imprisonment” of the plume in the plenum, and consequently to greater heat exchanges with the slab. This results in a low vertical temperature gradient and a good homogenization of the occupation zone

**Radiation**

Evaluating the impact of modelling radiation, a comparison between temperature distribution at the beginning (3 minutes) and in the end (12 hours) of the heating period the is presented in Figure (3.2). The major differences of the Figure (3.1) (i) and Figure (3.2) (b) are the temperature gradient in the occupation zone. The buoyancy driven ventilation strength is stronger in the case without the radiation model because all the power (50W) is transferred to the fluid by convection while in the case with radiation is transferred partly by convection and radiation. A major difference can also be distinguished visually, zones closer to the power source (under the suspended ceiling and the floor near the heat source) are heated by this phenomenon.

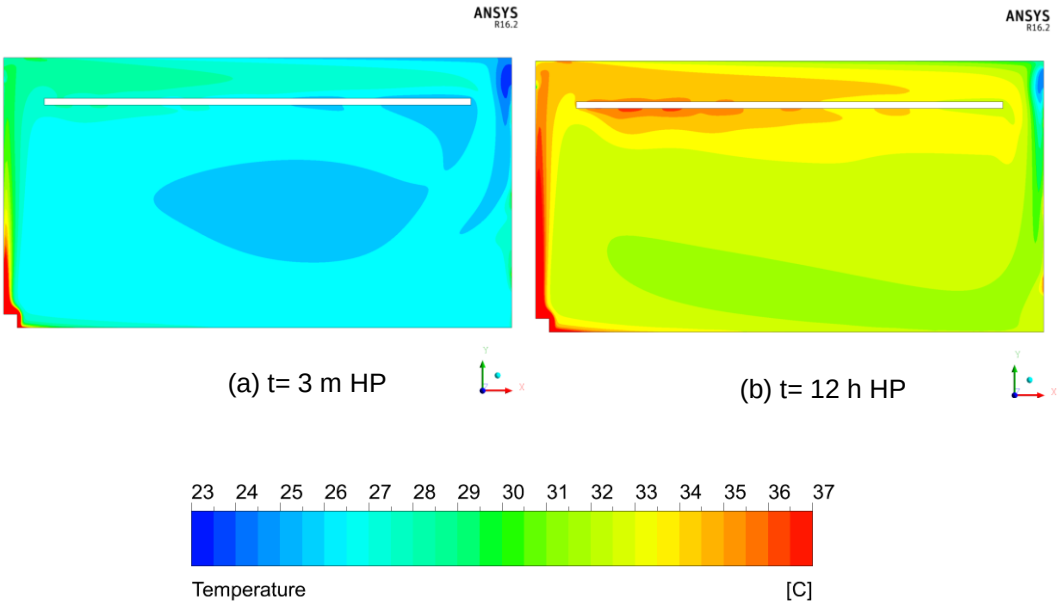


*Figure 3.2 - Radiation temperature in two heating periods. HP - Heating Period*

**Modelled Walls**

The influence of using realist walls was study. This study differs only theoretically from the previous study, except for the addition of a fluid domain in the inlet/outlet slots.

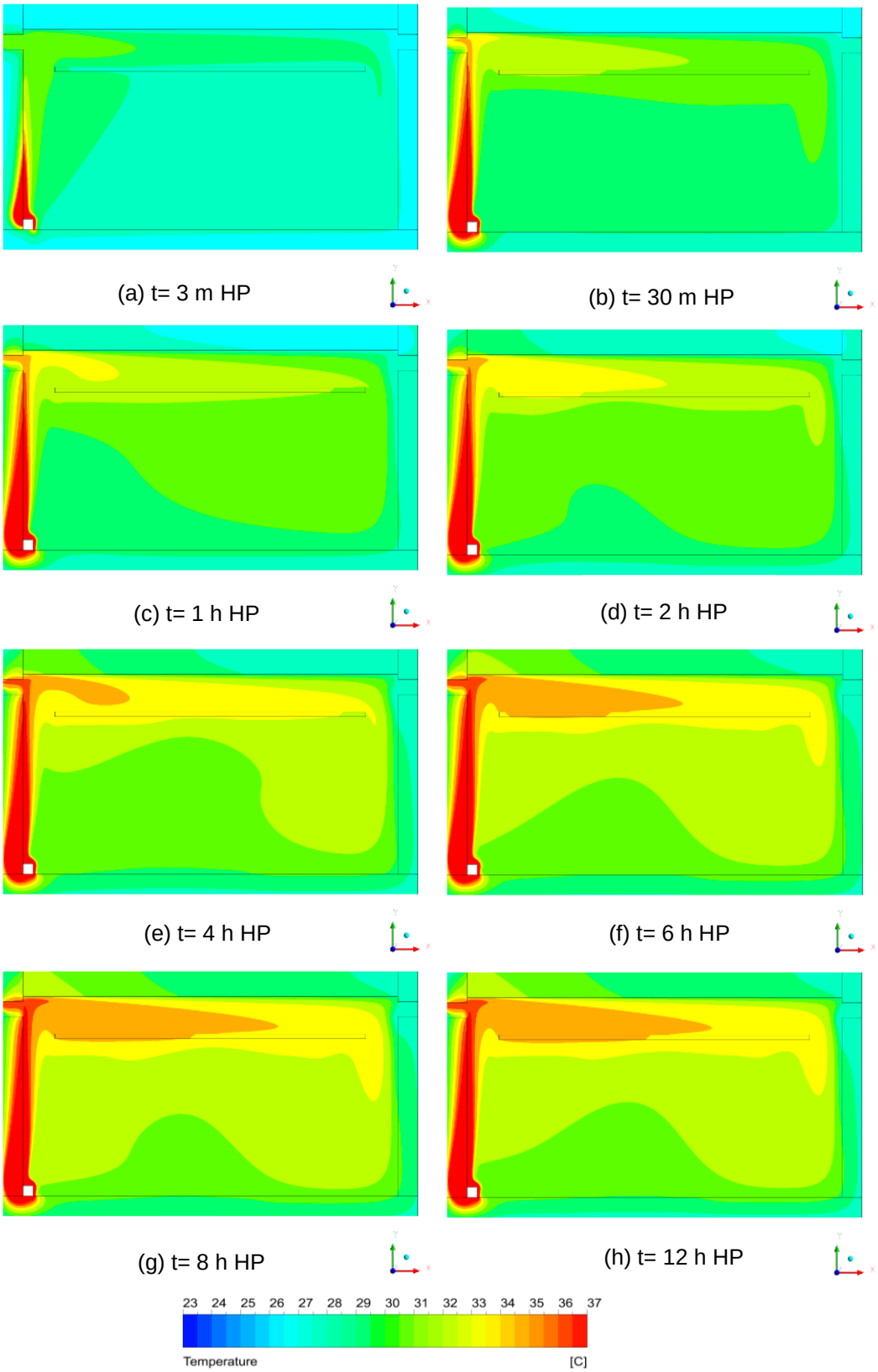


*Figure 3.3 - Temperature contour profiles with exterior walls for eight heating periods. HP - Heating Period*

As it is possible to see in Figure (3.3), the storage and dissipation of heat in the XPS walls, leads to smaller buoyancy forces.

### Night Cooling

Figure (3.4) and Figure (3.5) presents the temperature evolution, of the previous simulations during 36 hours, two heating periods, one cooling period, in two points of the fluid domain. These points are located in the middle of the occupation zone (point 1) and in the middle of the plenum (point 2).

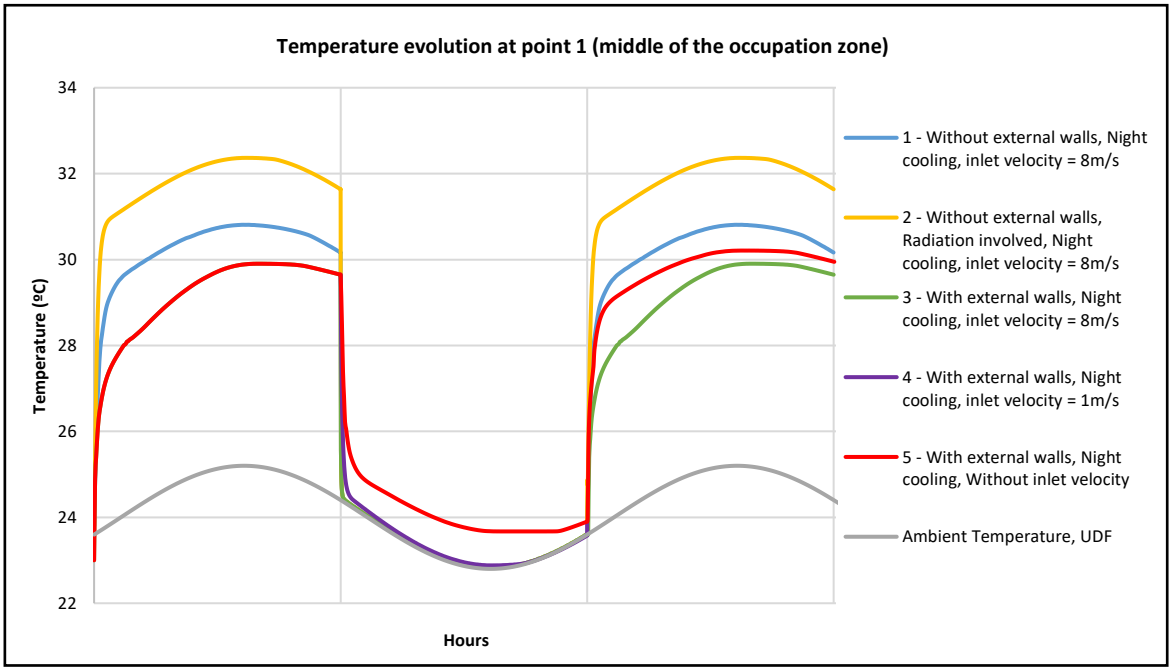


*Figure 3.4 - Temperature evolution at point 1 (37.5 cm; 20 cm), through 36hours*

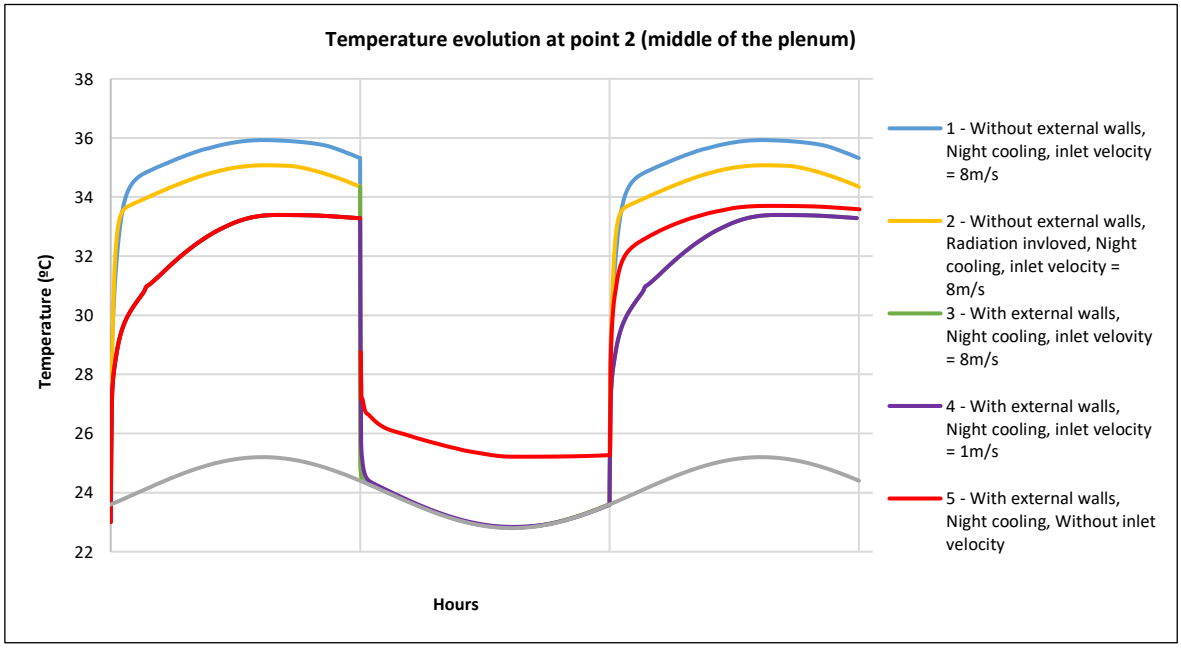


*Figure 3.5 - Temperature evolution at point 2 (37.5 cm; 36.25 cm), through 36hours*

Since the computational model was created based on a number of assumptions due to complexity of the problem, a direct comparison with the experimental data is not possible. Therefore, a different qualitative analysis was made. In general, the numerical temperature/velocity profiles and the behavior of certain temperature points with time, show close resemblance with their experimental counterparts (REF). Specifically, temperatures are lower near the floor and increase with height (temperature stratification). Velocity profiles also show similar results, in terms of recirculation and magnitude of the velocity.

Several conclusions can be drawn about the effect of using modelled walls (Figure 3.4, Figure 3.5, line 1 and 3/4/5). The first one, its clearly that lower temperatures are registered, due to the storage and dissipation of heat in the exterior walls of XPS near the power source. The second, is that there is a delay in the temperature evolution (higher response time) through the heating period, which can be seen by the slope between 8:00 HP – 16:00 HP and 18:00 HP – 20:00 HP.

Regarding the efficiency of night ventilation, the results shows that effectively exists a reduction in the next day temperature peak (Line 3/4 - Line 5). This is due to higher temperatures in the end of the cooling period without night ventilation, which makes the second heating period with higher domain temperatures.

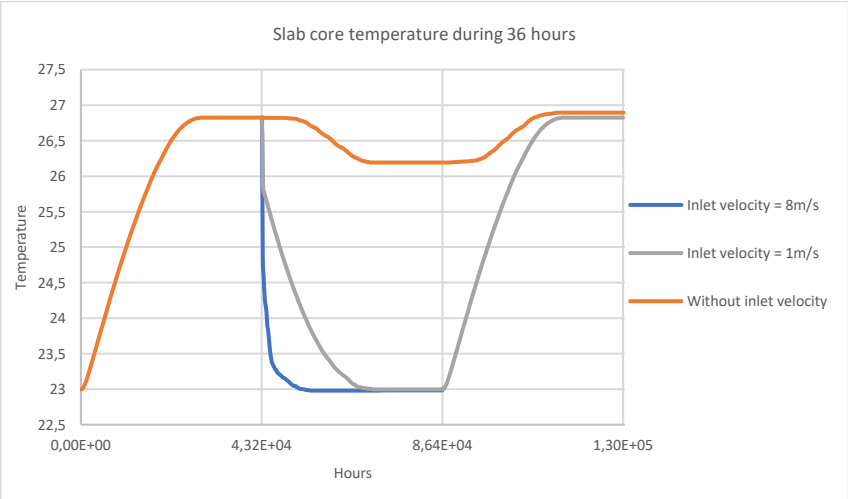


*Figure 3.6 - CFD temperature in slab core (37.5 cm; 42.5 cm) for 36 hours (HP – CP- HP)*

Figure 3.6 shows that the slab is effectively and efficiently cooled by the outside air coming from the inlet slot and passing all the way to the outlet slot, through the plenum formed by the suspended ceiling and the slab. It is possible to see the highly transient velocities in the domain, more specifically in the plenum, causes high heat transfer rates with the slab.

## 4. Conclusion

The goal of the work presented in this project has been to assess the modelling requirements and accuracy of CFD computations using RANS equations for forced and free convection flows in 2-D building enclosures. Ansys FLUENT 16.1 was employed for this purpose. The Boussinesq approximation was used, in a first stage, to study the buoyancy effects of the incompressible flow field, but for reasons of convergence, a solution was only possible for timesteps below 0.01s. This configuration was very time consuming, which led to the choice of using the ideal gas approximation to simulate the flow field.

It was found that only certain combinations of mesh density and time step size give physical/realistic results. The Courant number correlates these two variables and it was concluded that a smaller number should be employed to obtain solutions of acceptable engineering accuracy. In our case, a Courant number of 0.01.

The influence of the modelling external walls was studied, revealing that the thermal mass also plays an important role in the attenuation of indoor average/peak temperatures, working as a sink of heat and having a direct influence in the response time on the temperature evolution in the fluid domain.

The efficiency of night cooling was tested, three cases were simulated in order to evaluate the temperatures in the domain, in the next working day (12 hours). It was found that depends directly on, the air flow rate during the night period, and obviously, in power of the heat source during the previous heating period. The case without any type of air insufflation, during the night period, leads to higher domain temperatures in the beginning of the next heating period, which normally turns the next day average temperatures higher. In our case, the average domain temperatures, in the next heating period its approximately 0.3 ºC bigger. Regarding the slab core temperatures, the difference in the use of night ventilation is 3 ºC difference at the end of the cooling period.

Good quantitative comparisons could not be obtained because many simplifications were made, regarding the localization and design of the heat load, losses through the walls, and other parameters from the experiment. Nevertheless, the computations for velocity and temperature profiles shared qualitative agreement with the experimental data.

In summary, it is demonstrated in this work that CFD can model the flow field and heat transfer in building enclosures quite accurately with a proper choice of mesh density and the time step size. Night ventilation techniques can contribute to decrease significantly the cooling load of A/C. Designing buildings to be more ventilated more efficiently, can help reduce the overall cost of energy and promote a better indoor thermal comfort, since architectural elements, such as a suspended ceiling could work as an insulator.

## 5. References


[1] 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. 8, 2017.

[2] 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 .,” p.15, 2017.

[3] P. F. Linden, G. F. Lane-Serff, and D. A. Smeed, “Emptying filling boxes: the fluid mechanics of natural ventilation,” *J. Fluid Mech.*, vol. 212, no. 1, p. 309, 1990.

[4] M. J. Cook and K. J. Lomas, “Buoyancy-driven displacement ventilation flows: Evaluation of two eddy viscosity turbulence models for prediction,” *Build. Serv. Eng. Res. Technol.*, vol. 19, no. 1, pp. 15–21, 1998.