The Influence of Boundary Conditions on the Natural Ventilation in Buildings

Authors:

José Carlos Teixeira, University of Minho, 4800-058 Guimarães, Portugal, jt@dem.uminho.pt
Ricardo Sousa Lomba, University of Minho, 4800-058 Guimarães, Portugal, rsl@dem.uminho.pt
Pedro M. Lobarinhas, University of Minho, 4800-058 Guimarães, Portugal, pl@dem.uminho.pt
Eurico Seabra, University of Minho, 4800-058 Guimarães, Portugal, eseabra@dem.uminho.pt
Luis F. da Silva, University of Minho, 4800-058 Guimarães, Portugal, lffsilva@dem.uminho.pt

Abstract — *Buildings are an important source of energy consumption in developed countries. In Europe, buildings account for approximately 40% of final energy consumption. Increasing demands for thermal comfort and concerns on health and hygiene levels in confined spaces are likely to push this figure upwards. Ventilation is certainly a major tool in providing a desired level user's satisfaction.*

In this context, natural ventilation is an attractive method for guaranteeing an appropriate ventilation at a lower energy consumption. However the traditional techniques for the design of ventilation systems are of difficult application in such conditions. Factors such as thermal gradients, wind velocity, aperture layout are paramount to their correct operation.

Computational Fluid Dynamics techniques are becoming increasingly attractive in the design of ventilation systems. They can provide tailored solutions to unique systems in a flexible manner and at a low cost, as complex phenomena and sensitivity analysis can be included into the modelling. One of the crucial questions concerns the correct definition of reliable and appropriate boundary conditions. They often involve extending the computational domain into the vicinity of the building in order to give inlet/outlet conditions that are not dependent upon the computational domain.

In this paper, a CFD model has been implemented in order to simulate the air flow inside a standard building. The model solves the mass, momentum and energy for the air flow, coupled with the k-ε turbulence model. The equations are solved by a FV discretization technique in a structured grid. The results, tested as grid independent, have shown the influence of the computational domain and solution convergence into the air circulation inside the building.

Index Terms — *Natural ventilation, Sustainable systems, CFD.*

INTRODUCTION

Buildings are an important source of energy consumption in developed countries. In Europe, they account for approximately 40% of the final energy consumption. Increasing demands for thermal comfort and concerns regarding health and hygiene levels in confined spaces are likely to push this figure upwards. Ventilation is certainly a major tool in providing a desired level of user's satisfaction. The natural ventilation is an important and cost effective technique that, when properly used, contributes to the quality of the air indoors as it decreases the concentration of pollutants. It also reduces the death rate caused by respiration issues, frequently due to the bad quality of indoor air. Finally it also improves the thermal comfort of space and reduces the quantity of energy consumed by air conditioning systems. When Natural Ventilation is applicable to a specific architecture design it usually requires a complex study of flow simulation to assure the air flow is appropriately distributed and regions of low interior air quality are absent. In turn this increases the cost of the project.

Various authors have reported the use of Computational Fluid Dynamics (CFD) as a tool for the analysis of natural ventilation. Examples of CFD used in natural ventilation systems include the development of strategies that coupled the simulation for natural ventilation between the Building Simulation (BS) and CFD codes [1], and the study of the potential of the vaulted roofs for improving the natural ventilation through the action of the wind [2]. Another study consisted in the modeling of the transient flow development in a naturally ventilated room, containing a localized heat source [3]. The influence of internal partitions in the performance of the ventilation of a typical office with fresh air supplied by the floor was investigated [4], as well as the effect of opening locations in naturally ventilated room, with an internal source of heat [5]. In addition, various experimental studies on the subject, although not using CFD, contributed to a better understanding of natural ventilation, validating and giving credibility to CFD a design tool in such systems. Such examples include: a set of experiences within an office building in order to assess the possibility of implementing a system of natural ventilation [6]; the study of interactions between wind (inertia) and buoyancy in naturally ventilated building [7]; and the study of the parameters that influence night ventilation [8] amongst others.

In developing a computational model for a natural ventilation system there are key aspects to address. One of the most relevant is the specification of the boundary conditions. One route which is often followed tackles the problem by specifying a value for the air velocity at the inlet opening and the circulation inside the building is driven by the interaction between the buoyancy and inertia forces. This has the key advantage of simplifying the computational domain and its boundary condition, but it also makes an unrealistic assumption regarding the actual flow into the building. However, the flow through the openings results from the interaction between the free stream wind and the building; it becomes dependent upon the building orientation, thermal gradients. This approach requires the computational domain to be extended into the surrounding volume, increasing the computing cost, grid complexity. Therefore the domain definition and corresponding boundary conditions are a matter of optimization in order to guarantee the desired accuracy an acceptable cost. This paper addresses the definition of the computational domain and boundary conditions applicable to the modeling of natural ventilation systems.

DEVELOPMENT OF CFD MODEL

Modeling the fluid flow involves the simultaneous solution of the equations for mass, momentum and energy conservation on a 3D framework. These are discretized in a finite volume scheme and the resulting algebric equations are iteratively solved in the Fluent software. The flow is assumed isothermal (the energy equation is not solved) and turbulent. The standard k-ε model was used for the turbulence and the SIMPLE algorithm enables the pressure velocity coupling.

Once the model is defined with all the closure equations, the first step is to define a test geometry. On this subject one may consider a building of complex shape that, although more realistic, is of limited value for extrapolation to other conditions. Furthermore, the complex geometry makes it difficult to assess the basic mechanisms and separate the individual contributions to air flow inside the building. In this way, a simple geometry was defined as a test case. It corresponds to a building with parallelepiped shape with a dimension of 6 m x 6 m x 3 m. The ventilation apertures consist of an opening (150 mm x 300 mm) in the frontal wall (facing the wind direction), located 150 mm above the floor and placed in the middle of the façade. The other opening is in the shape of a chimney, located on the ceiling. It is placed 225 mm from the back wall (downstream side) and has a cross section dimensions of 300 mm x 300 mm and 1 m high. Fig. 1 illustrates the geometry described.

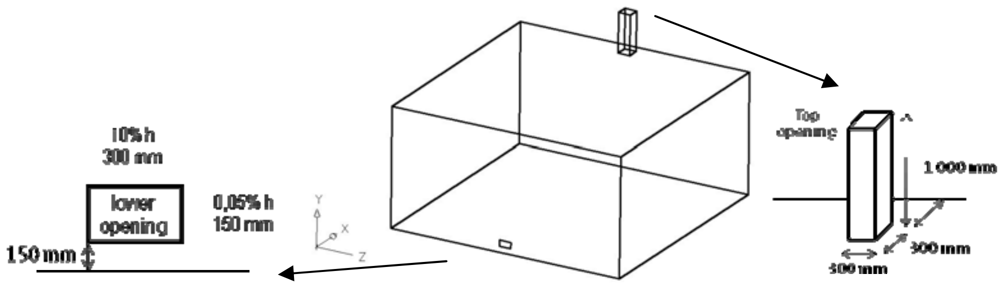


FIGURE 1
GEOMETRY OF THE CASE STUDY

The incorrect characterization (intensity and direction) of the boundary conditions assumed in the model is the major factor of discrepancy in the results obtained from CFD simulations [1]. As the value of the magnitude and direction of the velocity of the atmospheric boundary layer varies with the length and height of the opening, this cannot be defined a boundary condition of the type velocity inlet. This type of problem can be addressed by surrounding the building by a kind of "*box-the-air*", the domain, defining the conditions on the faces. In this way one is able to simulate conditions of natural ventilation. Fig. 2 shows the position of the domain and the building.

Assuming this configuration, the characteristics of velocity at the entrance of the building are not imposed, but the inlet flow (direction and intensity) results from the interaction of fluid flow with the geometry of the building and its vicinity.

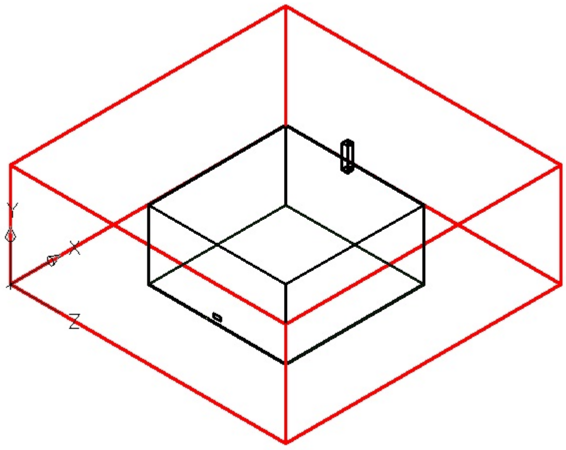


FIGURE 2
POSITION OF THE DOMAIN AND THE BUILDING

Regarding the boundary conditions at the edge of the computational domain, two major alternatives are found in the literature. Taking as the basis the work referred by Wang and Wong [1] and Asfour and Gadi [2], Fig. 3 shows the basic principles that can be considered.

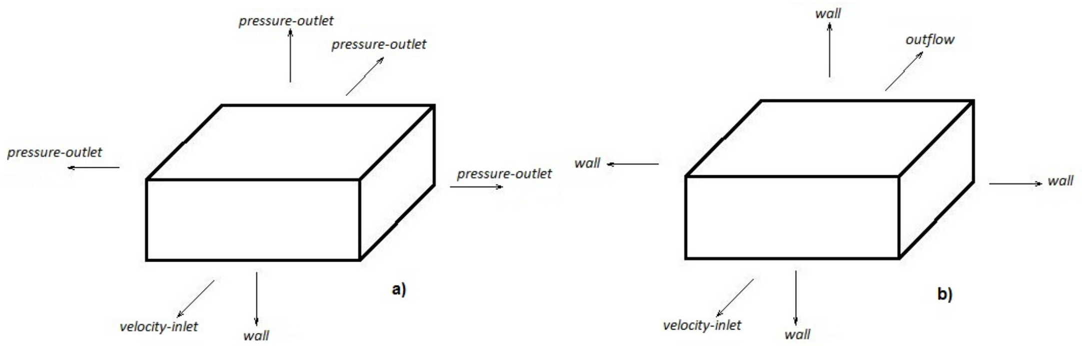


FIGURE 3
THE CF - A) MODEL OF WANG AND WONG [1], B) MODEL ASFOUR AND GADI [2]

The frontal face (upstream) always assumes a *velocity inlet* condition and the bottom a solid wall (zero velocity). The difference concerns the ‘free’ surfaces of the control volume. One option, a), is to assume a *pressure outlet* while the other, b), considers *walls* for three of them and an *outflow* condition on the back (downstream) surface.

Two other configurations were tested. One consists in replacing the boundary condition “*pressure-outlet*” by an “*outlet-vent*” type in Fig. 3-a) and the boundary condition “*wall*” by a “*velocity-inlet*” in Fig. 3-b). The analysis of the best configuration for the boundary conditions was evaluated for different dimensions of the domain and under the same conditions. The mesh was defined as structured (*Hexaedron-submap* type) with an average cell size of 200 mm. Actually, refinements near the regions of high gradients were introduced which required the definition of 50 grid blocks.

The convergence criterion was set at $1x10^{-3}$ for all the variables. It was found that with the use of the boundary condition *vent-outlet* and *pressure-outlet* the convergence was not reached. Fig. 4-a) and b) represent the evolution of the convergence criterion during the iterative process, considering the external domain to be 1x that of the building in all directions. The relative dimension of the domain, for example 1x, means that beyond the limits of the building one has 1.0 times the main dimensions of the building in the three directions (x,y,z). Other sizes for the domain were tested but convergence was not achieved.

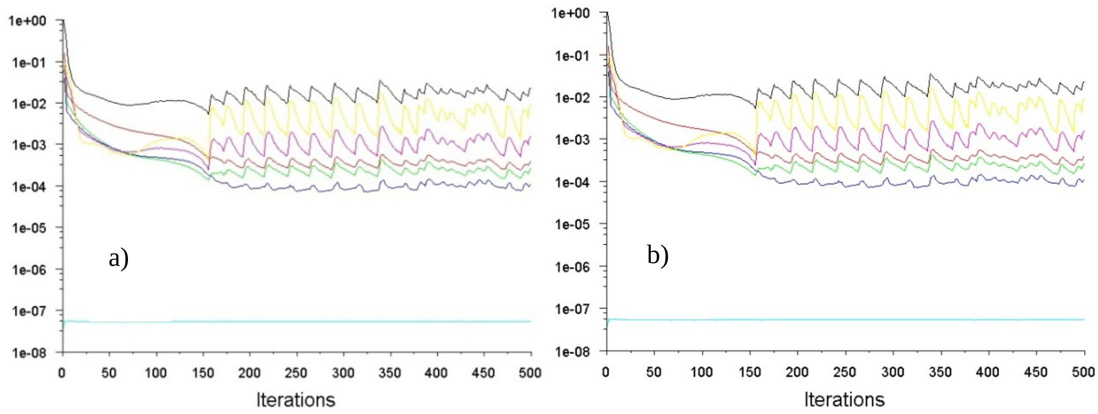


FIGURE 4

RESIDUAL VALUES OF THE CONVERGENCE CRITERION – A) *OUTLET–VENT*, B) *PRESSURE–OUTLET*

As observed in the residues plot (Fig. 4), these show strong oscillations and the behavior is similar for both, rendering these options inappropriate for the flow simulation. Kaye et al. [3] also applied the pressure outlet boundary condition in their work and could not reach convergence. The problem was overcome by establishing a criterion for the stabilization of the simulation. The focus on the boundary condition *outlet-vent* was not also advantageous. Apart from the issues of convergence, the use of these boundary conditions may be inadequate in terms of fluid flow as fluid leakage may occur through the side and top boundaries. Fig. 5 shows the velocity profiles along a direction perpendicular to the free stream velocity (z direction) on a mid plane and at an elevation of y=1.5 m (see Fig. 1 for the coordinate axis and the insert in Fig. 5). The profiles plotted represent various combinations for the boundary conditions and the size of the computational domain. The central region of zero velocity refers to the building.

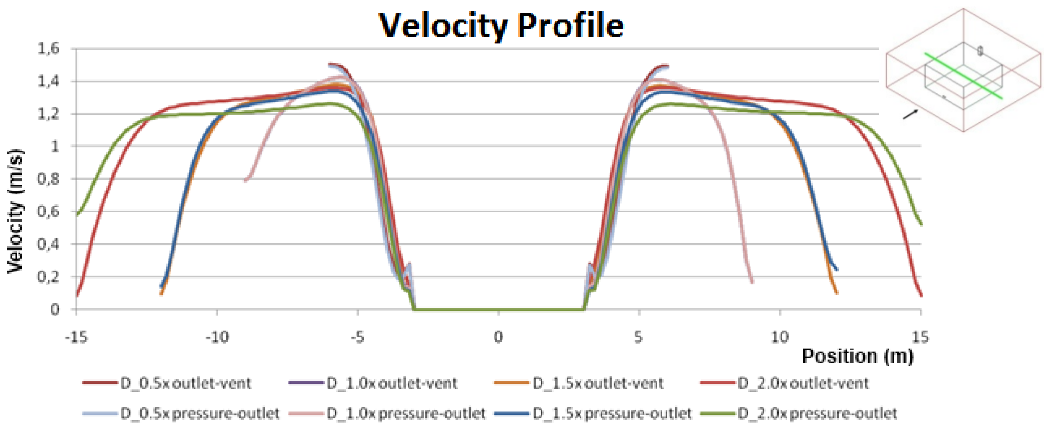


FIGURE 5

VELOCITY PROFILE USING THE BOUNDARY CONDITIONS *OUTLET-VENT* AND *PRESSURE-OUTLET*

The results depicted in Figure 5 should be taken with caution because convergence was not reached for the criterion specified. Nonetheless the data shows that near the side walls of the building the velocity profiles are close though some differences occur near the wall. In addition a lack of symmetry is also observed. This may be due to the non convergence in the solution and suggests that the convergence criterion selected is not too strict. This behavior is not due to the type of boundary condition because this particular choice for the boundary conditions also gives unrealistic conditions at the edge of the computational domain as one expects the velocity to approach that of the free steam velocity at the same elevation (approximately 1.2 m/s). The size of the boundary condition when coupled with the boundary condition shows a very strong influence on the results.

From the study of the other configuration of boundary conditions appeared different results. Using the configuration shown in Fig. 3-b) the simulation convergence is quickly obtained for all dimensions of the domain tested. Fig. 6 represents a typical chart of the convergence criterion for the dimension of the domain corresponding to 1.0x by using the configuration described above.

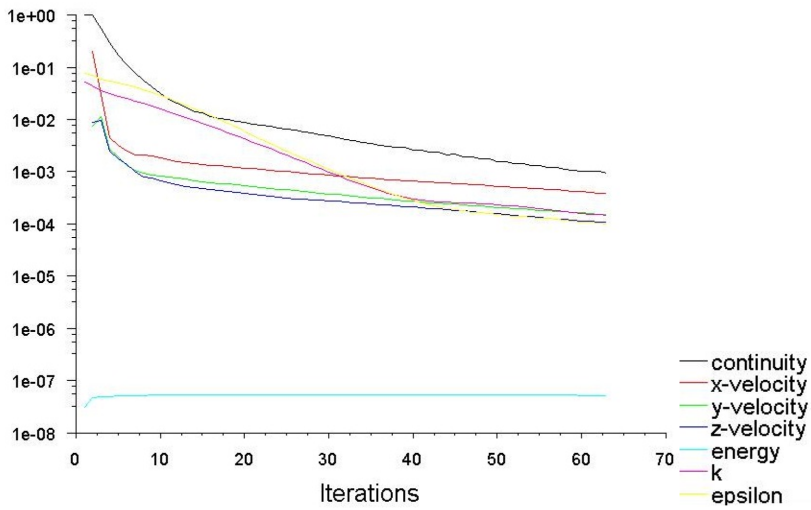
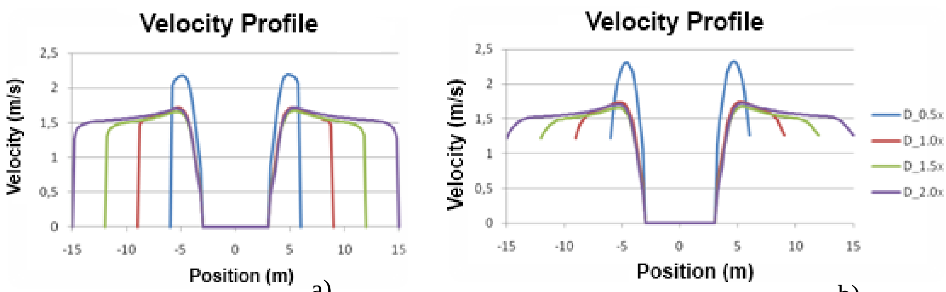


FIGURE 6

RESIDUALS FOR THE CONVERGENCE CRITERION USING THE BOUNDARY CONDITION *OUTFLOW*

The main advantage of these conditions is due to the fact that convergence is easily obtained, which in itself would be a key factor for their choice. Regarding the computational time this configuration is also advantageous because convergence occurs within very few iterations. In view of these observations it becomes evident that the last configuration is more appropriate. Fig. 7 represents the velocity profile at half height of the building (the same positions as in Fig. 5), and in a direction perpendicular to the direction of fluid flow, for various dimensions of the domain. Regarding the boundary condition for the side "walls" of the computational domain, a zero velocity was set in Fig. 7-a). This option is clearly inappropriate because the velocity at the edge is unrealistic. Nonetheless the velocity profile is similar for all sizes of the control volume in the vicinity of the building and the results also show symmetry. Except for the smallest of the computational domain (0.5x the building size) the profiles show a high level of coincidence within their region of overlap. In order to avoid the sudden drop in velocity that occurs along the boundaries of the domain, as shown in Figure 7-a), the boundary condition at the walls was changed in this scheme: i) on the side wall was defined as a tangential velocity, in the direction of flow, equal to that on the frontal wall at the same height (y coordinate) ii) on the top wall by a tangential velocity equal to the maximum wind speed expected at that elevation, and in the direction of the wind. Fig. 7-b) shows the smoothing achieved in the profile.



a) b)

FIGURE 7

VELOCITY PROFILE – A) BOUNDARY CONDITION WALL, B) BOUNDARY CONDITION *VELOCITY-INLET*

This new setting does not affect the convergence rate and accuracy. In terms of flow pattern, this option presents a behavior much meaningful from a physical point of view. As shown in Fig. 7-b), the velocity at the outer boundaries of the control volume approaches the free stream velocity of the wind. By increasing the size of the computational domain, the transition from the building's wall (of high velocity gradients) to the free stream condition occurs in a smoother way. The boundary layer present at the edges is unrealistic but 'removing' it would require a much larger control volume increasing dramatically the computational time. One should keep in mind that this discussion also extends into the vertical (y) coordinate, in the region on top of the building. From the point of view of the building analysis, this route is not necessary because the velocity profile and the flow details near the building wall are not affected by the size of the computational domain. This is not valid for the smallest of the control volumes (0.5 x). For this case the large reduction of the free area combined with the condition of tangential velocity yields a high acceleration of the flow. Flow is also symmetric which validates the grid used and the convergence criterion. As a brief conclusion the important criteria is to guarantee a good convergence and a finely detailed mesh near the high velocity gradients in order to capture the local flow patterns such as reverse flows. Analyzing all the features of the various configurations was chosen the last configuration.

The boundary conditions at the frontal, top and side walls of the computational domain require the knowledge of the wind velocity as a function of elevation. For an atmospheric boundary layer an empirical relationship may be employed:

$$U(z) = U(z_1)\left(\frac{z}{z_1}\right)^{\alpha} \quad (1)$$

where:

$U(z)$ is the wind speed at height z (m/s)
$U(z_1)$ is the wind speed at reference height (m/s), (in this work $U(z_1)$= 1, 2 and 3 m/s)
z_1 is the reference height (m) [typically 10 m]
z is the height (m)
α is an empirical coefficient which is a function of the ground roughness, (in this work α=0.25; construction area)

From the previous discussion (corresponding to the case depicted in Fig. 7-b), Fig. 8 shows schematically the boundary conditions suggested for the analysis of natural ventilation in buildings.

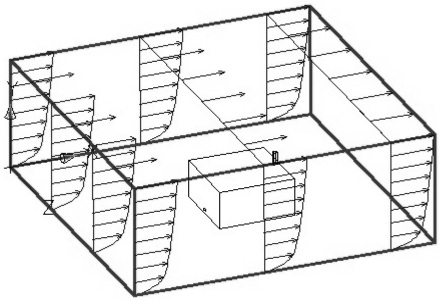


FIGURE 8
SCHEME OF THE VELOCITY PROFILES APPLIED

Taking into consideration the main options for the boundary conditions (Fig. 8) an analysis of the effect of the size of the computational volume upon the flow inside the building was carried out. The increase in size of the domain was performed in a systematic manner by increasing all its dimensions in steps of 0.5x those of the building. All simulations were made under the same conditions (cell type, algorithms, convergence criteria and model) in order to make valid comparisons among them. At this stage, the refinement and optimization of the mesh for each dimension of the domain was already carried out. Initially for the study of each dimension of the domain, it was defined three different mesh resolutions, as shown in Fig. 9. The influence of the mesh was evaluated by the criteria of "Fine-grid convergence index"[9].

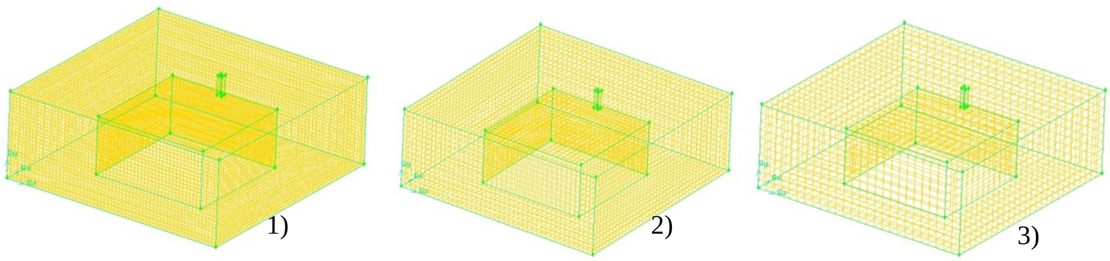


FIGURE 9
DIFFERENT MESH RESOLUTIONS

Fig. 10 shows the various velocity profiles obtained at an elevation of 1.5 m in the middle of the domain, in a direction perpendicular to the fluid flow (green line in the small side sketch).

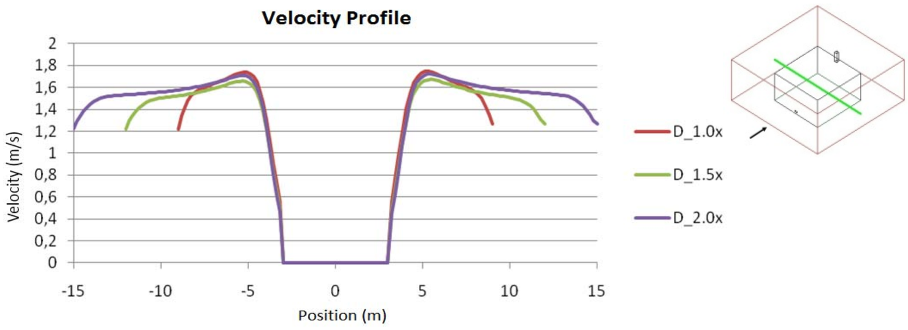


FIGURE 10
VELOCITY PROFILES OF DIFFERENT DOMAINS

All the velocity profiles for the different dimensions of the domains have a peak velocity near the limits of the building, due to the presence of the building and, consequently, by the reduction of the free passage for the fluid. In the zone corresponding to the surrounding area of the building [(-10,-5) and (5, 10)] the boundary layer profiles have a similar development. Interesting to notice that the domain D_2.0x shows the most stable profile in that area. Analyzing the velocity vectors on a horizontal plane (Fig. 11-a) and vertical plane (Fig. 11-b) for the domain 2.0x, it is observed the formation of a recirculation vortex. However, it is observed that the reattachment of the vortex occurs within a very short distance from the edge of the building. It can be concluded that the dimension of the domain is adequate for analyzing the influence of the building because the all the flow details can be captured within the domain. Therefore the domain adopted, D_2.0x, has the dimensions of 30 m x 30 m x 9 m.

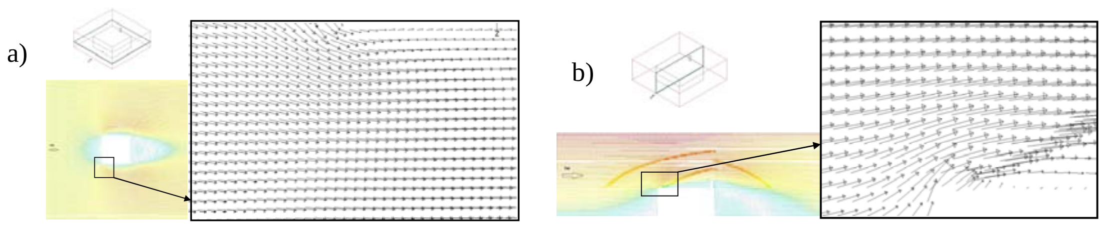


FIGURE 11
VELOCITY VECTORS: A) HORIZONTAL PLAN B) VERTICAL PLAN

Due to the huge dimension of the domain it is not suitable to use a refined and regular mesh for whole domain, because of the high computing resources and computation time required. Therefore, in the most important regions the mesh should be more refined at the expense of other zones. The building is the most important zone, particularly around the openings and its vicinity. This was achieved by dividing the domain into 50 blocks. The cell type was the same in all domain (*Hexaedron–submap*), with a total amount of 2 million of cells. The transitions between meshes with different sizes were made by a *boundary–layer* type in which the size is expanded at a geometric ratio 1.219. So, cells have an average size of 75 mm in central area and are of 550 mm in remote areas. Fig. 12 is illustrative of the mesh used.

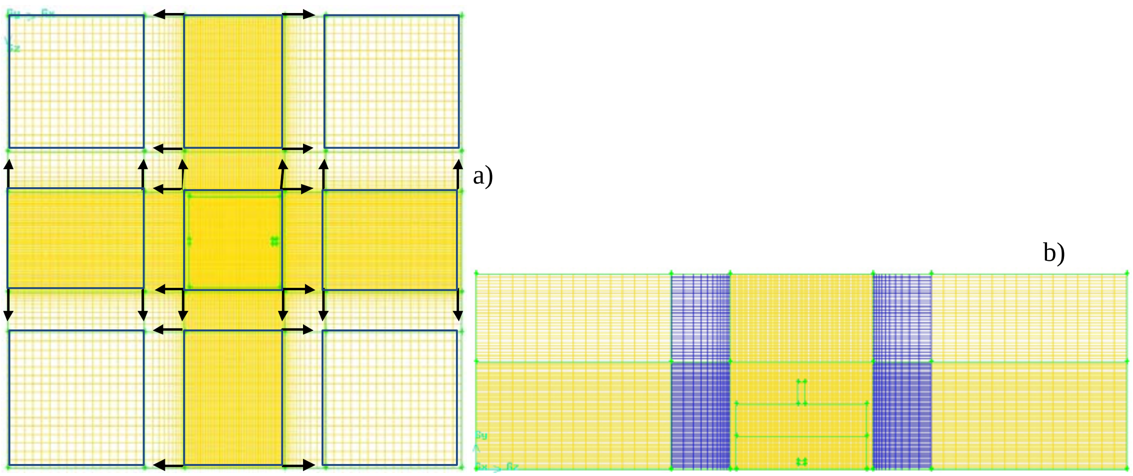


FIGURE 12
CONFIGURATION OF THE MESH USED: A) IN TOP VIEW B) LATERAL VIEW

CONCLUDING REMARKS

Throughout this work it was demonstrated the potential and versatility of CFD simulations in the study of fluid flow, with low cost, fast times and efficiency. The robustness of the results is obtained through a comprehensive study of the boundary conditions that better reflect the physical phenomenon to study. Thus, it is necessary to get a clear and objective idea of the parameters to test for a suitable choice of boundary conditions that best serve the purpose. The results show that convergence should always be of prime concern; otherwise unrealistic results may occur. The boundary conditions should include the velocity profile of the atmospheric boundary layer and the computational domain should be large enough to enable smooth transitions from the building walls. The refinement of the mesh also has a very important role in model quality; therefore its influence on important parameters must be carefully analyzed. The accuracy of a CFD solution is governed by the number of cells in the mesh. In general, a more refined mesh, with more elements, provides more precise values. Both the accuracy of a solution, the hardware and computation time needed is dependent on the refinement of the mesh. It should be a balance between the number of elements used and the simulation time.

REFERENCES

[1] Wang, L. and Wong, N. H. “Coupled simulations for naturally ventilated rooms between building simulation (BS) and computational fluid dynamics (CFD) for better prediction of indoor thermal environment”, *Building and Environment*, Vol. 44, Issue 1, 2008, pp. 95 – 112.

[2] Asfour, O. S. and Gadi, M. B. “Using CFD to investigate ventilation characteristics of vaults as wind-inducing devices in buildings”, *Applied Energy*, Vol. 85, Issue 12, 2008, pp.1126-1140.

[3] Kaye, N. B.; Ji, Y. and Cook, M. J. “Numerical simulation of transient flow development in a naturally ventilated room”, *Building and Environment*, Vol. 44, Issue 5, 2008, pp. 889 – 897.

[4] Lin, Z.; Chow, T. T.; Tsang, C. F.; Fong, K. F.; Chan, L. S.; Shum, W. S. and Tsai, L. “Effect of internal partitions on the performance of under floor air supply ventilation in a typical office environment”, *Building and Environment*, Vol. 44, Issue 3, 2009, pp. 534 – 545.

[5] El-Agouz, S. A. “The effect of internal heat source and opening locations on environment natural ventilation”, *Energy and Buildings*, Vol. 40, Issue 4, 2008, pp. 409 – 418.

[6] Su, X.; Zhang, X. and Gao, J. “Evaluation method of natural ventilation system based on thermal comfort in China”, *Energy and Buildings*, Vol. 41, Issue 1, 2008, pp. 67 – 70.

[7] Lishman, B. and Woods, A. W. “On transitions in natural ventilation flow driven by changes in the wind”, *Building and Environment*, Vol. 44, Issue 4, 2008, pp. 666 – 673.

[8] Artmann, N.; Manz, H. and Heiselberg, P. “Parameter study on performance of building cooling by night-time ventilation”, *Renewable Energy*, Vol. 33, Issue 12, 2008, pp. 2589 – 2598.

[9] Celik. I. B.; Ghia, U.; Roache, P. J. and Freitas, C. J. “Precedure for estimation and reporting of uncertainty due to discretization in CFD applications”, *Journal of Fluids Engineering*, Vol. 130, Issue 7, 2008.