GEOMETRY OF BUILDING'S COURTYARDS TO FAVOUR NATURAL VENTILATION: COMPARISON BETWEEN WIND TUNNEL EXPERIMENT AND NUMERICAL SIMULATION

Abel TABLADA Arch,[1]
Bert BLOCKEN Ph.D[2]
Jan CARMELIET Ph.D[3,4]
Frank DE TROYER Ph.D[5]
Han VERSCHURE Ph.D[6]

[1] Historian Office of Havana, Cuba. Faculty of Architecture, ISPJAE, Cuba. Postgraduate Centre Human Settlements, KULeuven, Kasteelpark Arenberg 1, Leuven, Belgium, abel.tablada@asro.kuleuven.ac.be
[2] Laboratory of Building Physics, Department of Civil Engineering, Katholieke Universiteit Leuven, Kasteelpark Arenberg 40, 3001 Leuven, Belgium, bert.blocken@bwk.kuleuven.ac.be
[3] Laboratory of Building Physics, Department of Civil Engineering, Katholieke Universiteit Leuven, Kasteelpark Arenberg 40, 3001 Leuven, Belgium, jan.carmeliet@ bwk.kuleuven.ac.be
[4] Building Physics Group, Faculty of Building and Architecture, Technical University of Eindhoven, P.O. Box 513, 5600 MB Eindhoven, The Netherlands
[5] PGC Human Settlements, Department of Architecture, Urban Design and Planning, Katholieke Universiteit Leuven, Kasteelpark Arenberg 1, Leuven, Belgium frank.detroyer@asro.kuleuven.ac.be
[6] PGC Human Settlements, Department of Architecture, Urban Design and Planning, Katholieke Universiteit Leuven, Kasteelpark Arenberg 1, Leuven, Belgium han.verschure@asro.kuleuven.ac.be

Keywords: Computational Fluid Dynamics (CDF), CFD simulation, natural ventilation, courtyard buildings, building geometry, compact morphology

Summary

Courtyard and building's geometry are crucial aspects to achieve efficient natural ventilation and healthy indoor conditions in a compact urban environment. The paper aims to validate 2-D computational fluid dynamics (CFD) simulations by comparing air flow conditions inside 5 different cavity ratios (width/height) with 2-D published wind tunnel experiments. The k-ε Standard model, with Non Equilibrium wall functions and first-order-accuracy schemes agrees well with the tunnel experiments concerning horizontal air speed (*U*) and air flow for a cavity ratio W/H=1.0. For the other cavity ratios (W/H=0.7, 0.5, 0.3, 2.0) the secondary recirculation area found in the tunnel experiment is not captured by the CFD simulations. The influence of the courtyard ratios and the presence of obstructions on the potential for natural ventilation are also analyzed. Cavity ratios 1.0 and 0.7 have the highest potential for natural ventilation due to their geometry that promotes the development of a strong vortex and high velocity magnitudes. The presence of obstructions on the courtyard's top corners provokes a weaker flow inside the cavity and therefore lower velocities and lower potential for natural ventilation.

1. Introduction

The Historical Centre of Old Havana in Cuba has a very compact morphology with narrow and orthogonal streets. Because of the high density of the building blocks, the crowded dwellings and the lack of natural ventilation, a large number of buildings do not fulfil the requirements for a comfortable and healthy environment. For this reason, high levels of asthma and other respiratory diseases have been reported in the compact urban environments of Havana (Díaz Véliz et al., 2000). New buildings inserted in Old Havana have to be adjusted to the existing compact morphology and, at the same time, provide a comfortable and healthy environment. Efficient natural ventilation by a proper design of the building and courtyard geometry can be a first step to achieve this goal (see Tablada et al, 2004).

The design of building and courtyard geometry requires insight in the nature and complexity of air flow around and inside buildings. The vast increase in computing performance in the past decades has promoted the use of Computational Fluid Dynamics (CFD) for this purpose (Blocken and Carmeliet 2004, Blocken et al. 2004, Alvarez, 1998). In these studies, CFD generally comprises solving the complex Reynolds-Averaged Navier-Stokes equations in combination with a turbulence model. Often, a k-ε turbulence model is employed to obtain closure. Turbulence models are, by definition, approximations. There exists no turbulence model that is valid for all types of fluid flow. The adequacy of a turbulence model for a given problem is typically assessed by CFD validation, i.e. by comparing the numerical results with carefully obtained experimental data, either from full-scale measurements or from measurements in wind tunnels.

The work reported in this paper is part of a larger research project which final aim is to offer recommendations concerning building and courtyard geometries in order to provide natural ventilation and thermal comfort inside dwellings. The two objectives of this paper are: (1) CFD validation of the air flow conditions inside 5 different cavities by comparison with the corresponding results from wind tunnel experiments reported in literature (Kovar-Panskus et al, 2002) and (2) to compare different geometries of courtyards in terms of air flow and pressure coefficient in order to see their potential for natural ventilation. The present paper does not yet include the study of cross ventilation with more than one courtyard neither the study of thermal effects. However, the results concerning the air flow conditions inside the courtyards obtained from the CFD simulations will be used in a further stage for thermal numerical simulation.

2. Comparison between CFD simulations and the wind tunnel experiment

2.1 Description of the wind tunnel experiment

The wind tunnel experiment with a scale of 1:500 performed by Kovar-Panskus et al (2002) aimed at providing data concerning air flow conditions in five different urban canyon geometries. Their study focused on two-dimensional (2-D) cavities with a height (H) equal to 106 mm and a variable depth (W) in order to create cavities with ratios W/H= 0.3, 0.5, 0.7, 1.0 and 2.0. The ratios of the 2-D urban canyons might be comparable with the geometry of building courtyards. The dimensions of the test section of the wind tunnel have a height of 1.37 m, a width of 1.07 m and a length of 9 m.

The neutrality stratified approach conditions were: free-stream velocity U_{ref} = 8 m s^{-1}, roughness length z_0 = 0.3 mm, displacement height d = 1 mm, boundary layer height δ = 737 mm, friction velocity u^*/U_{ref} = 0.050. The results of the wind tunnel experiment were also compared by Kovar-Panskus et al (2002) with numerical simulations using the 3-D CFD code CHENSI, which is based on the standard k-ε model.

The results of the experiments were given in graphical format, as a vector and streamline plot of the flow in the cavity and as graphs of the horizontal air velocity component (U) along five vertical lines located from the bottom of the cavity to 0.05 m above the cavity (see figure 1b).

2.2 CFD simulations and validation

2.2.1 Geometry, mesh and boundary conditions

The computational domain consists of a 2-D rectangular domain and the cavity. Figure 1a illustrates the rectangular domain which has the same height as the wind tunnel: H = 964 mm. Its length L = 1200 mm. The upstream length is 300 mm. The depth of the cavity is 106 mm. The meshes were constructed with the commercial preprocessor Gambit 2.1.6. Figure 1 illustrates the geometry of the computational domain as well as the computational mesh inside it. The mesh resolution is higher in the region of interest and where high flow gradients are expected (i.e. in the cavity and close to the wind tunnel floor).

The following boundary conditions were imposed: Lower horizontal edges of the section and the cavity's walls: wall with roughness Ks=0.001 m, Inlet: velocity-inlet, Outlet: zero static pressure, Upper edge of the section: wall with roughness Ks=0.0. The horizontal position (mm) of lines are as follows: Line 1: x=309, Line 2: x=331.8, Line 3: x=353, Line 4: x=374.2, Line 5: x=395.4.

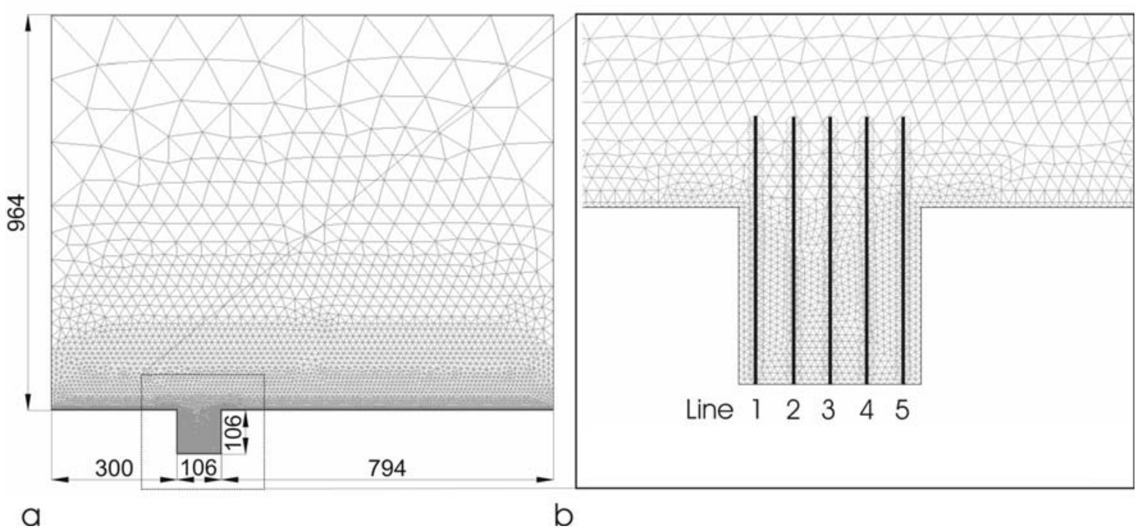

Figure 1 a: Geometry of the computational domain (mm) and distribution of the meshes. b: detail of the cavity W/H=1.0 and the position of the vertical lines along which the results were analysed.

The inlet wind speed profile in the simulations follows a power law (Eq. 1):

$$\frac{U(h)}{U_{ref}} = \left(\frac{h}{Z_{ref}}\right)^{\alpha}$$ Where U = horizontal air velocity, h = height and Z_{ref} = reference height (1)

The power law coefficient (α) was obtained from fitting the power law with the log law used in the experiments. The boundary conditions of the wind tunnel experiment provided in the consulted article include the value of the friction velocity u*=0.41 which is part of the logarithmic law. With α=0.18 a good correlation between the power and the logarithmic law was obtained. Therefore, the values of the inlet wind speed profile parameters are: free-stream velocity U_{ref}=8 m s^{-1}, roughness length z_0=0.3 mm, boundary layer height Z_{ref} (δ-d)=736 mm, friction velocity u*=0.41 m/s, power law exponent α=0.18, turbulent kinetic energy k=1.5 u^{*2} and the turbulence dissipation rate e= u^{*3} /(0.4*(x[1]+ks)).

2.2.2 Grid-sensitivity analysis

A grid-sensitivity analysis was performed in order to select the most appropriate meshes for the CFD simulation. Four cases were simulated with different meshes. Table 1 shows the dimension of the smaller cells and the total number of cells for each case. Figure 2 shows the comparison of the *U* values of cases *x*, *a*, *b* and *c* and of the wind tunnel experiment for lines 2 and 5, which show the highest and the worst agreements respectively.

Table 1: Dimension of the smaller cells (mm) and total number of cells for each case

Cases	Case *x*	Case *a*	Case *b*	Case *c*
Dimension smaller cell (m)	0.0042	0.003	0.0021	0.0015
Total number of cells	5283	10020	21919	40264

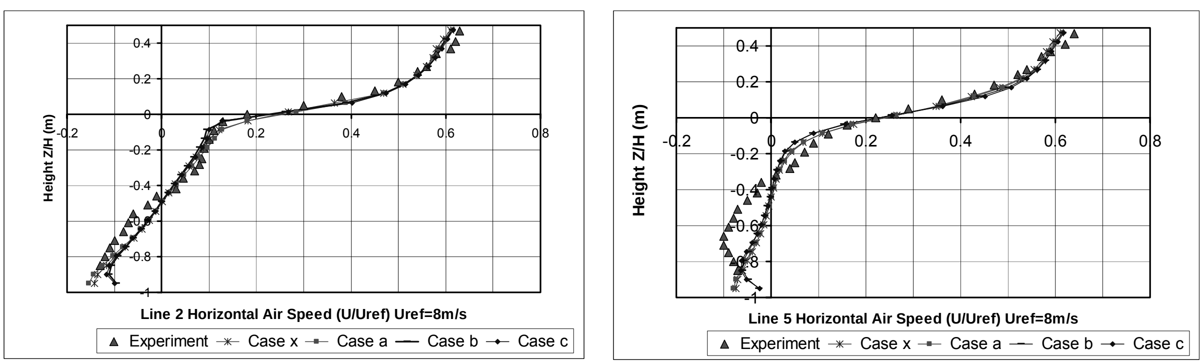

Figure 2 Comparison of horizontal air speed (U/Uref) between the cases x, a, b and c with the wind tunnel experiment. On the left: U/Uref of line 2. On the right: U/Uref of line 5.

Case *a* shows the best agreement for every line taking into account the values inside and outside of the cavity (R^2 from 0.983 to 0.996). Taking into account the values of *U* inside the cavity alone, case *b* shows the best agreement for lines 2, 4 and 5 (R^2 from 0.927 to 0.981), but on the contrary, in line 1 case *b* had very low correlation (0.63) in comparison with case *a* with has R^2 = 0.90. Thus, the meshes of case *a* are going to be used for the rest of the simulations since their agreement with the experiment is higher taking into account all vertical lines.

2.2.3 Parametric analysis of models

A parametric analysis was performed in order to determine the 'best' model by comparing the distribution of the simulated air flow and *U* values inside the cavity with the wind tunnel. In every case the turbulence model used was a k-ε model (Jones and Launder, 1972). The following parameters were taken into account in the analysis: type of turbulence model: Standard or Realizable, type of wall function: Standard or Non Equilibrium, discretization schemes: 1st or 2nd-order-accurate schemes. Table 2 shows the 8 cases with their modelling characteristics.

From the initial 8, cases 1, 3, 5 and 8 were selected because of their better agreement with the wind tunnel results. The highest correlation values with the wind tunnel experiment concerning the U value corresponds to Case 3 (k-ε Standard, Non Equilibrium wall functions, 1st-order-accurate schemes) having R^2 from 0.983 to 0.997. Case 5 and 8 show the worst agreement with the experiment even if the differences are not large (R^2 from 0.976 to 0.996).

Table 2: Sequence of cases according to the model types. All cases have exponent α= 0.18 and k= 1.5 u*²

Cases	Type of turbulence model	Type of wall function	Discretization schemes
1	Standard	Standard	1st Order
2			2nd Order
3		Non Equilibrium	1st Order
4			2nd Order
5	Realizable	Standard	1st Order
6			2nd Order
7		Non Equilibrium	1st Order
8			2nd Order

Figure 3 shows the comparison of cases 1 and 3. The reference wind speed U_{ref} was taken here as the horizontal wind velocity component at the top centre point of the cavity (U_{ref} =2.48 m/s instead of 8m/s). In this way the difference of velocity between the predicted cases and the experiment in the 5 vertical lines are seen with respect to the maximum velocity inside the cavity. The maximum error between the prediction and the experiment concerning the air velocity inside de cavity is around U/U_{ref}=20% and 25% for the case of line 4 and 5 respectively. For the rest of lines, the error is in the range of 10-15%. The main differences in line 4 and 5 are in the central zone of the cavity where the increase of the backwards horizontal air speed is higher in the wind tunnel than in the predictions.

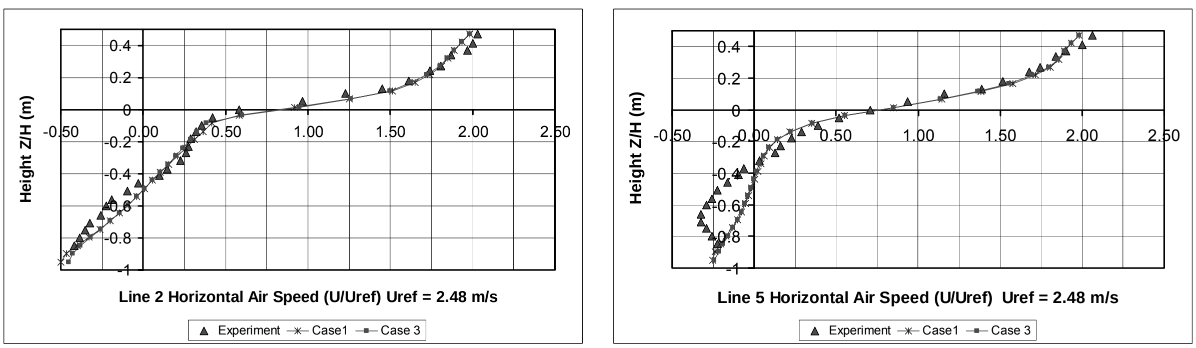

Figure 3 Comparison of horizontal air speed (U/Uref) between the cases 1 and 3 with the wind tunnel experiment. Uref=2.48m/s. On the left: Line 2, on the right: Line 5.

2.2.4 Different cavity ratios

As in the wind tunnel experiment the CFD simulations were also performed for the other four cavity ratios W/H=0.3, 0.5, 0.7, and 2.0. The agreement between the predictions of *U* inside the cavities with ratios W/H=0.5 and 2.0 was lower than in the case of the ratio equal to 1.0. A second simulation was performed in those cases using a finer mesh (case *c*). With the finer meshes better agreement was obtained for cavity ratio 0.5. The highest agreement corresponds, for every line, to the case 3 with R^2 from 0.976 to 0.983. In the cavity of ratio W/H=2.0 the best agreement correspond to case 3 for line 2, 4 and 5 with R^2 from 0.969 to 0.9745. The increase of air speed in the central zone of the cavity with ratio equal to 0.5 is less pronounced in the predictions and that can be the result of both the different position of the vortex inside the cavity and the possible error from the wall function equation, that underpredicts the values of air velocity near the walls of the cavity. This last error was also reported in the comparison of the wind experiment with the CFD predictions made by Kovar-Panskus et al (2002) using CHENSI software.

Figures 4 and 5 show the flow regime inside the cavity for the cavity ratios 1.0 and 0.5 both for the CFD predictions and the wind tunnel experiment. For the cavity rations W/H= 0.7, 0.5, 0.3, the centre of the vortex was lower in the CFD calculation than in the tunnel experiment. Besides, even with finer meshes, the recirculation areas which appear in the experiment near the ground next to the downstream wall, in the case of ratio 0.7 and 2.0, and near the ground, for the ratios 0.5 and 0.3 is not found in the CFD prediction.

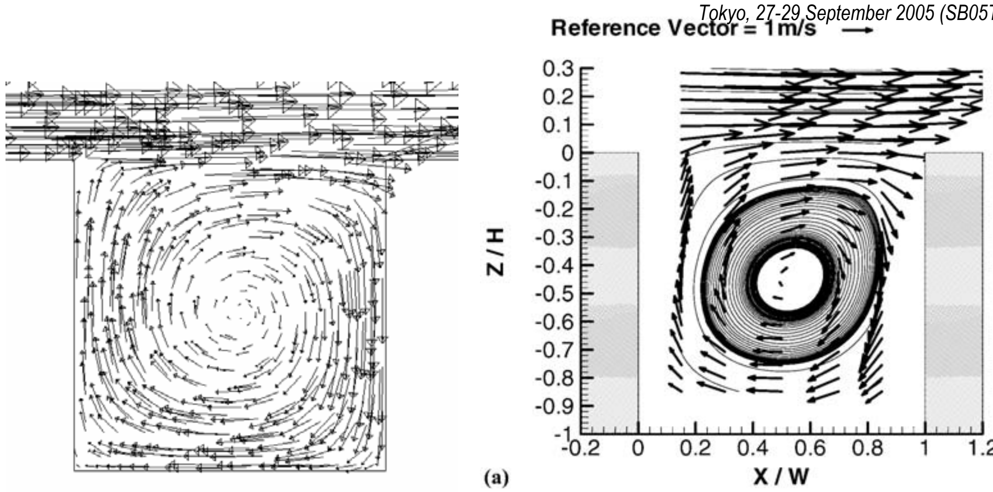

Figure 4 On the left: Fluent simulation, air flow inside the cavity with ratio W/H= 1.0, on the right: Wind tunnel experiments, mean velocity vectors and streamtraces on centre-plane for W/H=1.0, A. Kovar-Panskus et al. (2002).

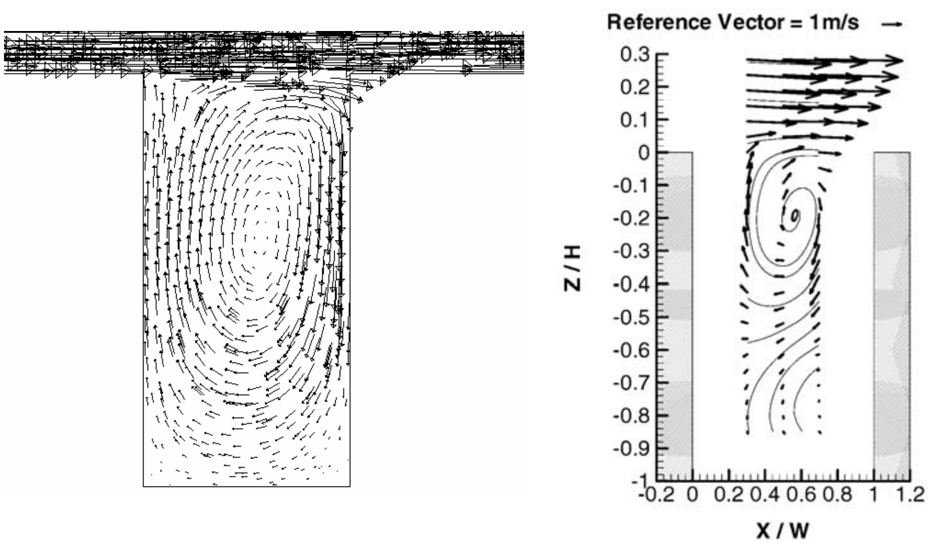

Figure 5 On the left: Fluent simulation, air flow inside the cavity with ratio W/H=0.5, on the right: Wind tunnel experiments, mean velocity vectors and streamtraces on centre-plane for W/H=0.5, A. Kovar-Panskus et al. (2002).

3. Potential for natural ventilation in courtyards with different geometries

3.1 Scale x 100

In order to verify the validity of the future simulations that will use a bigger scale, a series of simulations for the same cavity ratios were performed with a scale multiplied by 100. Thus, the cavity of 106 mm height became in a cavity with a size of 10.6 m, which is close to the actual size of a courtyard.

All the dimensions of the working section were multiplied by 100 as well as the meshes sizes and the values of Z_{ref} and z0. The values of U_{ref} and u* remain 8 m/s and 0.41 m/s respectively. Then, the values of the wind profile parameters are: α=0.18; U_{ref}=8 m/s; Z_{ref}=73.6 m; z0=0.03 m; u*=0.41 m/s; k=1.5 u^{*2}. Case 3 (Standard, non equilibrium wall functions, 1st-order-accurate schemes) and case 8 (Realizable, non equilibrium wall functions, 2nd-order-accurate schemes) were used for the simulations with a bigger scale.

Figure 6 shows a comparison of the air velocity inside the cavity between the scaled cases with the original case 3 and 8 with cavity ratio W/H=1.0. The results show a good agreement between the velocity values of the original scale and the new scale. It was found that case 3 and case 3 scaled had very close correlation.

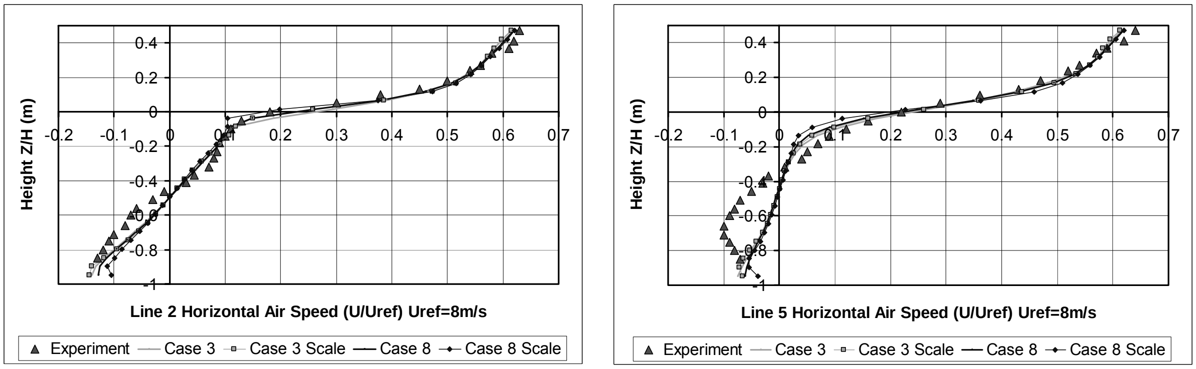

Figure 6 Comparison of horizontal air speed (U/U_{ref}) between the cases 3 and 8 with the wind tunnel experiment. On the left: Line 2, on the right: Line 5.

3.2 Potential for natural ventilation on 5 cavity ratios

Based on the validation in section 2, CFD is used to compare different types of courtyards with respect to the potential for natural ventilation. The data obtained from the simulations are going to be used as input parameters for thermal simulations (Energy-Plus, 1996) in order to assess thermal comfort inside dwellings. Pressure coefficient (Cp) on the courtyard walls and relative velocity magnitude (V/U_{ref}) inside courtyards are the parameters used to evaluate the potential of natural ventilation. Figure 7 shows the Cp values both for the downstream and upstream wall of the courtyard for every cavity ratio. Figure 8 illustrates the velocity magnitudes on line 3 (central line of the courtyard) and line 5 (closest to upstream wall) for every cavity ratio.

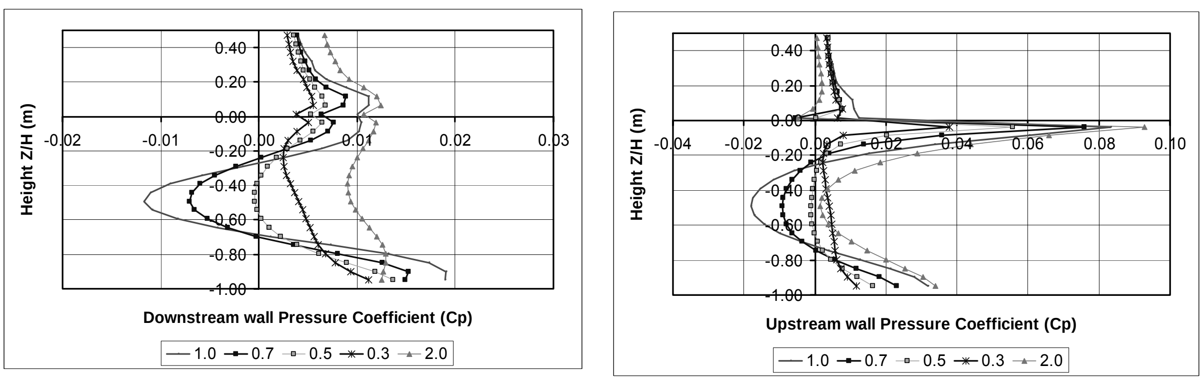

Figure 7 Pressure coefficients (Cp) of downstream (left graph) and upstream (right graph) courtyard walls for ratios W/H=1.0, 0.7, 0.5, 0.3 and 2.0.

According to the Cp values there is a poor potential for natural ventilation inside courtyards. The highest positive values are on the upper corner of the upstream wall (from 0.04 on ratio 0.3 to almost 0.1 on ratio 2.0). On the rest of the upstream wall and in the downstream wall the Cp values are very close to zero what indicates that in case of open windows the ventilation could be very poor by natural forces. The cases with cavity ratios 1.0 and 2.0 have the highest positive and negative Cp values along the whole walls. Cases with ratio 0.5 and 0.3 have the closest values to zero what means less potential for natural ventilation. The low Cp values can be explained for the skimming character of the flow inside the cavity (Oke 1988).

Concerning the velocity magnitudes, the extreme lines 1 and 5 have the highest values for every cavity ratio and line 3 has the lowest velocity magnitudes in the central part of the cavity coinciding with the centre of the vortex. Case with ratio 1.0 has the highest values in every line of the cavity while cases with ratios 0.3 and 0.5 have the lowest values. Case with ratio 2.0 has lower velocity magnitudes for line 1 than cases 1.0 and 0.7. When the cavity ratio is close to 1.0 the vortex can develops sufficiently creating high air speeds. On the contrary, when walls are positioned far away, the strength of the vortex speed decreases near the downstream wall. In the case that walls are positioned too close, the vortex will only develops in the upper part leading to a decrease of the air speed. Therefore, even if the Cp values are very low, the higher velocity magnitudes in courtyards with ratios 1.0 and 0.7 bring higher potential for natural ventilation if wind catcher systems or window special design are provided. However, further simulations should be performed when windows are open since the air flow between the courtyard and the indoor space might change considerably the vortex conditions.

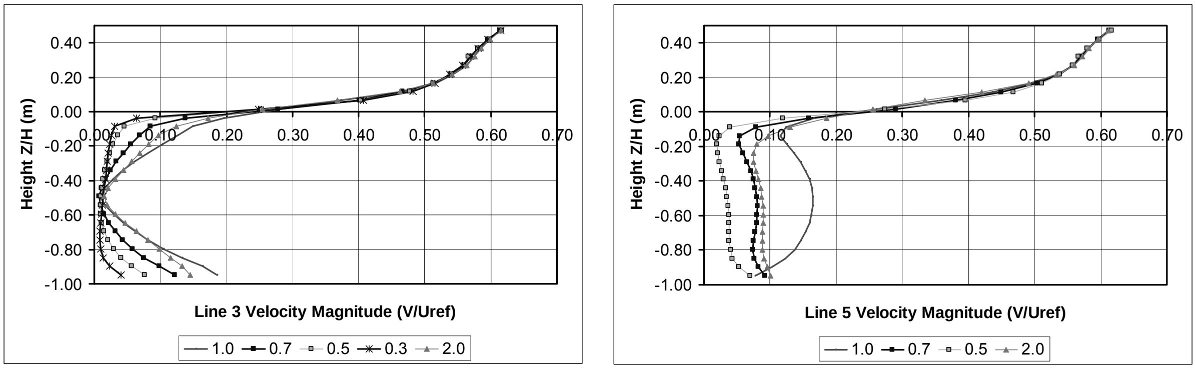

Figure 8 Comparison of velocity magnitude (V/U_{ref}) between different cavity ratios for line 3 (left graph) and line 5 (right graph).

3.2 Potential of natural ventilation on different courtyard geometries with obstructions

The final stage of this work was a comparison of five cavities with a different surrounding geometry in terms of relative velocity magnitudes (V/U_{ref}). The surrounding geometry consists of a block of 3m high by 10m long that is situated at one or at both sides of the cavity's top corner. These obstructions reproduce common elements above actual courtyards in Old Havana. Figure 9 illustrates three of the five courtyards geometries. Case I has no obstruction (same geometry as case with cavity ratio W/H=1.0), case II and III have 1 obstruction at one of the sides, case IV has obstructions at both sides of the cavity, and case V has also obstructions at both sides but having a cavity depth of 7m from the ground level or 10m from the upper part of the obstacle. The values of the wind profile parameters are: α=0.18; U_{ref}=8 m/s; Z_{ref}=73.6 m; z0=0.03 m; u*=0.41 m/s; k=1.5. The model used for the simulation was equal to Case 3 (Standard, non equilibrium wall functions, 1st-order-accurate schemes).

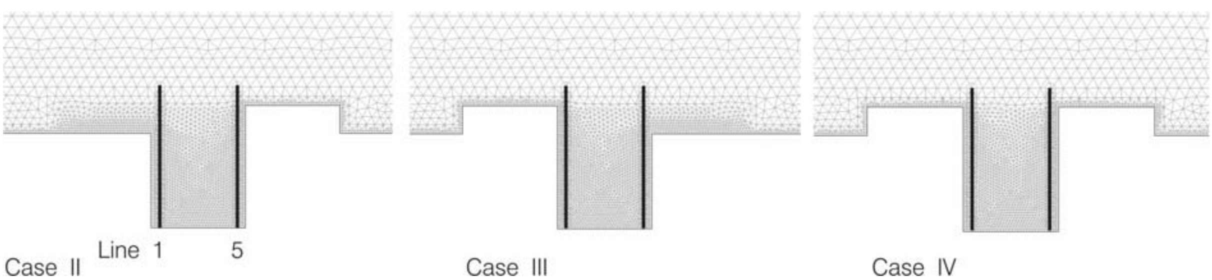

Figure 9 Position of the courtyards' obstructions and lines. Lines 1 and 5 coincide with the relative position of lines on the experiment. On the left: Case II, on the center: Case III, on the right: Case IV.

Figure 10 illustrates the air flow inside and outside the cavities of cases II and III. The relative velocity magnitudes of Line 1 and 5, which are less than 1 meter from the downstream and upstream wall respectively, are expressed in figure 11. For both lines, case I, which has no obstructions, has the higher V/U_{ref} (around 0.2). On the contrary, case III, which has an obstruction on the downstream side, has the lowest values. Case II has the second highest values thanks to the obstruction on the upstream side that acts as a wind catcher (see figure 10). However, this fact provokes lower velocities inside the cavity than in case I which has no obstruction.

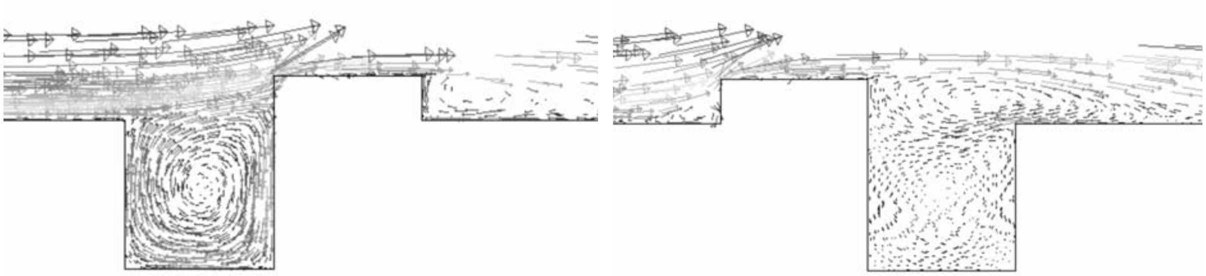

Figure 10 Air flow inside and outside cavities with different obstructions. On the left: Case II, on the right: Case III.

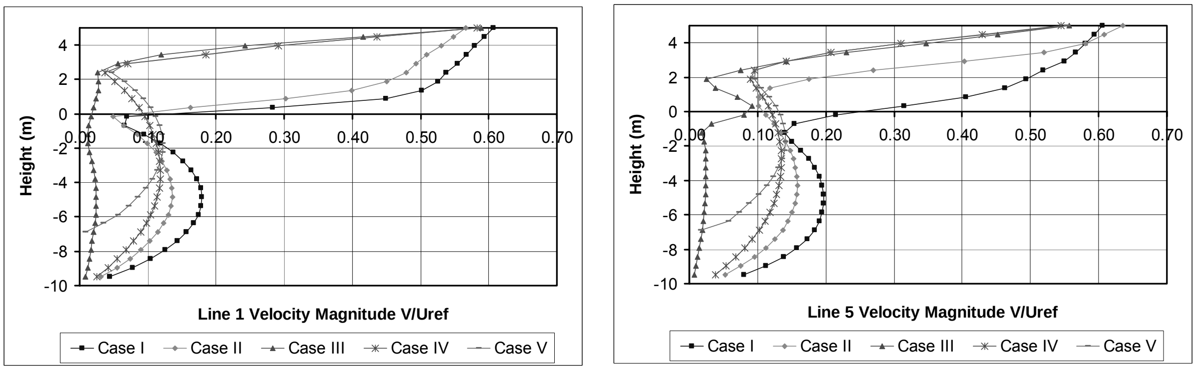

Figure 11 Comparison of velocity magnitude between cavities with different obstacles for line 1 (on the left) and line 5 (on the right).

4. Conclusions

A 2-D wind tunnel experiment and CFD simulations using Fluent software were compared in terms of horizontal air speed and air flow inside 5 different cavity ratios. The CFD simulations agree well with the experiment for cavity ratio 1.0. Nevertheless, the simulation underpredicts the air speed close to the cavity walls. For the rest of cavity ratios the CFD modelling does not simulate the secondary recirculation areas that appeared in the tunnel experiment, but the horizontal air speed was close to the experiment result.

The Cp values and the velocity magnitudes of the different courtyard ratios provide elements to assess the potential for natural ventilation inside dwellings facing the courtyard. Cavity ratio 1.0 and 0.7 have the highest potential for natural ventilation due to their geometry that promotes the development of a strong vortex and high velocity magnitudes. The presence of obstructions on the courtyard's top corners provoke a weaker flow inside the cavity and therefore lower velocity magnitudes and potential for natural ventilation.

Further numerical simulations concerning air flow conditions inside different courtyard's geometries must be performed in order to confirm the results of this work and to obtain a more comprehensive understanding of the interaction of the wind with the irregular geometry of courtyards and roofs. Comparison between CFD simulations and air velocity inside actual courtyards from climatic measurements (Tablada et al, 2004) will also be carried out. Besides, air flow simulations in case of more than one courtyard coupled to thermal building simulation models should also be performed in view of designing courtyard buildings with higher potential for natural ventilation, thermal comfort and healthy conditions for the particular situation of the compact urban environment of Old Havana.

References

Alvarez, S., Sanchez F. and Molina J. L., 1998, Air flow pattern at courtyards. Proceedings of PLEA 98, pp. 503–506, Lisbon.

Blocken B, Carmeliet J. 2004. Pedestrian wind environment around buildings: Literature review and practical examples. Journal of Thermal Envelope and Building Science 28(2): 107-159.

Blocken B, Roels S, Carmeliet J. 2004. Modification of pedestrian wind comfort in the Silvertop Tower passages by an automatic control system. Journal of Wind Engineering and Industrial Aerodynamics 92(10): 849-873.

EnergyPlus, 1996-2004, US Department of energy, http://www.eere.energy.gov/buildings/energyplus/ .

Díaz Véliz, R., Díaz Machado, A., Molina Esquivel, E., Suárez Rodríguez, B., (2000), The quality o fair in Cuba, II Latin American Conference on air quality and health, Mexico City, Mexico. http://www.cepis.ops-oms.org/bvsci/E/fulltext/2encuent/cuba2.pdf

Fluent 6.1.22, Fluent Intercorporated, 1998-2005.

Kovar-Panskus, A., Louka, P., Sini, J.-F., Savory, E., Czech, M., Abdelqari, A., Mestayer, P. G., and Toy, N. 2002, 'Influence of geometry on the mean flow within urban street canyons – A comparison of wind tunnel, experiments and numerical simulations', Water, Air, and Soil Pollution: Focus 2: pp. 365–380.

Jones, W.P. and Launder, B.E., 1972, The prediction of laminarization with two equation model of turbulence. Int. J. Heat Mass Transfer 15, pp. 301–314

Oke, T. R., 1988, Street design and urban canopy layer climate, Energy and Buildings 11, pp. 103-113.

Tablada, A., Barceló, C., De Troyer, F., 2004, Micro-climatic Measurements in the Belen Area of Old Havana and three Courtyard Buildings: Comparison with the Meteorological Station Data, Proceedings of PLEA 2004, Eindhoven.