

Integration of the natural cross ventilation in the CFD software UrbaWind

Karim FAHSSIS^a, Guillaume DUPONT^a, Stéphane SANQUER^a, Pierre LEYRONNAS^a

^a Meteodyn, Nantes, France

Presenting Author: Karim.fahsis@meteodyn.com, Tel: +33 (0) 240 71 05 05

Corresponding Author: Stephane.sanquer@meteodyn.com, Tel: +33 (0) 240 71 05 05

Abstract

Nowadays, a lot of energy is spent for air-conditioning in the cities for offices and private-housing. A good knowledge of the urban climatology around the buildings could allow using the wind for natural ventilation. UrbaWind is an automatic computational fluid dynamics (CFD) code developed in 2008 by Meteodyn to model the wind in urban environment. A module was recently added to assess the buildings ventilation.

First UrbaWind integrates climatology according to the geographic location of the site. Giving the influence of the shape and the disposition of the buildings on the wind behaviors, UrbaWind solves the equations of fluid mechanics with a specific model which allows taking into account the urban environment effects such as vortexes, venturi or wise effects. Finally, the software is able to compute the wind flow inside each internal volume according to the openings of the buildings.

This paper presents this software that has been designed for energy engineers to optimize the use of energy inside a building. This is also an important tool for architects and project managers to make a building. The shape of the building as well as the orientation and the location of the openings can be designed with the awareness of the wind-induced natural ventilation.

Keywords: Cross natural ventilation, simplified macroscopic method, wind modeling, CFD, urban climatology

1. Introduction

Nowadays, a lot of energy is spent for heating, ventilating and air-conditioning (HVAC) in the cities for offices and private-housing. Natural ventilation driven by winds is used in buildings in hot and humid climate to replace the indoor air with fresh outdoor air without energy consumption. Moreover, flows generated by the cross ventilation into the building can increase the thermal comfort without changing the indoor air temperature.

The natural ventilation of a building is driven by the combined forces of wind and thermal buoyancy. The thermal buoyancy depends on the temperature differences between the indoor air to the outdoor air. The wind driven ventilation depends on the wind behaviors, the interactions with the building envelope and the openings or others air exchange devices as inlets or chimneys. An analytic model for combined effects was defined by lots of authors among others Hunt and Linden ([1]), Li and Delsante ([2]). In this paper, only the ventilation associated to the wind forces is considered.

For a simple volume with two openings, the cross wind flow rate was written by Aynsley et al. ([3]):

$$Q = \sqrt{\frac{Cp_1 - Cp_2}{\frac{1}{A_1^2 C_1^2} + \frac{1}{A_2^2 C_2^2}}} U_{WIND} \quad (1)$$

Where A_1 and A_2 are the area of the openings 1 and 2, Cd_1 and Cd_2 their discharge coefficients given as functions of the openings aerodynamic behavior, and U_{WIND} is the reference wind velocity.

The cross ventilation depends on the pressure on façades and on the wind magnitude. The pressure on the envelope of the building can be given with tables (Clarke. J et al. [4], Eurocode [5]) or with parametric models based on analysis of results from wind tunnel tests (Grosso [6]). These tables or correlations are potentially inapplicable when buildings are irregular and different from the simple shape or flanked by others in a complex urban area with canyons, corner and trees. In these configurations, the wind fluctuates strongly and the pressure coefficients and the wind-driven natural airflows could be evaluated with difficulty. Wind pressure coefficients for complex buildings could be deduced from wind tunnel tests with scale down models or with computational fluid dynamics (CFD) methods.

Equation (1) can be used for a single ventilated volume with two openings. As soon as the number of openings exceeds two or the internal volume is divided in few sub volumes, the analytic method cannot be used. A resolution for non linear system as the nodal approach has to be used. The unknown quantities are the pressure in each sub volume. The main hypothesis for this simplified macroscopic approach is to consider the pressure constant in the volume and the indoor velocity negligible compared to the flow velocity inside the openings. That leads to make restriction of the method according the porosity of the building walls. Serfert et al. ([7]) and Karava et al. ([8]) show the formation of a stream tube inside the volume as soon as the openings areas are large enough and the openings are located in such a way that they are aligned. Hence, the internal pressure cannot be considered constant. The maximum porosity is between 10% to 33% depending on the formation of a streamtube. As soon as the porosity exceeds the bound, the use of a more sophisticate method as CFD or zonal modeling is needed. Nevertheless, for engineering purposes, wind pressure coefficients for sealed buildings and flows relations based on approximate applications of the Bernoulli equation are commonly used for predicting wind-driven cross ventilation in buildings with openings.

The air exchange between outdoor and indoor depends on the openings discharge coefficients. According Chiu and Etheridge ([9]), the discharge coefficient of the inlet C depends on the tangential flow around the throat.

$$C = f\left(\frac{V_{//}}{u}\right) \quad (2)$$

Where u is the normal velocity throw the opening, $V_{//}$ the tangential velocity parallel to the wall. f depends only on the shape of the opening. On the other hand, the outlet discharge coefficient is not influenced by the wind. For a cross flow velocity ratio $V_{//}/u$ in the range [0-10] the coefficient C decreases linearly with $(V_{//}/u)^2$ from 0.7 to 0.3 for a simple sharp edge orifice and for steady flow ([9]). Endo et al. ([10]) developed a new methodology named Local Dynamic Similarity Model taking into account the tangential flow in the calculation of the flow crossing the inlet openings. The difficulty remains to evaluate correctly the component of the flow parallel with the façade especially when its porosity for large opening is high enough to modify the wall pressure. In this case, CFD approach coupled with the local dynamic model gives corrects results. In our development, the opening area is less than 10%

of the façade area and the flows in urban area are strongly turbulent and the flows parallel with the façade are less sheared than in the previous study. So we expected the ratio $V_{//}/u$ to be low in order to consider the classical discharge coefficients in the first development of our macroscopic approach.

The knowledge of the urban climatology i.e. the wind around the buildings is crucial in order to evaluate the air quality and thermal behavior inside the buildings as soon as air and heat exchange depends on the wind pressure on façades. As we can see in the equation (1), the air exchange depends linearly on the wind speed of the urban place where the architectural project will be built. CFD tools and zonal modelings use pressure as input but no link has been made yet in software with the real urban climatology. The purpose of this work is to provide designers with a software devoted to calculate the wind on the buildings (CFD tools), to compute with a macroscopic method the mass flow rate incoming to the building for each wind characteristic (incidence and velocity magnitude), and finally to give cross ventilation statistics according to the local climatology. UrbaWind is an automatic computational fluid dynamics (CFD) code developed in 2008 by the company Meteodyn to model the wind in urban environment. A module is added to simulate the flow entering into the buildings in order to quantify the natural cross ventilation.

2. Description of the CFD code URBAWIND

2.1. Mathematical modeling

UrbaWind solves the equations of Fluid Mechanics, i.e. the averaged equations of mass and momentum conservations (Navier-Stokes equations). When the flow is steady and the fluid incompressible, those equations become:

$$\frac{\partial \rho \bar{u}_i}{\partial x_i} = 0 \quad (3)$$

$$-\frac{\partial(\rho \bar{u}_j \bar{u}_i)}{\partial x_j} - \frac{\partial \bar{P}}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \overline{\rho u'_i u'_j} \right] + F_i = 0 \quad (4)$$

The turbulent fluxes are parameterized by using the so-called turbulent viscosity. This viscosity is considered as the product of a length scale by a speed scale which are both characteristic lengths of the turbulent fluctuations.

The turbulent viscosity μ is considered as the product of a length scale by a speed scale which are both characteristic lengths of the turbulent fluctuations. The speed scale is given as the square root of the turbulent kinetic energy multiplied by the density. The turbulent kinetic energy is solved using the transport equation by including the production and dissipation terms of the turbulence where the turbulent length scale L_T varies linearly with the distance at the nearest wall (ground and buildings).

Boundary conditions are automatically generated. The vertical profile of the mean wind speed at the computation domain inlet is given by the logarithmic law in the surface layer, and by the Ekman function [11]. A ‘Blasius’-type ground law is implemented to model frictions (velocity components and turbulent kinetic energy) at the surfaces (ground and buildings).

The effect of porous obstacles is modeled by introducing a sink term in the cells lying inside the obstacle:

$$\vec{F} = -\rho \cdot V \cdot C_d \cdot |\vec{U}| \cdot \vec{U} \quad (5)$$

2.2.Mesh generation

This meshing is generated by following the computed wind direction. The size of the computation domain is automatically generated so that the minimum distance between a building and a boundary condition is equal to six times the height of the highest building of the simulation. The generated meshing is Cartesian and unstructured (using of overlapped meshing), with automatic refinement near the ground, obstacles and of course at result points locations. Usually the vertical mesh dimension around the building has been taken equal to 20 cm.

2.3.MIGAL-UNS Solver

The MIGAL-UNS solver has been regularly used for some years now, and has already been fully validated on number of academic cases [12].

MIGAL is an iterative linear equations solver which updates simultaneously the wind speed components as well as the pressure on the whole computational domain, so-called “coupled resolution”. This method demands more storage capacity, but it has the advantage to dramatically increasing the convergence process.

The discretized equations linear system, is transformed via an incomplete LU decomposition ILU(0) [13]. A preconditioner of type GMRES is used in order to increase the terms in the main diagonal so that the solving robustness is improved. Moreover, MIGAL uses a multi-grid procedure which consists in solving successively the equations on different grid levels (from the finer one to the coarser one). This method accelerates the convergence of the low frequency errors, which are known to be the limiting factor in the convergence process. For the type of problem solved here, different tests have shown a better convergence for the “V cycles” method than for the “W cycles” method. The number of grid levels is automatically computed by MIGAL. For the multi-grid computation, MIGAL-UNS uses an agglomeration method which joins together and agglomerates control volumes near the fine grid. This process is executed recursively until having generated a whole sequence of coarse meshes. Once the grid hierarchy is defined, one still has to determine how the equations are created on the coarse meshes and how their solutions will be used in order to decrease the errors of long wavelengths on the finest grids. MIGAL employs a Galerkin’s projection method for generating the equations on the coarse grid. This technique, so-called "Additive Correction Multi-grid" consists in generating the equation of the coarse mesh as the sum of the equations of each corresponding fine cell. Once the solution is obtained on the coarse grid, it is introduced by correcting the values calculated previously on the fine grid with the calculated error.

3. Description of the natural ventilation module

3.1.The simplified macroscopic theory

The simplified macroscopic method’s hypotheses are the followings:

- The pressure is constant inside the volume
- The velocity in the volume is negligible compared to the velocities into the openings. It means that the flow incoming is fully dissipated in the volume before the extraction across the outlet openings. In our approach, the porosity of the wall is bound in general at 10% expected when the openings are far enough or in such geometrical configurations to avoid the formation of a streamtube of fresh air in the volume.
- The discharge coefficients are theses of the openings without strong tangential flows. The value are compiled in a tables for various type of openings (windows, louvers, air inlet), for various geometrical behaviors (length, height, openings angles).

The simplified macroscopic approach consists in calculating the flow rate Q_k through the N openings for all the volume of the building as follow:

$$- \text{ For fully turbulent incoming flows if } P_k > P_{\text{int}} \Rightarrow Q_k = A_k C_k \sqrt{\frac{2(P_k - P_{\text{int}})}{\rho_k}} \quad (6)$$

$$- \text{ For fully turbulent outgoing flows if } P_k < P_{\text{int}} \Rightarrow Q_k = -A_k C_k \sqrt{\frac{2(P_{\text{int}} - P_k)}{\rho_k}} \quad (7)$$

Where P_k and P_{int} are respectively the pressure outside the volume around the openings and the internal pressure, A_k and C_k are respectively the geometric area and the discharge coefficient of the openings k . ρ_k is the density of air (constant in our approach).

$$\text{For each volume the equilibrium is stated as } \sum_{k=1}^N \rho_k Q_k = 0 \quad (8)$$

3.2.The resolution

The internal pressure depends on the air exchange and the external pressure on the building envelopes. So the mass flow rate depends on the internal pressure too. Network modeling is based on the concept that each zone can be associated to a pressure node. For multi volumes buildings, each room is represented by a node connected by flow paths such doors, windows and forms a network. The unknowns are the pressure nodes and the known values are the pressure outside the buildings (see (6) and (7)). When multi-volume configuration is considered, the resolution of the non linear system is based on the Newton-Raphson iterative method ([14]). For single node approach as described here, the only unknown is the internal pressure P_{int} . The equation solved is after mixing the equations (6) to (8):

$$F(P_{\text{int}}) = 0 \quad (9)$$

For one pressure node the resolution is the Newton iterative algorithm:

$$P_{\text{int}}^{n+1} = P_{\text{int}}^n - \omega \frac{F(P_{\text{int}})}{\left(\frac{\partial F}{\partial P_{\text{int}}} (P_{\text{int}}) \right)} \quad (10)$$

Where P_{int}^n is the internal pressure at iteration n and ω is the under-relaxation factor to enhance relaxation. In our application ω is between 0.5 and 0.7. The initial internal pressure P_{int}^0 , crucial with the Newton method to achieve quickly the convergence, is deduced from the external pressure and the openings behaviors as follows:

$$P_{\text{int}}^0 = \frac{\sum_{k=1}^N C_k^2 A_k^2 \times P_k}{\sum_{k=1}^N C_k^2 A_k^2} \quad (11)$$

For a classical building with a uniform repartition of openings on the walls, this formula gives a value of the internal pressure coefficient $C_{p_{\text{int}}}$ in the range [-0.1; -0.2] close to the well-known value ([6]). The iterative method is stopped when the residual flow rate is under an ε value of the whole flow rate.

4. Example of applications: a simple individual house

The building is a simple house (see figure 1) with an internal volume of 288 m^3 (length: 12 meters; width: 8 meters; height: 3 meters) and a sloping roof inclined at 30° .

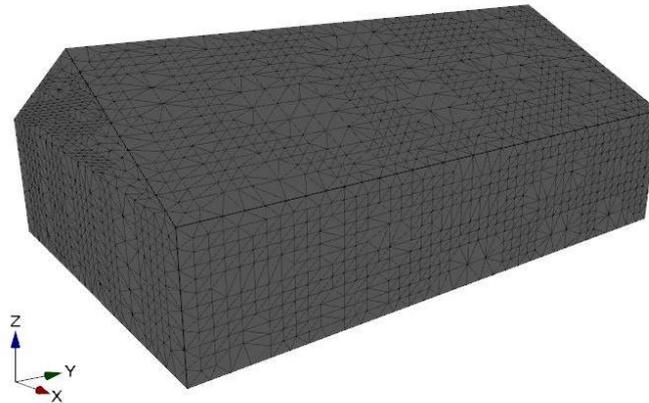


Figure 1: View of the simple individual house

The house is ventilated across the openings that are for several cases:

- Case n°1: 12 classical inlets located at 2 meters from the ground all over the vertical walls. The flow rate-pressure curve is $Q = 2.10^{-3} \times \Delta P^{0.6}$
- Case n°2: 2 full opened windows centered into two vertical walls connected. The flow rate-pressure curve is $Q = 0.84 \times \Delta P^{0.5}$ with $C = 0.65$ and $A = 1 \text{ m}^2$.
- Case n°3: 2 full opened windows centered into each largest vertical walls. The flow rate-pressure curve is $Q = 0.84 \times \Delta P^{0.5}$ with $C = 0.65$ and $A = 1 \text{ m}^2$.
- Case n°4: 2 full opened windows fitted in the same vertical wall. The flow rate-pressure curve is $Q = 0.84 \times \Delta P^{0.5}$ with $C = 0.65$ and $A = 1 \text{ m}^2$.
- Case n°5: 4 full opened windows centered into each vertical walls. The flow rate-pressure curve is $Q = 0.84 \times \Delta P^{0.5}$ with $C = 0.65$ and $A = 1 \text{ m}^2$.

The house is built in a suburb area. The roughness parameter useful to model the wind upstream the house is $z_0=50 \text{ mm}$. The dimensions of the computation domain are $108 \text{ m} \times 108 \text{ m}$ centered on the house. The minimal cell height normal to the ground and to the house walls are respectively 0.2 m and 0.1 m . The number of cells is between 160000 and 215000.

The simulation of the wind flow and the pressure on the envelope were carried out for each wind incidences from 0° to 330° with a step of 30° . For these simulations the reference mean velocity at 10 m was 10 m/s . The figure 2 shows an example of pressure field on the house envelope for a wind blowing normal to one of the largest façade (90°). The mean pressure coefficient on the middle of the upstream and downstream faces are respectively around 0.72 and -0.38 in accordance with the reference data for this building with a ratio length versus height of $\frac{1}{4}$ ([6]).

For each set of pressure, the flow rate was evaluated following the macroscopic method described in 3. The real ventilation can be estimated for the real urban climate around the house by taking into account the occurrence of every couple velocity/incidence. The user can set in his own meteorological data based for example in the data of the nearest meteorological station, corrected by the effects of the local topography. An example of climatic date is shown in figure 3.

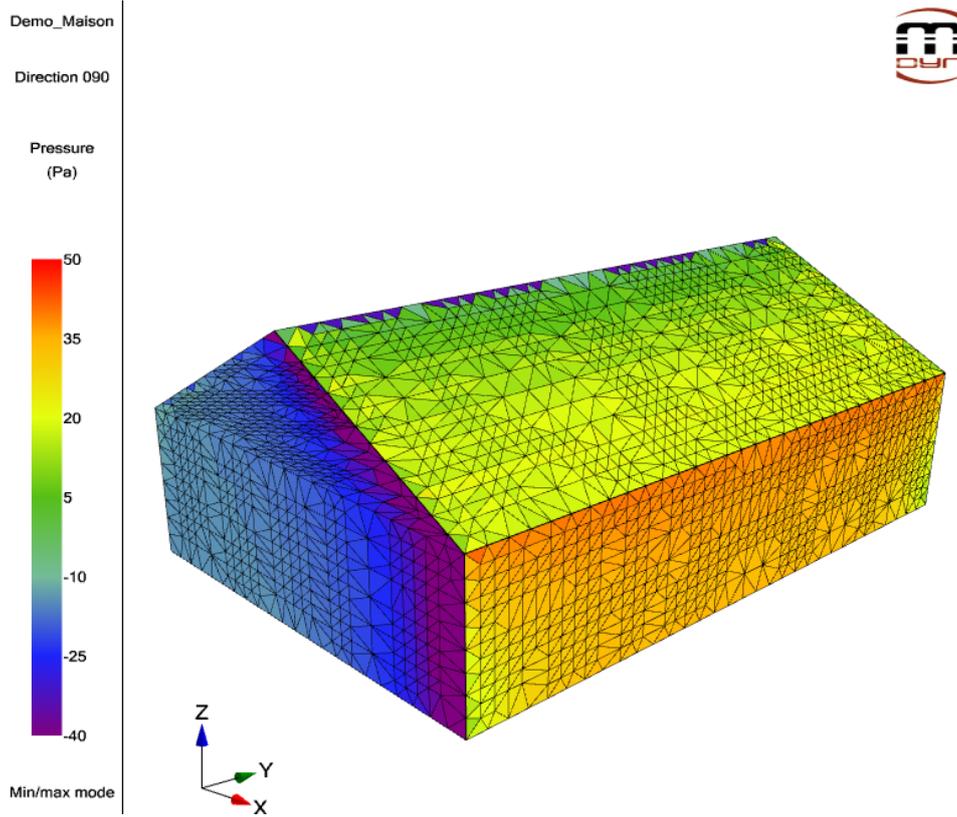


Figure 2: Mean pressure field on the house envelope for a wind normal to the largest wall $U_{wind}=10$ m/s – wind incidence 90°)

The flow rate computations were carried out for every wind occurrences according the urban climatology. The mean mass flow rate for every opening and the ventilation frequency statistics are given for several mean time steps (3 hours, 8 hours, day, night) according several periods (month, season and year) as soon as the climatology samples are long enough and time dependent.

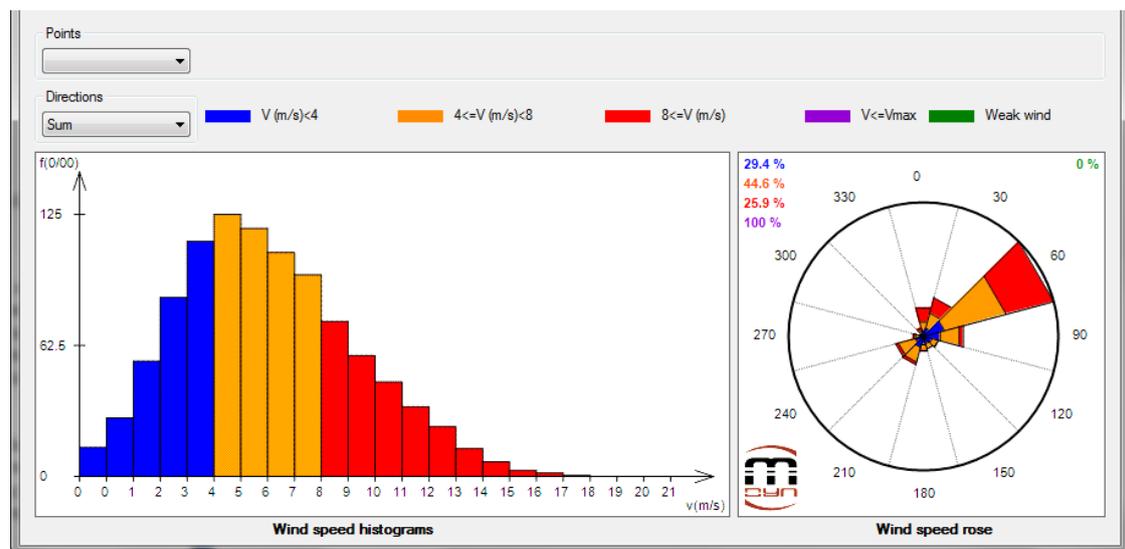


Figure 3: Wind statistics at the reference point - Wind speed histogram and rose

The figure 4 shows an example for two windows ($A=1 \text{ m}^2$, $C=0.65$) fitted at the middle of the largest façades (case 3). According the relation (1) with $C_{p1}=0.72$, $C_{p2}=-0.38$ and a mean wind speed about 7 m/s (see figure 3), the mean flow rate $Q=3.4 \text{ m}^3/\text{h}$ and the mean air change rate ACH of the volume is 42 volumes per hour. These values are higher than the mean value based on the climatology where $Q=1.9 \text{ m}^3/\text{h}$ and $ACH=24$ volumes per hour. The discrepancy about 50% is due to deviation about 30° of the main wind direction in this place (60°) according to the two openings axis (90°). In this case the pressure coefficients on the wall are less favorable for the cross ventilation. For the case 5 with 4 openings, ACH is 50 vol/h roughly twice the case 2's one with 2 openings.

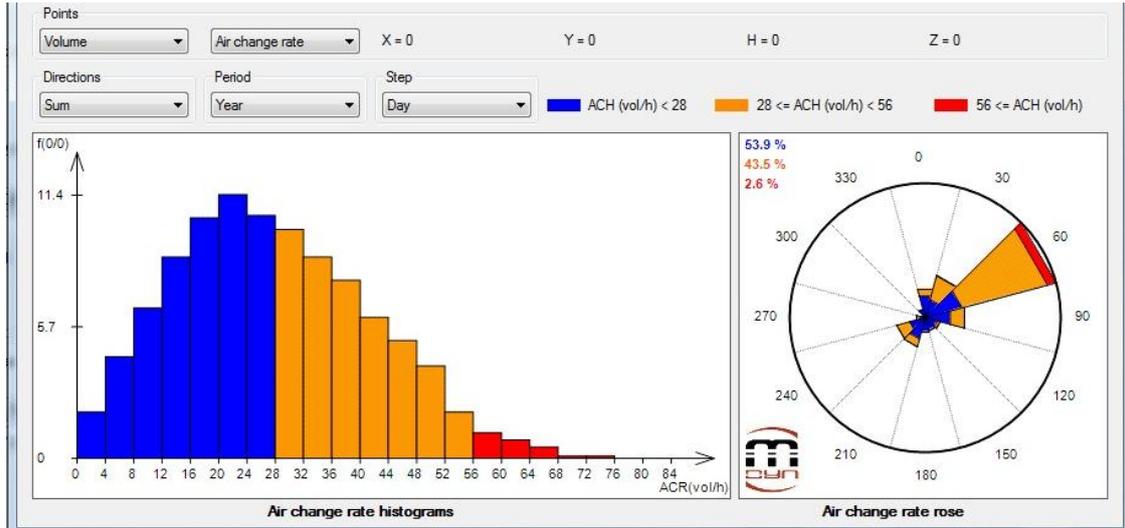


Figure 4: Statistics of the natural ventilation of the house - Histograms and roses

The air change rate of the volume ACH depends on the windows or inlet area but also on the position of the openings on the house walls. We introduce a global ventilation parameter GVP depending on the buildings shape and its openings in order to help the designer to distinguish the good configurations from the less efficient configurations.

$$GVP = \frac{ACH \times V_{house}}{\sum_k A_k C_k \times U_{WIND}} \quad (12)$$

Flow rate computations were carried out for the four configurations with a constant wind speed U_{WIND} of 2 m/s and wind incidence occurrences equal for every direction. Results are collected in the following table.

	openings type	configurations	ACH Vol/h	ACH Vol/s	V house (m ³)	Uwind (m/s)	$\Sigma A C$ (m ²)	GVP
Case 1	8 air Inlets	all around	0.12	3.3E-05	288	2	0.0144	0.33
Case 2	2 windows	corner	7.2	0.0020	288	2	1.3	0.22
Case 3	2 windows	cross	8.9	0.0025	288	2	1.3	0.27
Case 4	2 windows	same face	5.2	0.0015	288	2	1.3	0.16
Case 5	4 windows	All the faces	17.9	0.0050	288	2	2.6	0.28

The analysis of the results shows us:

- The global ventilation parameter is almost constant within 20% when at least two openings or inlets are on different faces of the building (GVP=0.16 to 0.22).
- The cross configuration (case 3) is the more efficient case but just 10% more than the corner configuration (case 2)
- The “only face” configuration (case 4) is the least efficient case about 40% compared to the full cross configuration (case 3)
- The air inlet configuration (case 1) is the less efficient configuration with air change rate about 0,1 vol/h. Nevertheless, this value has to be connected to a wind speed of 2 m/s. When the wind speed is not enough to allow a good air renewal, the designer can use the global parameter of the building in order to increase free area of the openings to reach the adequate fresh air rate.

5. Conclusion

Natural ventilation is less effective in urban environment and not easy to be designed because of the complexity of wind velocity behaviors. Hence air exchanges across the buildings openings are function of the wind speed in the urban area, the building shape and the openings efficiency. Designers have to validate the choice of openings (positions, dimensions and efficiency) in order to enhance cross natural ventilation and to allow a good air quality and comfort without energy consumption by replacing the air-conditioning systems. In this context, the software Urbawind was upgraded to introduce the natural cross ventilation. The interface allows the computation of the flow rate incoming and out-coming across every air inlets and windows. The software first computes the pressure fields on the building envelope for every wind incidence with the wind reference velocity, then evaluates air exchanges for each climatology event and finally builds statistical data of the air exchanges useful for designers.

Nomenclature:

Q	: flow rate (m ³ /s)
C_p	: pressure coefficient
U_{wind}	: reference wind at 10 meter above the ground (m/s)
P	: static pressure (Pa)
ΔP	: static pressure difference across the opening (Pa)
A	: geometric area of an opening (m ²)
C	: discharge coefficient
k	: opening index
n	: iteration index of the macroscopic method
ACH	: air change rate of the volume (volume per hour)
GVP	: global ventilation parameter
\bar{u}_i	: Components of the mean wind vector in a Cartesian frame (m/s)
u'_i	: Components of the turbulent fluctuations (m/s)
\bar{P}	: Mean pressure value (Pa)
μ	: Air dynamic viscosity (Pa.s)
ρ	: Air density (kg/m ³)
C_d	: Volume coefficient of frictions, which is proportional to the porous obstacle density
V	: Volume of the considered cell (m ³)
z_0	: Roughness length (m)

References:

- [1] Hunt GR, Linden PF, The fluid mechanics of natural ventilation – displacement ventilation by buoyancy-driven flows assisted by wind, *Building & Environment* 1999, 34: 707-720
- [2] Li Y, Delsante A, Natural ventilation induced by combined wind and thermal forces, *Building & Environment* 2001, 36: 59-71
- [3] Aynsley RM, Melbourne W, And Vickery BJ, *Architectural aerodynamics*, Applied science Publishers LONDON 1977
- [4] Clarke J, Hand J, Strachan P, A building and plant energy simulation system. Energy Simulation Resarch Unit, Department of Mechanical Engineering, Univeristy of Strathclyde. 1990
- [5] Eurocode 1 : Actions on structure – Part 1-4 : general actions – Wind Actions, EN 1991-1-4, CEN/TC 250
- [6] Gosso M, Wind pressure distribution around buildings : A Parametrical Model, *Energy and Buildings* 1992, 18: 101-131.
- [7] Serfert J, Li Y, Axley J, Rösler M, Calculation of wind driven cross ventilation in buildings with large openings, *Journal of Wind Engineering and Industrial Aerodynamics* 2006, 94: 925-947
- [8] Karava P, Stathopoulos T, Athienitis AK, Cross-ventilation building design: application of particle image velocimetry, 12nd ICWE 2007, 1407-1414
- [9] Chiu YH, Etheridge DW, External flow effects on the discharge coefficients of two types of ventilation opening, *Journal of Wind Engineering and Industrial Aerodynamics* 2007, 95: 225-252
- [10] Endo T et al., Development of a simulator for indoor airflow distribution in a cross-ventilated building using the Local Dynamic Similarity Model, *International Journal of Ventilation* 2006, 5(1): 31-42
- [11] Garratt JR, *The atmospheric boundary layer*, Cambridge Atmospheric and space sciences series. 1992
- [12] Ferry M, The MIGAL solver, Proc. Of the Phoenix Users Int. Conf., Luxembourg, 2000
- [13] Ferry M, New features of the MIGAL solver, Proc. Of the Phoenix Users Int. Conf., Moscow, Sept. 2002
- [14] Deru M, Burns P, Infiltration and Natural Ventilation Model for Whole-Building Energy Simulation of Residential Buildings, March 2003, Conference Paper, NREL/CP 550-33698,