Quantification of Ventilated Façade Effect Due to Convection in Buildings

Buoyancy and Wind Driven Effect

Miguel Mora-Pérez[1],Gonzalo López-Patiño[2], P.Amparo López-Jiménez[*3]

Hydraulic and Environmental Engineering Department. Universitat Politècnica de València
Camino de Vera s/n 46021 Valencia, Spain

[*1]miguelmorap@gmail.com; [2]glpatin@upv.es; [3]palopez@upv.es

Abstract

The external layer in a building has a paramount role under the building's energy behaviour point of view. The ventilated façade is a passive system installed on buildings to improve the global energy behaviour. The ventilated façade performance is described. Ventilated façade is mainly based on convection and radiation. This contribution focuses on the convective effect due to buoyancy driven respect to wind forces in the ventilated gap and their influence on the building thermal behaviour. To do so, several computational fluid dynamics models are undertaken with 1.5m/s, 0.5m/s and 0.2m/s wind velocities and a wind temperature of 298K and 300K. In the modelled conditions, the temperature of the external face of the façade was decreased due to the ventilated façade effect at 0.2m/s. 35% of the temperature reduction was due to natural convection buoyancy-driven effect and the rest due to wind-driven effect. With higher wind velocities (1.5m/s) the temperature reduction due to the convective effect was negligible. According to these simple trials, it can be concluded that the buoyancy driven forces have influence only below a certain threshold of wind velocity. The paper helps to better understand the behaviour of the ventilated façade installed in a building and allow designers to quantify the influence of the façade on the global building energy balance.

Keywords

Convection; Natural Ventilation (NV); Ventilated Façade (VF); Computational Fluid Dynamics (CFD)

Introduction

Government bodies, architects and engineers are becoming increasingly aware of the importance to reduce energy usage and CO_2 emissions, resulting in an increasing number of new solutions applied to improve the energy behaviour of the buildings. Thermal behaviour is one of the most important to be considered. The main responsibility of the heat exchange between the indoor and outdoor environment of a building is the building façade. It is responsible for the performance of several physical, dynamic and thermal parameters of the whole system (Kolokotroni 2012). Thermal comportment of the façade is crucial to understand the energy behaviour and to reduce the energy wastings of the buildings. Up to now, many techniques have been used to reduce this heat exchange through the façade. Initially, the façade thermal performance was improved by increasing thickness of insulating materials: as higher was the indoor-outdoor temperature difference; greater was the coating material thickness. Secondly, the quality of the coating material was improved with better isolation materials. Nowadays, much sophisticated techniques have been studied and already used such as ventilated façades (VF), which have many energy implications (Balocco 2002).

Particularly, a ventilated façade is a system used to improve the thermal behaviour of the buildings. This system allows designers to improve the energy behaviour in both, new building design and rehabilitation in the existing ones. The ventilated façade system is generally made up of an external cladding layer attached to the last continuous insulating external layer of the building by means of a mechanical structure (generally made of aluminium profiles). The cladding layer is made of glass, marble, ceramic, etc. and it defines the external appearance of the building. An air cavity at least 3 cm width is thus created between the insulation and the cladding layer. The mechanical structure must allow an upward a continued air flow in the façade. Thus, this air cavity is naturally ventilated as a result of solar radiation on the slabs and the ensuing convention within the cavity. There are two different ventilated façades depending on the way that the different slabs are installed. On one hand, the continuous VF (close joints) in which the upward flow is completely continuous, homogeneous and symmetrical along the wall (Patania 2010). On the other hand, the façades with open joints between each cladding, are commonly known as OJVF (Open Join Ventilated Façade) which turn the flow discontinuous, inhomogeneous and much more complex (Sanjuan 2011b).

Numerical models of air movement have been developped in VF and other building structures, such as solar chimneys or atriums by means of CFD techniques (Hussain and Oosthuizen, 2013). For instance, in the work of Seytier and Naraghy (2013), the radiative and convective effect in the buoyancy along a solar chimney was studied, successfully validating the computational fluid dynamics (CFD) uses for convection simulation. The influence of convection in buildings façades is a permanent topic of interest in research (Pasztory et al, 2011; Clark et al, 2013; Suarez, 2012; Wilmer and De Carli, 2012) always concluding the importance of the façade in the reduction of energy needs in the whole building, and the importance of an accuracy model for the deep knowledge of the façade performance.

Natural convection is one of the main principles for VF (Gratia 2004b). Heat transfer is classified into three main mechanisms: thermal conduction, radiation and convection. The fluid flow can be forced by external mechanisms or by buoyancy forces: natural convection. Natural ventilation (NV) is explained by two phenomena: wind driven ventilation and buoyancy-driven ventilation. Wind driven ventilation occurs due to the pressure difference in the façade surfaces produced by wind forces. Buoyancy-driven ventilation occurs as a result of the directional buoyancy forces that results from temperature differences between the interior and exterior (Linden 2009). This paper is focused on the thermal convection performance of an open join ventilated façade.

This effect has been partialli investigated. Mingotti (2011) described the fluid mechanics of the natural ventilation of a narrow-cavity double-skin façade. Increasing the height of the façade, the buoyancy effect could be improved leading to a faster flow. This faster flow will improve the energy performance of the VF. The research presented by Gratia (2004a) concluded that the ventilation of the façade is driven primarily by wind on the upper floors, where buoyancy heads are small, but by buoyancy on the lower floors, where buoyancy heads are large.

Different modelling approaches are used in predicting building ventilation including analytical, empirical, experimental and CFD models (Chen 2009). Recent studies have been done to improve the experimental data available in the literature: Lomas K.J. et al. studied a hybrid advanced naturally ventilated system in a new building (Lomas 2009), Giancola (2012) carried out an experimental assessment and modelling of the performance of an OJVF, Sanjuan (2011a) developed and experimental validation of a computational model for OJVF and González (2008) adjusted and energy computational model to analyze its energy performance in a building. In those, the fluid and thermal performance of OJVF has been investigated using both experimental and numerical methods. These studies confirm CFD as a reliable tool to model the fluid behaviour in this particular application. Furthermore, CFD simulation techniques enable designers to understand the behaviour of the systems and to predict whether it will work as expected or not. Thus the designer can optimize their constructive solutions in real scale models in an efficient computational way and not by expensive trial-and-error methodologies. The advantage of using these models resides in the fact that they can reproduce real problems of Fluid Mechanics to any degree of complexity.

In this contribution, the objective is to quantify the influence of the buoyancy-driven ventilation to increase the air velocity in the air due to the directional buoyancy force. This phenomenon is mathematically modelled by CFD techniques applied in a ventilated façade.

Methodology and General Objective

This research aims to quantify the convective flow effect due to buoyancy-driven forces in the air gap of an Open Join Ventilated Façade. The wind velocity and the temperature differences between the façade faces are the parameters that need to be defined, modified and then compared.

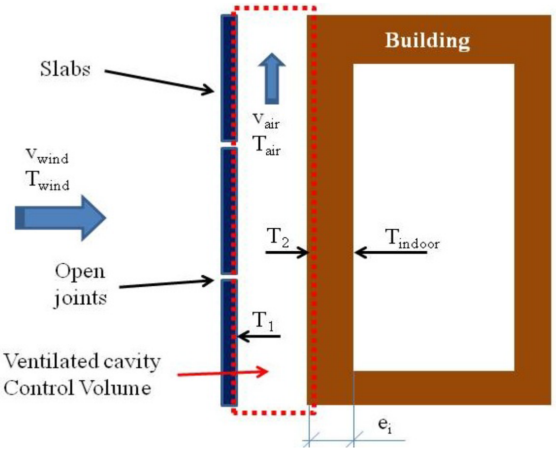


FIG. 1 CONTROL VOLUME AND TEMPERATURE DEFINITION

First stage on the CFD analysis method consisted of defining a simplified OJVF geometry in which the fluid phenomenon was studied. The model was composed of a geometry, boundary conditions and mesh definition. The geometry and the mesh definition remained constant in all simulations.

Figure 1 shows the control volume where the research was focused. It is composed by the indoor face of the cladding material, the ventilated gap and the external building's face. The volume was especially chosen to determine the temperature effect of the air flow through the cavity in the building external wall.

The CFD model simulated the air velocity in the gap taking into account the wind velocity and all the hydrodynamic effects: the narrow apertures of the OJVF, the friction forces, the wind driven flows, the buoyancy natural ventilation, etc. Wind velocity (v_{wind}), temperature (T_{wind}) and the temperature of the internal face of the cladding layer (T_1) were set at different values specially chosen to quantify the buoyancy-driven effect (BDE) in the ventilated gap. The external façade layer temperature T_2 was provided by final CFD simulation.

The first set of six simulations has been used as a reference (Non-BDE simulations) to analyze the thermal and fluid performance of the air in the ventilated façade. The set includes three pairs of simulations in which wind velocity is decreased: 1.5m/s, 0.5m/s and 0.2m/s respectively. Wind and the indoor cladding face temperatures were set at 298K and 300K in each pair of simulations; T_1=T_{wind}=298K and T_1=T_{wind}=300K respectively. Then, air velocity in the ventilated layer and temperature in the external façade (T_2) were calculated. These simulations were used as reference because there was no air movement due to the BDE because there wasn't any temperature gradient.

The second set of three simulations has been done to analyze the BDE in the air through the ventilated gap in comparison with the first set of simulations. The wind velocity was set at 1.5m/s, 0.5m/s and 0.2m/s in each simulation. Wind temperature was set at 298K and the indoor cladding face temperature was set at 300K in each simulation. Then, air velocity in the ventilated layer and temperature in the external façade (T_2) were calculated.

A third set of simulations has been carried out to notice how the increase of the temperature difference affects the BDE in the air through the ventilated gap with high and low wind velocities (1.5m/s and 0.2m/s respectively). The wind temperature was set at 298K and the indoor cladding face temperature was set at 302K, 304K and 306K for low velocity (0.2m/s) and at 306K for high velocity (1.5m/s). Then, air velocity in the ventilated layer was calculated. These data of velocity and temperature of wind correspond to measurements performed in Valencia (Spain) in June. The temperature gradients have been carefully chosen according to Mora-Pérez et al. (2011).

To simulate the wind velocity modulus, it is necessary to set and adapt wind data for height with the correct approach profile of mean wind-speed. In general, in a big vertical scale, the potential law for velocity modulus is accepted as vertical velocity profile (Justus and Mikhail, 1976). Nevertheless, in this particular case, in order to make a simplification and considering that the present research describes a very low structure compared with the atmospheric boundary layers, constant values for wind velocity have been considered to perform the numerical model.

The first sets of simulations (used as reference) were compared with the second set to analyze the influence of the wind on the natural ventilated flow in terms of convection. In the first simulations, the BDE didn`t exist because there wasn`t any temperature gradient. Moreover, the second set of simulations was set at 2K temperature gradient (T_1-T_{wind}). This temperature gradient produced a BDE that was quantified by comparison respect the case without it. Absolute results in each simulation have no sense in this research. Additionally, the air velocity and the temperature of the air in the ventilated gap and the building`s façade temperature (T_2) were analyzed.

Finally, the third set of simulations compared at lower and higher wind flows on how the temperature gradient affected the buoyancy-driven forces. The temperature gradient was set at 4K, 6K and 8K at low velocity (0.2m/s) and 8K at high velocity (1.5m/s). In addition, the air velocity in the ventilated gap was analyzed.

Modelling Strategy

CFD Solver Applied to Ventilation in the Façade Gap

The depicted methodology is an investigation with CFD and its application research in building systems. The literature is profuse in documents based on research applications of CFD, including experimental validations. Marioscini (2011) investigated experimentally and numerically the thermal comportment of a real OJVF in winter conditions. Wang (2007) modelled and validated the impacts of ventilation strategies and façades on indoor thermal environment for naturally ventilated residential buildings. Omar (2007) recommended CFD as a reliable method to study systems that have no access to laboratory or full-scale testing facilities. This reference found agreement between experimental and numerical approach although the good agreement was restricted in the majority of the cases to conditions of calm wind. The agreement with experimental results strongly supported the use of CFD for studying the fluid behaviour in a ventilated façade.

CFD allows designers to understand the fluid behaviour in different systems and to take better design decisions. Compared to other references like Giancola (2012) and Sanjuan (2011a), who contrasted the results of the CFD simulation with real experimental results; this contribution assumes that CFD simulations are right to represent the fluid behaviour. CFD is used as a design tool as Kang J.H. (2008) used the methodology to improve NV in a large factory building and Mora-Pérez et al. (2011) to quantify the efficiency of the ceramic ventilated façade.

Mathematical Model

CFD is a detailed modelling technique based on mathematical models which discretises the space in small cells where mass and momentum conservation equations are solved by the code. These equations are solved in a geometrical domain defined by boundary conditions and taking into account turbulent phenomena. The continuity or mass conservation equation solved by the software used is expression (1).

$$\frac{\partial \rho}{\partial t} + \nabla \rho \vec{v} = S_m \tag{1}$$

Where ρ is the fluid density, $\vec{v}$ is velocity and S_m represents the mass contained in the control volume. The momentum equation is considered in equation (2).

$$\frac{\partial(\rho\vec{v})}{\partial t} + \nabla\rho(\vec{v}\vec{v}) = -\nabla p + \nabla\overline{\tau} + \rho\vec{g} + \vec{F} \tag{2}$$

Where p is the static pressure, $\overline{\tau}$ the stress tensor defined in expression (3) and $\vec{g}$ and $\vec{F}$ the gravitational and outer forces respectively. μ is the eddy viscosity and I is the unit tensor.

$$\overline{\tau} = \mu\left[\left(\nabla\vec{v} + \nabla\vec{v}^T\right) - \frac{2}{3}\nabla\vec{v}I\right] \tag{3}$$

A general purpose CFD software package has been used: STAR-CCM+. The equations solved by STAR are discretized according to Finite Volume method (FVM). In FVM the solution domain is subdivided into a finite number of small cells called Control Volumes (CVs). Usually CVs are defined by a suitable grid and computational node is assigned to the CV center. All variations of FVM share the same discretization principals. They are different in relations among various locations within integration volume. The integral form of Navier Stokes equations are applied to each CV, as well as the solution domain as a whole. Summing all the equations for all CVs is obtained the global conservation equation since surface integrals over inner CV faces cancel out. The result is a set of linear algebraic equations with the total number of unknowns equal to the number of cells in the grid. Appliying this discretization to equation (2) the following discrete equation for velocity is obtained (CD Adapco 2011), where v_g is velocity grid, "f" refers to each face in the discretization method, and "a" is the face area vector.

$$\frac{d}{dt}(\rho v V)_0 + \sum_f \left[v\rho\left(v - v_g\right)\, a\right]_f = \sum_f \left(pI \cdot a\right)_f + \tau \cdot a + (\rho + + F)a_f \tag{4}$$

Simplifications

Several simplifications were asumed to reduce the computational time:

- The modelling is considered steady.
- No radiation effect is considered in the model, as the study is focused on the convective effect.
- The study is focused on the ventilated air gap between the internal sheet of the cladding layer and the external face of the building, where the convective effect takes place.
- The capacity of the cladding material to accumulate heat is not taken into account. The study is focused in the fluid region.

Modelled Geometry

The geometry modelled is a simplification of a OJVF installed in a building exposed to wind. The geometry consists of two columns with 14 slabs separated by horizontal and vertical joints 2 mm width.

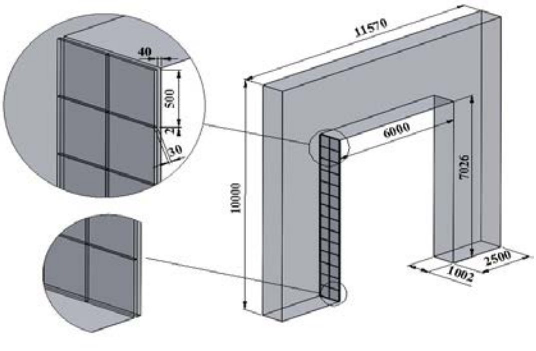


FIG. 2 BUILDING AND PANEL DIMENSIONS (mm)

Each column represents half piece of the cladding material which made the external ventilated layer. Each slab is 0.5 m. height, 0.03 m thickness and 1 m. width. The ventilated gap simulated is 40 mm. thickness and 1.002 m width. The ventilated façade is installed on a building in a wind tunnel. The building is 6 m depth. The wind tunnel is 10 m height, 1.002 m width and 11.57 m depth. Detail of the air gap and the dimensions of the building model are shown in Figure 2.

Boundary Conditions and Physical Description

The simulation has been performed under steady conditions in a 3D model, with constant density fluid flow. Turbulence effects have been included using the K-Epsilon model and segregated flow. The gravity model was used as it permitted the inclusion of the buoyancy source terms in the momentum equations when using the segregated flow model.

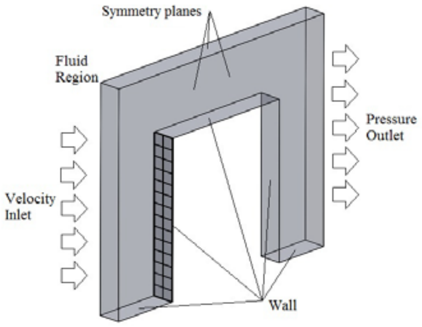


FIG. 3 BOUNDARY CONDITIONS FOR CFD MODEL

The whole domain was defined as fluid region (air) with boundaries. The boundaries settings were specially chosen to simulate the BDE in a building with an OJVF in a wind tunnel. The wind tunnel was defined by three symmetry planes (both laterals and the top face), a wall (bottom face), a velocity inlet in front façade and a pressure outlet at the end of the wind tunnel. The cladding and building's faces were defined as walls with a roughness of 2.5E-7 m. (Table 1)

TABLE 1 BOUNDARY CONDITIONS SPECIFICATIONS

Type	Surface (Wind tunnel)	Properties
Velocity Inlet	The front face	Constant velocity. 1.5m/s, 0.5m/s and 0.2m/s perpendicular to the building
Pressure outlet	The back face	Constant athmospheric pressure
Wall	All building and façade faces and the bottom face	Constant Roughness height = 2.5E-7 m.
Symmetry Plane	The upper and lateral faces	

CFD Mesh and Convergence

The simulation was done with a numerical method which is solved by the finite volume technique. The fluid model was solved by calculating the flow-equations on the nodes within the cells. The accuracy of the result depends on the definition of the nodes. A structured mesh was created with a refined grid near the joints. In addition, the mesh near the wall was set fine enough to allow the enhanced wall treatment to solve the near wall region all the way to the sub laminar region.

A mesh analysis was done to find out the optimum between the smallest number of nodes and the accuracy of the results. A grid-independence study, including the number of nodes and the size of the enlarged domain was performed in order to assess the validity of the numerical computational procedure.

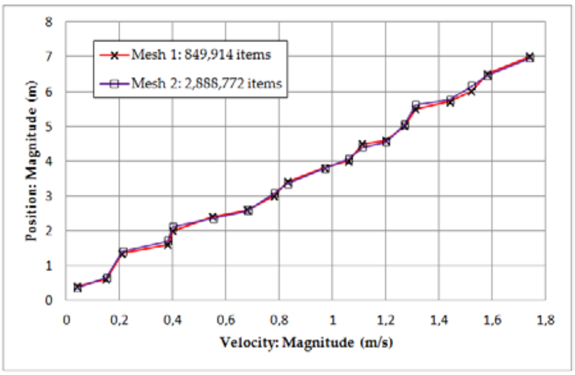


FIG. 4 VELOCITY COMPARISON BETWEEN MESHES

Two meshes were modelled with a computer Intel® Core™ i5 CPU M520 @2.4GHz. M1 with 849,914 items (M1: accuracy of 2 mm. and CPUTime 716.5 s) and M2 with 2,888,772 items; 1 mm accuracy and CPUTime 2226.16s). Velocity magnitude in a line in the ventilated façade was compared. Figure 4 shows that the velocity difference between models was less than 0.04%. Furthermore, the M1 model lasted 3 times more than the M2 model. Therefore the first mesh was selected. Figure 5.1 shows the detail of the mesh selected and Figure 5.2 shows the CFD residuals calculation.

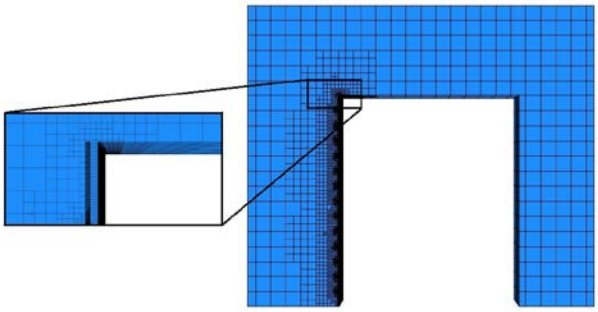


FIG. 5.1 DETAIL OF THE SELECTED MESH (M1)

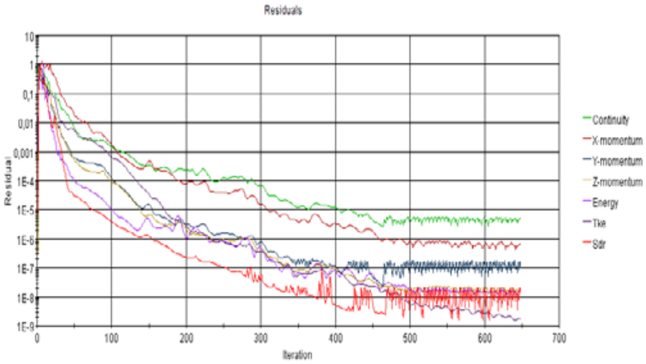


FIG. 5.2 RESIDUALS OF THE CFD CALCULUS FOR M1 MODEL

Wall treatment is necessary for model up proper boundary conditions. A numerical verification was made to check that the model was all right. In this case the high-y+ wall treatment used implies the wall-function-type approach, in which it is assumed that the near-wall cell lies within the logarithmic region of the boundary layer. A good rule is that the wall-cell centroid should be situated in the logarithmic region of the boundary layer (y+>30) (CD-Adapco). In this case, this requirement was accomplished for all walls.

CFD Post-processing

The last step in the CFD modelling strategy consists on obtaining the calculated parameters useful for the analysis. Post-processing allows designers to obtain XY plots (velocity, pressure, temperature, etc.), 2D and 3D vector representations, streamlines, numerical data, etc. In this case, the parameters used were air velocity in the ventilated air gap and the temperature of the external face of the building (T_2). The velocity of the air was determined in the centre of the ventilated gap in the middle of the panel. The temperature of the face was determined in the middle of the panel at the buildings face side.

Analysis of Results

Analysis on Air Gap Velocity

Several set of simulations were done to analyze the influence on the ventilated gap of both the buoyancy and the dynamic pressure effect due to wind. The dynamic pressure acting on the façade is generated when the kinetic energy of the wind is converted into pressure energy due to the braking of the wind on the fixed surface of the building.

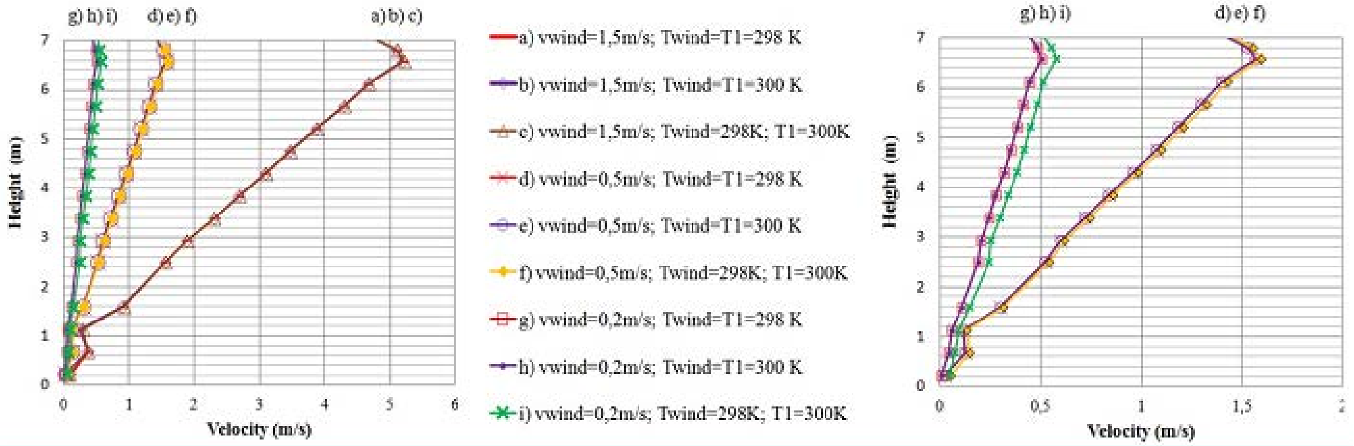
To analyze the relative weight of wind dynamic pressure over the buoyancy effect, different situations have been considered. On one side, wind speed has been changed, keeping the same temperature difference between the internal face of the cladding and wind. On the other side, for a low wind speed, in which the buoyancy effect is significant, it has been changed the temperature gradient between the cladding and wind.

Then, a first set of simulations have been performed without considering the buoyancy effect (Non-BDE simulations). The air into the gap should not be heated. To get this the wind (Twind) and the internal face of the cladding (T1) temperatures were the same. The air movement in the chamber was only due to wind dynamic pressure (Figure 6 lines a-b, d-e, g-h).

TABLE 1. TEMPERATURES AND VELOCITIES CONSIDERED

	Simulation								
Case	a	b	c	d	e	f	g	h	i
BDE	No	No	Yes	No	No	Yes	No	No	Yes
T_{wind} (ºK)	298	300	298	298	300	298	298	300	298
T_1(ºK)	298	300	300	298	300	300	298	300	300
v_{wind} (m/s)	1,5	1,5	1,5	0,5	0,5	0,5	0,2	0,2	0,2

Figure 6 shows the evolution of wind speed inside the gap (v_{air}) for the simulations listed in Table 1.



(a) (b)

FIG. 6 AIR VELOCITY IN THE VENTILATED GAP

It is noted that the higher the wind speed (v_{wind}), and thus the dynamic pressure on the façade, the higher the velocity of air inside of the gap (v_{air}). Furthermore, at medium and high wind speeds, the buoyancy effect is not appreciable compared with the dynamic pressure effect. By contrast, at low wind speeds (simulations g, h, i) there is a certain influence of the buoyancy effect with respect to the dynamic pressure effect, due to the small value of the latter. As the façade is opened in the lower part, the wind driven effect causes a greater effect in this range of wind velocities due to the great dynamic pressure generated by the wind in comparison with the buoyancy effect.

The second and third set of simulations were considered the bouyancy effect (BDE simulations). The internal face of the cladding temperature was greater than that of the wind, as there was a progressive transmission of heat to it. Air movement in the gap was due to dynamic pressure and bouyancy effect.

TABLE 2. TEMPERATURES AND VELOCITIES CONSIDERED

	Simulation						
Case	c	f	i	k	m	n	p
BDE considered	Yes	Yes	Yes	Yes	Yes	Yes	Yes
T_{wind} (ºK)	298	298	298	298	298	298	298
T_1(ºK)	300	300	300	302	304	306	306
v_{wind} (m/s)	1,5	0,5	0,2	0,2	0,2	0,2	1,5

The temperature gradient between the wind and the internal face of the cladding material (T_1-T_{wind}) was increased from 2K to 4K, 6K and 8K (third set of simulation) with a low wind velocity (0.2m/s).

Figure 7 shows the influence of the buoyancy effect when the wind dynamic pressure is reduced. The greater the temperature gradient between the air and the cladding is, the greater the buoyancy effect and the air velocity inside the gap are.

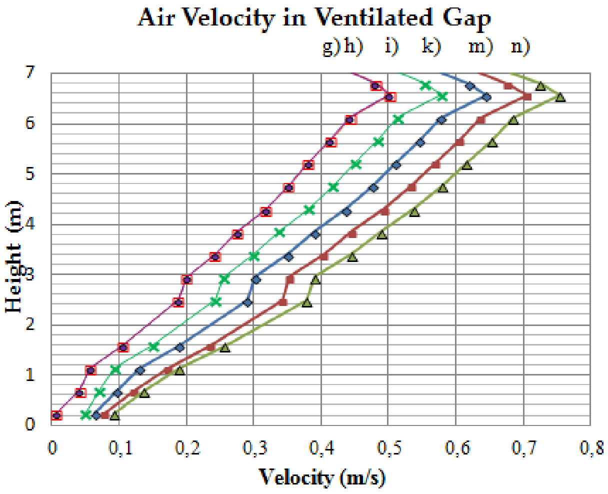


FIG. 7 AIR VELOCITY IN THE VENTILATED GAP AT 0.2m/s WIND VELOCITY.

Finally, to verify the reduced influence of the bouyancy effect at high dynamic pressure, a new simulation has been done (simulation p). It has been considered the major buoyancy effect (high gradient of temperature, T_{wind} = 298ºK, T_1 = 306°K) for the dynamic pressure corresponding to v_{wind} = 1.5m/s. In figure 8 the evolution of velocity in the gap together with the simulations a, b, c, (Table 1) is shown. There were no difference observed in the evolution of v_{air} velocities. Then, the air velocity increased as the BDE produced by increasing the temperature gradient at high velocities was negligible.

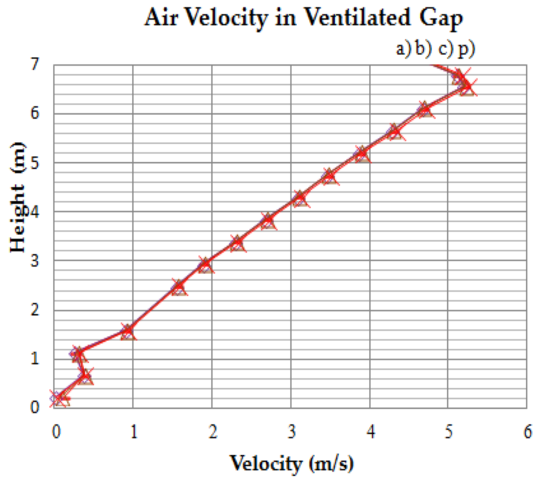


FIG. 8 AIR VELOCITY IN THE VENTILATED GAP AT 1.5m/s WIND VELOCITY

Analysis on Façade Temperature (T_2): Heat Flow to the Building

An analysis has been done to the temperature on the outer side of the façade (T_2), which was in contact with the gap, for the same cases listed in Table 1.

The objective is to verify how it will be affected the temperature T_2, and therefore the thermal comfort inside the building, by varying the dynamic pressure, due to the air velocity, and the buoyancy effect.

Figure 9 shows results from the different simulations. For those cases in which the BDE was excluded from consideration, the temperature of the façade was the same as the wind, the inner cladding and the air in the ventilation gap. For those cases in which BDE was considered, the air temperature inside the gap (T_{air}) was generally higher than the temperature on the façade (T_2). There was a mismatch in this behavior, due to edge effect, on the top of the gap.

By heat transfer, the greater the flow velocity of the fluid through the gap, the lower the temperature reached. This is the reason why the air temperature in the case c is lower than for the case i (Figure 9).

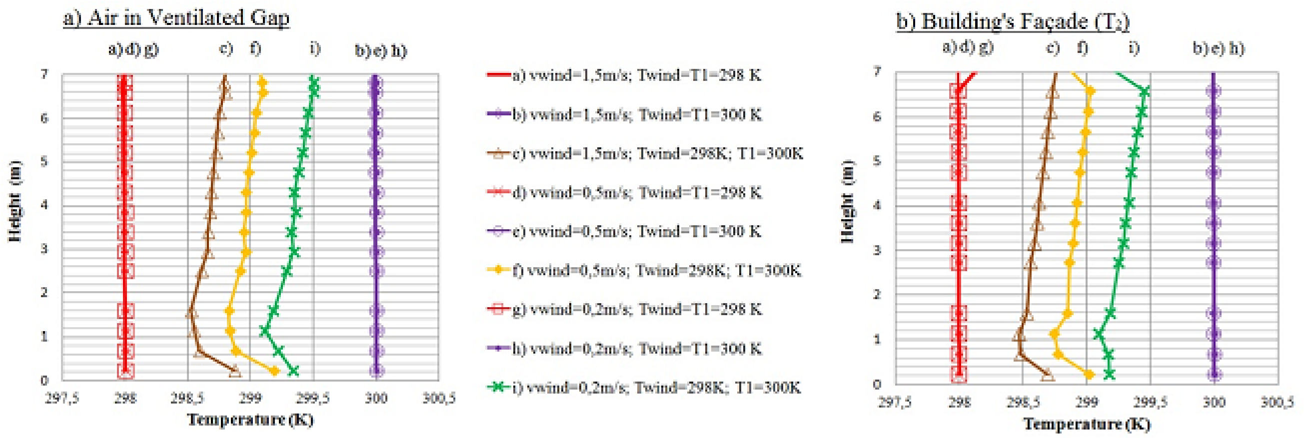


FIG. 9 TEMPERATURE PERFORMANCE

For a given wind velocity (fixed dynamic pressure) the air temperature along the façade increased due to the heat transmitted from the cladding. The temperature gradient along the façade for the case where the buoyancy effect was significant (case c) was of the same order of magnitude as for the case in which buoyancy effect as hardly seen (case i), although air temperature (T_{air}) was higher. This behaviour confirmed that the buoyancy effect was minor compared with the effect of the dynamic pressure.

From the buildings energy behaviour point of view, the main objective of the ventilated façade is to reduce the load gain of the building to decrease the cooling power required to keep a comfortable indoor environment in summer. The thermal performance of the ventilated façade relies on buoyancy: first of all, the slabs of the exterior coating are heated up. Then, the heat load is transferred through the cladding material to its interior face. Finally, it produces an ascending mass flow air in the ventilated cavity by natural convection. The thermal behaviour of the façade is different depending on whether it is summer or winter as it was explained by Giancola (2012) in an experimental case. In summer, the ventilated façade reduces the heat transfer to the indoor environment. Equation 4 shows that the total load gain q (W/m^2) in the building depends on the U-coefficient (W/m^2/K) and the temperature gradient of the inner and the outer face of the wall. Equation 5 shows how U-coefficient is calculated. It is considered the width of each material layer by which the external wall e_i (m.) is built, its heat-conductivity coefficient k_i (W/m/K) and the indoor and outdoor convective-coefficient h_i & h_e respectively (W/m^2/K).

$$q = U_{wall} \cdot (\Delta T) = U_{wall} \cdot (T_2 - T_{indoor}) \frac{W}{m^2} \tag{4}$$

$$U = \frac{1}{\left(\frac{1}{h_i} + \sum\left(\frac{e_i}{k_i}\right) + \frac{1}{h_e}\right)} \tag{5}$$

The present report quantifies the energy reduction due to the natural convection buoyancy-driven effect in the ventilated cavity. The U-value remained constant for all calculations and the unique change was the temperature drop. 35% of the ventilated façade energy savings were due to the buoyancy-driven effect. This result was calculated with the temperature drop in the building's façade (Figure 9, "h" line, T_2) with respect to the temperature of the inner face of the ventilated slab (Figure 9, "i" line, T_1) for the low wind velocity (v_{wind}=0.2m/s) case. In these particular conditions, the buoyancy driven effect supposed a reduction of 10 W/m^2 (out of a total of approximately 28W/m^2 reduction in cooling power due to the presence of the air in the ventilated façade of the building).

Conclusions

A ventilated façade is a powerful system used to improve the thermal behaviour of the buildings. Natural ventilation (NV)plays an important rol in the ventilated façade performance and can be explained by two phenomena: wind-driven ventilation and buoyancy-driven ventilation. While wind is the main mechanism of wind-driven ventilation, buoyancy-driven ventilation occurs as a result of the directional buoyancy force that results from temperature differences between interior and exterior walls. In this contribution, the natural convection thermal effect of an OJVF and its capacity for cooling in summer conditions has been quantified.

On the one hand, as wind velocity was increased over 0.5m/s, no air velocity changes in the ventilated gap due to the buoyancy-driven effect (BDE) could be observed. The velocity increase due to BDE was negligible with respect to the effect of the wind-driven forces. In the particular case of 1.5m/s wind velocity and 8K temperature gradient between wind and the indoor cladding face, the air velocity increase due to BDE was negligible. Consequently, the behaviour of the air velocity in the ventilated façade mostly depends on outdoor weather conditions. On the other hand, as wind velocity was decreased under 0.5 m/s, the air velocity was increased due to the BDE having a higher specific weight in the total air velocity change. Furthermore, in the operating conditions, at low wind velocities (v_{wind}=0.2m/s) as the temperature gradient determining the BDE was increased, the velocity was also proportionally increased. Therefore the BDE became more important in the behaviour of the ventilated façade in terms of air velocity.

In addition, the quantification of the temperature drop in the external face of the building had the most important effect on determining the thermal and energy performance of the building with a ventilated façade passive system. In the modelled conditions (2K temperature drop between the air and the internal cladding face temperature and v_{wind}=0.2m/s), the temperature of the external face of the façade was decreased due to the ventilated façade effect at low wind velocities. The 35% of the temperature decrease was due to natural convection buoyancy-driven effect; and 65% due to wind-driven effect.

In this study, the worst case for air movement in the ventilated gap was analysed: the ventilated façade was closed at his bottom. Further research should be done with a ventilated façade opened at its bottom part.

The main conclusions summarized as a points are the following ones:

- At high wind velocities (>1.5m/s) the buoyancy-driven effect is negligible with respect to the wind-driven effect in summer.
- At high wind velocities, the buoyancy-driven effect doesn't influence the ventilated façade cooling effect in summer.
- At low wind velocities (<0.5m/s) the buoyancy-driven effect has a higher specific weight in the total air velocity change in the façade.
- The more the air velocity in the ventilated façade is accelerated, the more the cooling effect is reached.
- At low wind velocities (<0.5m/s); the higher the temperature gradient is, the more cooling by the buoyancy-driven effect is achieved.

TABLE 3. NOMENCLATURE

Magnitude	Unit	Description
v_{wind}	m/s	Wind velocity
T_{wind}	K	Wind temperature
v_{air}	m/s	Air velocity in the ventilated gap
v_g	m/s	Velocity grid
T_{air}	K	Air temperature in the ventilated gap
T_2	K	External façade layer temperature
T_1	K	Internal face of the cladding layer temperature
T_{indoor}	K	Indoor temperature
e_i	mm	Width of the building
ρ	Kg/m^3	Fluid density
$\vec{v}$	m/s	Fluid velocity
S_m	m^2	Mass source in the control volume
t	s.	Time
p	Pa	Static pressure
$\bar{\tau}$	N/m^2	Stress tensor
g	m/s^2	Gravitational force
μ	Pa·s	Eddy viscosity
F	N	Outher force
q	W/m^2	Surface heat transfer
k	W/m/K	Conductivity coefficient
h	W/m^2/K	Convective coefficient
U	W/m^2/K	Overall heat transfer coefficient
I		Unit tensor

REFERENCES

Balocco, C. "A simple model to study ventilated façades energy performance". Energy and Buildings, Vol 34, Pp 469-47. 2002.

CD-Adapco. Star CCM+ User's Manual. 2011.

Chen Q. "Ventilation performance prediction for buildings: a method overview and recent applications". Building and Environment, Vol 44. Pp 848 – 858. 2009.

Clark, J.; Peeters, L.; Novoselac, A. 2013. Experimental study of convective heat transfer from windows with Venetian blinds. Building and Environment, Vol. 59, N 1, Pp 690-700.

Giancola, E. ; Sanjuan, C. ; Blanco, E. ; Heras, M.R. "Experimental assessment and modeling of the performance of an open joint ventilated façade during actual operating conditions in Mediterranean climate". Energy and Buildings, Vol 54. Pp 363 – 375. 2012.

Gratia E., De Herde, A. "Is day natural ventilation still possible in office buildings with a double-skin façade?" Building and Environment, Vol 39. Pp 399 – 409. 2004.

Gratia E, De Herde A. "Optimal operation of a south double-skin façade". Energy and Buildings, Vol 36. Pp 41 – 60. 2004.

González M. Blanco E. Pistono J. "Adjusting and energy simulation model by means of CFD techniques to analyze open-joint ventilated façade performance, in: WREC-X", Glasgow. Solar and Energy, Vol 85. Pp 1851 – 1863. 2008.

Hussain, S.;. Oosthuizen, P.H. 2013. Numerical investigations of buoyancy-driven natural ventilation in a simple three-storey atrium building and thermal comfort evaluation. J. of Applied Thermal Engineering. Vol. 57. Pp 133-146.

Jong-Hoon Kang, Sang-Joon Lee. "Improvement of natural ventilation in a large factory building using a louver ventilator". Building and Environment 43. Pp 2132 – 2141. 2008.

Justus CG, Mikhail A. 1976. Height variation of wind speed and wind distributions statistics. Geophysical. Research Letters; 3(5):261-4.

Kolokotroni, M., Robinson-Gayle S., Tanno S., Cripps A. "Environmental impact analysis for typical office façades". Building Research and Information. January-February. 2002.

Linden, P.F. "The Fluid Mechanics of Natural Ventilation". Annual Review of Fluid Mechanics 2009. Pp 201-208. 2009.

Lomas, K.J., Cook M.J., Short C.A. "Commissioning hybrid advanced naturally ventilated buildings: a US case study". Building Research and Information, Vol 37. Pp 397 – 412. 2009.

Marioscini, C.; Strachan, B.; Semprini, G.; Morini, G.L. "Empirical validation and modeling of a naturally venilated rainscreen façade". Energy and Buildings, Vol 43. Pp 853 – 863. 2011.

Mingotti, N., Chenvidyakarn T., Woods A.W. "The fluid mechanics of the natural ventilation of a narrow-cavity double-skin façade". Building and Environment, Vol 46. Pp 807 – 823. 2011.

Mora-Pérez, M.; López-Patiño, G.; Bengochea-Escribano, M.A.; López-Jiménez, A. "Quantification of the efficiency of the ceramic ventilated façade by means of computational fluid mechanics techniques". Boletín de la Sociedad Española de Cerámica y Vidrio, Vol 50. Pp 99-108. 2011.

Omar S. A., Mohamed B. G. "A comparison between CFD and Network models for predicting wind-driven ventilation in buildings". Building and Environment, Vol 42. Pp 4079–4085. 2007.

Pasztory, Z.; Peralta , P. Peszlen , I. Multi-layer heat insulation system for frame construction buildings. J. Energy and Buildings. Vol 43. N 2–3, February–March 2011, Pp 713–717.

Patania, F. ; Gagliano, A. ; Nocera, F. ; Ferlito, A. ; Galesi, A. "Thermofluid-dynamic analysis of ventilated façades". Energy and Buildings, Vol 42. Pp 1148 – 1155. 2010.

Sanjuan, C. ; Suárez, M.J.; Blanco, E.; Heras, M.R. "Development and experimental validation of a simulation model for open joint ventilated façades". Energy and Buildings, Vol 43. Pp 3446 – 3456. 2011.

Sanjuan, C. ; Suárez, M.J.; González, M. ; Pistono, J.; Blanco, E. "Energy performance of an open-joint ventilated façade compared with a conventional sealed cavity façade". Solar and Energy, Vol 85. Pp 1851 – 1863. 2011.

Seytier, C. and Naraghi, M. 2013. Combined Convective-Radiative Thermal Analysis of an Inclined Rooftop Solar Chimney. J. Sol. Energy Eng. 135(1).

Suárez,M.J. Sanjuan,C. Gutiérrez,A.J.; Pistono,J. Blanco,E. 2012 Energy evaluation of an horizontal open joint ventilated façade, Applied Thermal Engineering, Volume 37, May 2012, Pages 302-313.

Wang L., Wong N. H. "The impacts of ventilation strategies and façade on indoor thermal environment for naturally ventilated residential buildings in Singapore". Building and Environment, Vol 42. Pp 4006–4015. 2007.

Wilmer, P.; De Carli, M. Evaluation of various CFD modelling strategies in predicting airflow and temperature in a naturally ventilated double skin façade. 2012. Applied Thermal Engineering, May, 2012, Vol.37, Pp.267-275."

Miguel Mora was born in 1986 in Banyeres de Mariola (Spain). He earned MSc in Mechanical Engineering at Universitat Politécnica de València in 2010. He earned a Ms in Energy Technology for Sustainble Development at Universitat Politècnia de València in 2012. He is an engineering professional with a passion for research in product development to reduce energy consumption.

He has worked for international companies as Idom Architecture and Engineering. He is currently working on Research and Development programmes at both public (Universitat Politècnica de Valencia) and private companies

(Vaillant Group).

Mr. Mora has published several articles related with energy and Computational Fluid Dynamic research: “CFD model of air movement in ventilated façade: comparison between natural and forced air flow” in the International Journal of Energy and Environment (Volume 4-3, 2013) and “Quantification of the efficiency of the ceramic ventilated façade by means of computational fluid mechanics techniques” in the Glass and Ceramic Journal (Vol.50-2, 2011).

Gonzalo López-Patiño is Industrial Engineer, Assistant Professor in the Hydraulic and Environmental Engineering Department at the Universidad Politécnica de Valencia.

He is currently Director of the Master Programme in Construction and Industrial Facilities at Universitat Politècnica de València.

Mr López-Patiño has more than a decade of experience in modelling building and urban facilities, especially in fluids applications (air, water and other industrial flows) and actually he has focused his researches on sustainable applications on building constructions.

P. Amparo López-Jiménez was born in Valencia. She graduated in Industrial Engineering in Universitat Politècnica de València (Spain) and Bachelor (Honours) in Combined Engineering in Coventry Polytechnic (UK). Finally, she became Doctor in Hydraulic Engineering in Universitat Politècnica de València.

She is Associate Professor in Hydraulic and Environmental Engineering Department at the Universidad Politécnica de Valencia. She is currently the Associate Director of the UPV Hydraulic and Environmental Engineering Department.

Dr López-Jiménez has more than a decade of experience in research and teaching in Engineering fields, she is author and editor of several publications about Hydraulic an Environmental Engineering and Fluid Dynamics. She has participated in national and international R&D projects and co-organized International Seminars and Networks. She is an experienced University Teacher, an active researcher and a former practicing engineer.