447: Numerical study on thermal and fluid dynamic behaviour of an open-joint ventilated façade

Marcos González [1]*, Eduardo Blanco [1], José Luis Río [1], Jorge Pistono [1], Cristina San-Juan [2]

Departamento de Energía, Universidad de Oviedo, Gijón (Asturias), Spain[1]*
gonzalezfmarcos@uniovi.es
Unidad de Eficiencia Energética en la Edificación, CIEMAT, Madrid, Spain[2]

Abstract

Open-joint ventilated façade is a new building system that disposes tiles hung on a metallic-frame structure attached to the exterior face of the conventional wall. One of the most claimed features about ventilated façades is their ability to reduce cooling thermal loads of buildings, taking advantage of the buoyancy effect created by solar radiation inside the ventilated cavity. This paper introduces the thermo fluid-dynamic behavior of such framework, analyzed with CFD techniques. It leads to conclude that, if well designed, ventilated façades may be able to get valuable energy savings in summer cooling of buildings.

Keywords: Ventilated façade, CFD, energy-efficient building, solar passive design

1. Introduction

Open-joint ventilated façade is a new external building element erected upon a thermally insulated wall by means of a metallic-frame structure. Over that structure a coating material (metallic, ceramic, stone or composite) is placed in an arrangement of slabs. Although different fastening systems exist for that arrangement, the vast majority of them present a space of air between the slabs and the inner wall that creates a vertical cavity. In order to allow thermal expansion, a series of thin gaps or joints are shaped from slab to slab, enabling ambient air to enter and leave the cavity. A simplified layout representing a generic ventilated façade is shown in Fig.1. Combining solar radiation and temperature differences can produce natural convection under certain conditions, cooling the wall and reducing heat gain through it. If such situation could be quantified, building engineers would be provided with a new green-building design capability.

During the last years the sales of home air-conditioning units have been growing continuously, mostly due to the fall in their costs. This situation is transferring the otherwise winter electric peak-power demand to the summer, particularly in southern-European countries like Spain. To cope with this issue the European Union developed Directive 2002/91/CE [1] about energy performance of buildings, then translated to Spanish Technical Building Code (Código Técnico de la Edificación or CTE) [2] inside its HE Basic Document about energy saving (Documento Básico HE sobre ahorro de energía) and Royal Law (Real Decreto) 47/2007 [3] about certification schemes for new buildings' energy performance. Further information about the Spanish legislation can be found in Ref. [4].

This new legal context tries to motivate the adoption of building systems pointed towards energy efficiency and sustainability, and brings the need to carry out thorough studies in order to obtain effective design tools for these systems. It would also be of interest to investigate the performance of a ventilated façade in winter, and then assess both situations providing a whole-year analysis and design tool.

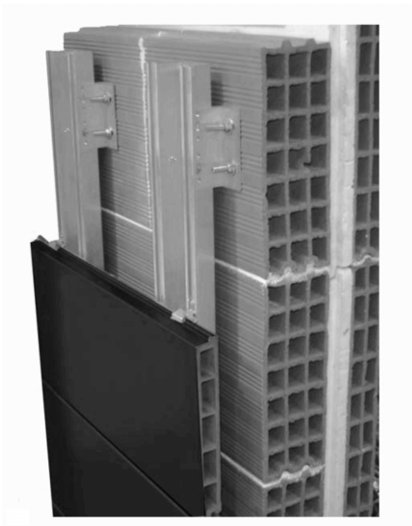

Fig 1. Typical open-joint ventilated façade

Ventilated façade is by itself an ambiguous concept and it is frequently associated with double-skin ventilated façade (DSF). The open-joint ventilated façade (OJVF from now on) here explored is quite different. First and most outstanding: materiality. DSF use to be glazed whilst OJVF is built with opaque materials. Other important differences are: size, shape and locations of the openings and cavity width and depth. Such distinctions affect both fluid dynamic and thermal behavior and consequently its design factors and methods. However a brief review on DSF could help to understand some common

aspects; design parameters and correlations presented by Pappas and Zhai [5] are particularly interesting on that subject. Another cladding system presenting similarities with OJVF is the building integrated photovoltaic thermal (BIPV/T) system, broadly studied by Moshfegh and Sandberg [6], Brinkworth [7, 8], Brinkworth et at. [9] and Liao et al. [10]. Additionally, some authors have been trying to simplify the element in order to obtain rough solutions: Balocco [11], Ciampi et al. [12, 13] and Lorente [14] applied analytical methods. Balocco also approached the problem using dimensional analysis [15]. It is remarkable to notice that neither the above mentioned works nor any other known to the authors have introduced significant information about the OJVF heat transfer problem.
Additionally, recent works [16, 17] have tried to use a simple energy simulation model to predict the performance of the OJVF, finding good agreement with CFD and experimental results, despite the uncertainties involved in these models.

2. OJVF heat transfer problem

For a better explanation here we compare the heat transfer problem in the OJVF with the traditional air cavity wall, which consists of two layers separated by a space of air. The air reduces heat conduction to a negligible value but allows convection and radiation to transfer heat between layers. If the cavity is closed the air moves in a loop gaining heat and rising near the hot layer and sinking along the colder. On the other hand if the air is free to enter and leave the cavity, like it is through the joints in the OJVF, a new unknown intervenes: mass flow rate of air circulating along the cavity. Fig.2 shows the heat transfer mechanisms involved.

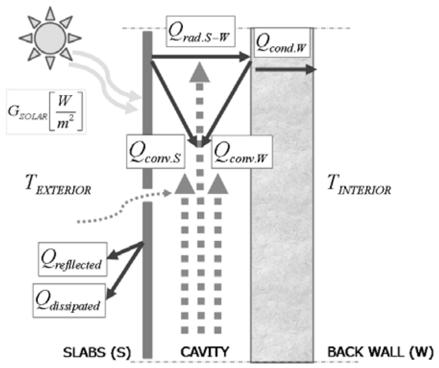

Fig 2. OJVF heat transfer problem definition

The solution of the closed cavity problem is straightforward, just solving the thermal balances in both walls, because the amount of heat loss in one must be equal to the amount of heat gain in the other. The OJVF is not so simple, since the air circulates along the cavity removing (or increasing) heat at an unknown rate, and since there is natural convection, air motion and thermal field are strongly coupled and we must solve simultaneously both groups of equations: the Navier-Stokes governing air motion and the energy equation governing heat transfer. On a common sunny day in summer, the equation linking the heat fluxes in the OJVF slabs is written as follows (for the steady state):

$$G_{SOLAR} = \dot{Q}_{reflected} + \dot{Q}_{dissipated} + \dot{Q}_{conv.S} + \dot{Q}_{rad.S-W} \quad (1)$$

Where G_{SOLAR} is the mean solar radiation perpendicular to the wall, $\dot{Q}_{dissipated}$ and $\dot{Q}_{reflected}$ the rate of heat dissipated and reflected from the outer slabs (S) to the exterior air, $\dot{Q}_{conv.S}$ the convective heat transfer from that surface to the air inside the cavity and $\dot{Q}_{rad.S-W}$ the amount of heat radiated from the outer slabs (S) to the inner wall (W).
In the same way the inner walls reach equilibrium according to the next equation:

$$\dot{Q}_{rad.S-W} = \dot{Q}_{conv.W} + \dot{Q}_{cond.W} \quad (2)$$

With $\dot{Q}_{conv.W}$ the convective heat transfer from the inner wall (W) to the air inside the cavity and $\dot{Q}_{cond.W}$ the total heat gain added to the room, i.e.: façade thermal load. We must remove that load and others (such as occupation, lightning, electronic equipment$\cdots$) in order to maintain thermal comfort.
The sum of the convection terms in equations (1) and (2), $\dot{Q}_{conv.S}$ and $\dot{Q}_{conv.W}$, represents the heat rate evacuated by ventilation out of the façade and thus prevented from reaching the room's interior environment. The thin joints further complicate air movement and persuade us not to try to approach the problem with a simplified model. Instead, here we use the conjugate heat transfer method [18], extending the capabilities of a CFD code with radiation and conduction models. Though some papers have classified that approach as computationally expensive [19], it is remarkable that the model here studied is relatively small (only one storey high) and that the final goal of this work is to find out the influence of the ventilated cavity by resolving accurately the air flow inside it.

3. Methodology. Numerical model

Computational Fluid Dynamics (CFD) is a group of techniques that allows us to predict the behavior of a fluid in motion. Generally the solution process consists of the following steps:

- Define a domain for the problem, where the equations are going to be solved;
- Divide the domain into a group (or grid) of volumes (discretize);
- Set up a group of boundary conditions;
- Complete the problem, if necessary, with turbulence and radiation models;

- Iterate the equations until convergence is reached.

The following subsections explain the particular details of the ventilated façade model.

3.1 Domain

To properly capture the effect of the joints it is necessary to create a 3-dimensional model. The geometry and dimensions of the ventilated façade here analyzed are collected in Table 1. No bearing structure has been included in this model, in an effort to minimize its complexity and hence the computational cost. The properties of the materials considered are listed in Table 2.

Table 1: CFD model dimensions

Dimension	Value	Unit
Total domain height	2.425	m
Total domain width	0.6025	m
Total domain depth	1.06	m
Vertical joints height	5	mm
Horizontal joints width	2.5	mm
Cavity depth	5	cm
Slab thickness	1	cm
Inner wall thickness	13	cm

3.2 Grid

Creating an adequate grid to solve a CFD problem is always a delicate task: the finer it is the better solution we get but it comes with a price, the larger resources consumed, in terms of computation time and computer memory. The geometry above proposed demands a highly refined grid, allowing us to estimate air flow through the joints. Fig.3 shows the solution proposed, with nearly 1 million tetrahedrons. Notice the volume reduction towards the domain's boundaries and the increase towards the cavity and the joints, where flow is expected to be more complex.

Table 2: Material's properties

Element	ρ	C_p	k	μ
Unit	Kg/m^3	$J \cdot Kg/{}^{o}K$	$W/m \cdot {}^{o}K$	$10^5 \cdot Kg/m \cdot s$
Slabs	2800	1000	3.5	–
Inner Wall	1000	1000	0.046	–
Air	1.145	1010.178	0.025	1.841

3.3 Boundary conditions

Boundary conditions are a series of assumed values required to perform the simulations. The accuracy of CFD highly depends on correct guesses for these assumptions, so it is important to choose them carefully. The major problem encountered in this case was the thermal boundary for the inner wall. If we consider the building's interior air at constant temperature (air-conditioned room) and uniform temperature distribution it is possible to define a global (radiation plus convection) heat transfer coefficient for the inner wall with a constant value. Such assumption is not expected to introduce serious errors, as long as the convection in that wall is always natural. Table 3 lists other values here adopted, following literature recommendations and general CFD guidelines.

Table 3: Boundary conditions

Surface	Boundary Type	Defined variable	Value
Domain Top	Pressure Outlet	Pressure (Pa)	0
Domain Bottom	Pressure Inlet	Pressure (Pa)	0
Domain Front	Wall	Temperature (${}^{o}C$)	$T_{EXTERIOR}$
Domain Back (inner wall)	Wall	Heat Transfer	$T_{EXTERIOR}$ h
Cavity Top	Wall	Heat Flux	0 (Adiabatic)
Cavity Bottom	Wall	Heat Flux	0 (Adiabatic)
Left Lateral	Symmetry	–	–
Right Lateral	Symmetry	–	–

3.4 Turbulence model

The geometry of this model presumably introduces turbulent flow regimes. Direct simulation of the turbulence for this problem would require huge resources, and therefore it is not an option. Instead, the instantaneous governing equations can be time-averaged, ensemble-averaged, or otherwise manipulated to remove the small scales, resulting in a modified set of equations that are computationally less expensive to solve. However, the modified equations contain additional unknown variables, and turbulence models are needed to determine these variables in terms of known quantities. The standard $\kappa - \varepsilon$ model [18] was used here, as a good compromise between accuracy and solving time [19].

3.5 Radiation model

Thermal radiation is the transfer of heat from one body to another by electromagnetic waves. As it has been shown by previous works [20] radiative heat transfer cannot be neglected in a two parallel plate configuration like the OJVF and thus an equation must be added. The discrete ordinates (DO) radiation model [21] has been chosen. That model solves the radiative transfer equation for a finite number of discrete solid angles. Air is considered to be a non-participating medium. Finally, solar radiation has been introduced as a thermal load inside the slabs, uniformly distributed.

3.6 Other considerations

To further reduce computation times, the Boussinesq approximation [22] was used to model buoyancy (free convection) effects. All other thermo-physical properties were assumed to be constant and the flow steady. As comparative background the traditional cavity wall was modeled and solved, adopting the same dimensions and materials as for the OJVF.

4. Results and discussion

Based on the above models, different environmental conditions were considered. Summer simulations were carried out with $T_{INTERIOR} = 24.5^{\circ}C$, $T_{EXTERIOR} = 30^{\circ}C$ and solar radiation varying from 0 to $800 W/m^2$. In winter these data were: $T_{INTERIOR} = 22^{\circ}C$, $T_{EXTERIOR} = 8^{\circ}C$ and solar radiation varying from 0 to $600 W/m^2$. Detailed results are shown for one case: solar radiation $400 W/m^2$ in summer.

4.1 Wall temperatures

As long as the exterior air is cooler than the solid materials of the wall (generally this is true when the façade is exposed to solar radiation), the ventilated structure keeps cooler than the cavity wall. Fig. 4 shows the temperature rise in both configurations, note the cooling effect of the air. The exterior edges of the slabs are cooler due to the presence of vertical joints.

4.2 Heat fluxes

Making use of CFD codes one can easily calculate the heat fluxes involved in the ventilation process from the velocity and temperature fields, allowing us to formulate the energy balance of the façade. Fig. 3 presents this balance in the studied case. The non-uniform temperature field and the presence of joints highly change the heat fluxes along the wall. From this figure it is easy to see the position of the joints and the direction of the ventilation air.

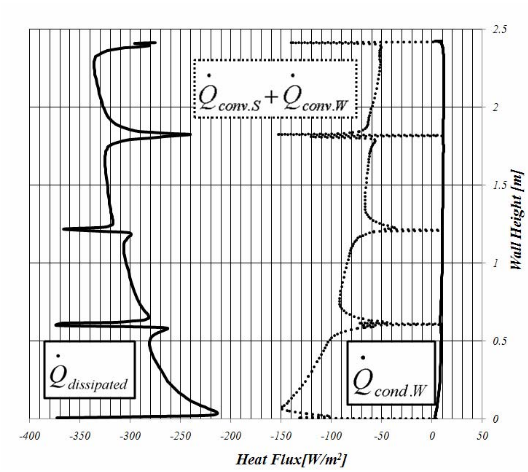

Fig 3. OJVF heat fluxes

Note that $\dot{Q}_{cond.W}$, representing the total heat flux entering the room, has been considered positive, whereas the heat rejected from the façade (dissipated and ventilation heat) has been plotted as negative, trying to identify the direction of the heat flux.

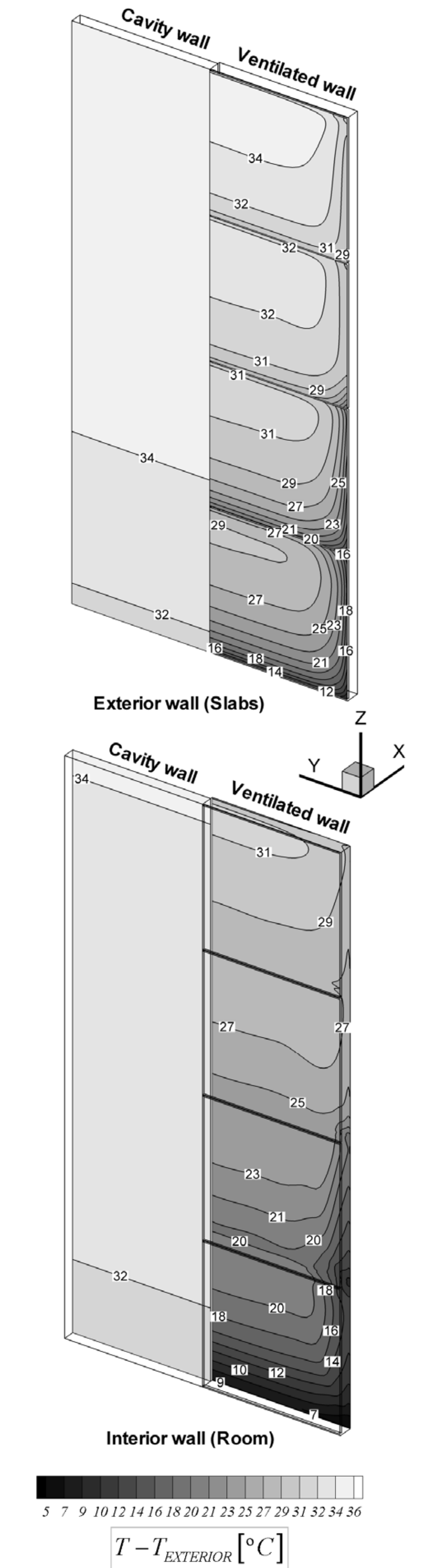

Fig 4. Comparison of temperature rises over ambient

4.3 Velocity field

The strong coupling between thermal and air velocity fields in the OJVF problem leaded us to employ CFD techniques. As we said above this is essential to evaluate the effect of the heat rate extracted from the cavity by ventilation. It is interesting to note that the air movement inside the cavity is heavily three-dimensional, due to the presence of the joints. Fig. 5 shows the vertical velocity profiles along the cavity. To complete the results, Fig. 6 presents the stream lines of the air moving along the cavity.

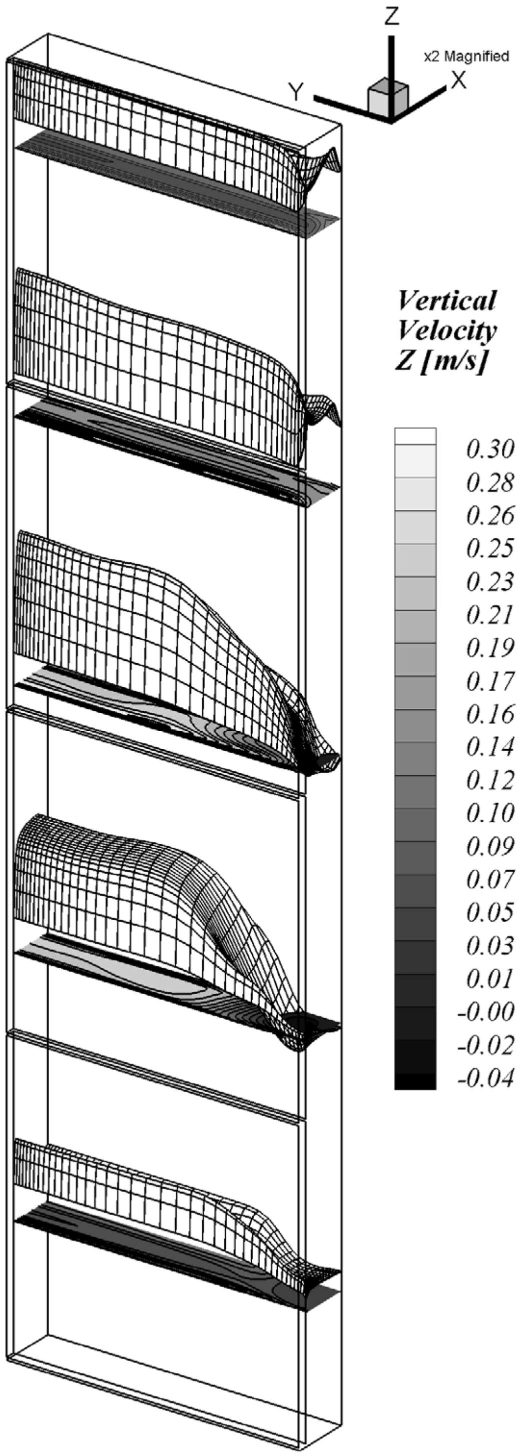

Fig 5. Z-velocity profiles inside the ventilated wall

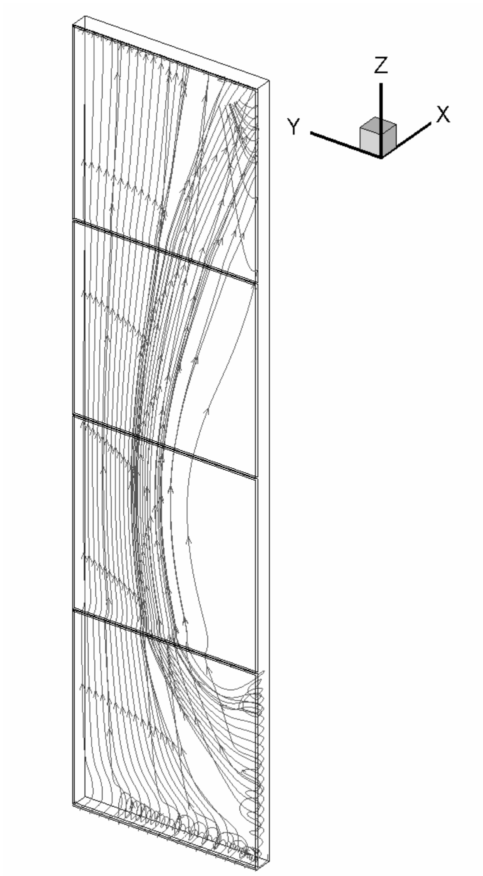

Fig 6. Streamlines of air moving along the cavity

4.4 Extended results

As we have seen, solving both temperature and velocity fields allowed us to quantify the heat fluxes involved in the OJVF problem. Among these fluxes one is of paramount importance: the heat conducted inside the room. We can make use of it to evaluate the performance of the OJVF for different environmental conditions. Here we compare the OJVF results with those obtained for a conventional two-pane airtight cavity wall, considering the same materials and dimensions.

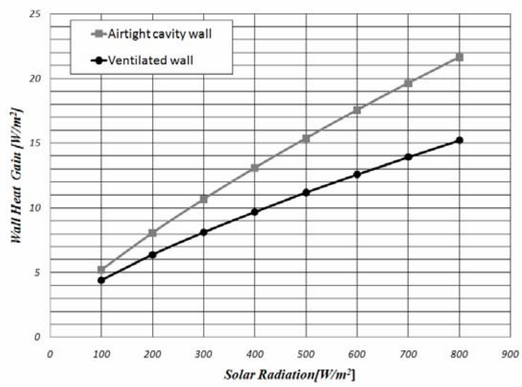

Fig 6. Wall heat gain. Ventilated vs. sealed cavity. Summer case

Figs. 6 and 7 present the heat gain through both configurations for the summer and winter cases as a function of the incident solar radiation. According to these plots both configurations achieve almost the same heat gain at low insolation. But when solar contribution grows, buoyancy causes air movement along the cavity, extracting heat and thus reducing its temperature and the total wall heat gain

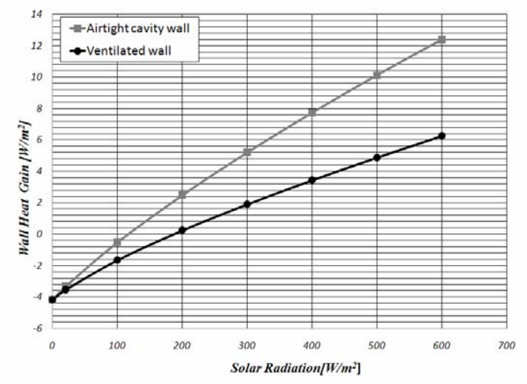

Fig 6. Wall heat gain. Ventilated vs. sealed cavity. Winter case

5. Conclusions

In this paper a CFD model of an open-joint ventilated façade has been discussed. For the proposed configuration, it is possible to get a valuable saving of energy, particularly for the summer season.

Although few data are available, the results generally agree with previous works, supporting the concept of ventilated façade as a green-building element, capable of performing better than the traditional solutions. However, further work is needed to improve and validate the model here presented.

6. Acknowledgements

ARFRISOL, ref. PS-120000-2005-1 is a scientific-technologic project qualified as Strategic by 2004-07 National Plan of Research, Development and Diffusion, co-financed by the European Regional Development Funds and the former Science and Education Ministry. Thanks are due to all members of PSE-ARFRISOL partnership for their co-operation.

7. References

1. Directive 2002/91/CE (2002) of the European Parliament and of the Council of 16 December 2002 on the energy performance of buildings.
2. Ministerio de Vivienda (2006). Código Técnico de la Edificación (in Spanish).
3. Ministerio de la Presidencia (2007). Real Decreto 47/2007 de 19 de Enero (in Spanish).
4. X. García Casals (2006). Analysis of building energy regulation and certification in Europe: Their role, limitations and differences. *Energy and Buildings,* 38: p. 381-392.
5. A. Pappas, Z. Zhai (2008). Numerical investigation on thermal performance and correlations of double skin façade with buoyancy-driven airflow. *Energy and Buildings*, 40: p. 466-475.
6. B. Moshfegh, M. Sandberg (1998). Flow and heat transfer in the air gap between photovoltaic panels. *Renewable and Sustainable Energy Reviews*, 2: p. 287-301.
7. B.J. Brinkworth, R.H. Marshall, Z. Ibarahim (2000). A validated model of naturally ventilated PV cladding. *Solar Energy*, 69(1): p. 67-81.
8. B.J. Brinkworth (2000). Estimation of flow and heat transfer for the design of PV cooling ducts. *Solar Energy,* 69(5): p. 413-420.
9. B.J. Brinkworth (2002). Coupling of convective and radiative heat transfer in PV cooling ducts. *ASME Journal of Solar Energy Engineering,* 124: p. 250-255.
10. L. Liao, A. Athienitis, K. Park, M. Collins, Y. Poissant (2005). Numerical study of conjugate heat transfer in a BIPV-Thermal system, in: *Proceedings of ISEC2005 2005 International Solar Energy Conference*, Orlando-Florida (USA).
11. C. Balocco (2002). A simple model to study ventilated façades energy performance. *Energy and Buildings*, 34: p. 469-475.
12. M. Ciampi, F. Leccese, G. Tuoni (2003). Ventilated facades energy performance in summer cooling of buildings. *Solar Energy*, 75: p. 491-502.
13. M. Ciampi, F. Leccese, G. Tuoni (2005). Energy analysis of ventilated and microventilated roofs. *Solar Energy*, 79: p. 183-192.
14. S. Lorente (2002). Heat losses through building walls with closed, open and deformable cavities. *International Journal of Energy Research*, 26: p. 611-632.
15. C. Balocco (2004). A non-dimensional analysis of a ventilated double façade energy performance. *Energy and Buildings*, 36: p. 35-40.
16. M. González, E. Blanco, J. Pistono (2008). Adjusting an energy simulation model by means of CFD techniques to analyze open-joint ventilated façades energy performance, in: *Proceedings of WREC-X 2008 World Renewable Energy Congress,* Glasgow (UK).
17. E. Naboni (2007). Ventilated opaque walls-A performance simulation method and assessment of simulated performance. *Lawrence Berkeley National Laboratory Environmental Energy Technologies Division. Seminar Notes.*
18. B.E. Launder, D.B. Spalding (1974). The numerical computation of turbulent flows. *Comp. Meth. Appl. Mech. Energy,* 3: p. 269-289.
19. Q. Chen (1995). Comparison of different $\kappa - \varepsilon$ models for indoor airflow computations. *Numerical Heat Transfer, Part B*, 28: p. 353-369.
20. B.J. Brinkworth (2002). Coupling of convective and radiative heat transfer in PV cooling ducts. *ASME Journal of Solar Energy Engineering*, 124: p. 250-255.
21. E. H. Chui, G. D. Raithby (1993). Computation of radiant heat transfer on a non-orthogonal mesh using the finite-volume method. *Numerical Heat Transfer, Part B*, 23: p. 269-288.
22. D.D. Gray, A. Giorgini (1976). The validity of the Boussinesq approximation for liquids and gases. *International Journal of Heat and Mass Transfer*, 19: p. 545-551.