



Jul 1st, 12:00 AM

Model of ventilated façade in buildings by using CFD techniques

P. Amparo López-Jiménez

Miguel Mora-Pérez

Gonzalo López-Patiño

Miguel A. Bengochea Escribano

Follow this and additional works at: <https://scholarsarchive.byu.edu/iemssconference>

López-Jiménez, P. Amparo; Mora-Pérez, Miguel; López-Patiño, Gonzalo; and Bengochea Escribano, Miguel A., "Model of ventilated façade in buildings by using CFD techniques" (2010). *International Congress on Environmental Modelling and Software*. 201.
<https://scholarsarchive.byu.edu/iemssconference/2010/all/201>

This Event is brought to you for free and open access by the Civil and Environmental Engineering at BYU ScholarsArchive. It has been accepted for inclusion in International Congress on Environmental Modelling and Software by an authorized administrator of BYU ScholarsArchive. For more information, please contact scholarsarchive@byu.edu, ellen_amatangelo@byu.edu.

Model of ventilated façade in buildings by using CFD techniques

P. Amparo López-Jiménez^a, Miguel Mora-Pérez^a, Gonzalo López-Patiño^a, Miguel A. Bengochea Escribano^b

a. Hydraulic and Environmental Engineering Department, Universidad Politécnica de Valencia, Spain. palopez@gmmf.upv.es, mimope4@etsii.upv.es, glpatin@gmmf.upv.es

b. KERABEN GRUPO. SA. ma.bengochea@keraben.com

Abstract: In some countries, summer over-heating is a big problem in a building's energy balance. Ventilated façades are a useful tool when applied to building design, especially in bioclimatic building design. A ventilated façade is a complex, multi-layer structural solution that enables dry installation of the covering elements.

The objective of this paper is to quantify of the improvement in the efficiency of the thermal behaviour of buildings when this sort of façade is installed. These improvements are related to the ventilation capacity of the additional structure to the shield mainly for saving cooling power in summer in warm countries. This effect is due to convection produced in the air gap of the façade. This convection depends on the air movement inside the gap and the heat transmission in this motion. These quantities are mathematically modelled by Computational Fluid Dynamics (CFD) techniques using a commercial code: STAR CCM+. The proposed method allows an assessment of the energy potential of the ventilated façade and its capacity for cooling.

Keywords: Ventilated façade, Computational model, fluid mechanics

1. INTRODUCTION. VENTILATED FAÇADE

Actual energetic situation encourages engineers and architects to install strategies in buildings that can potentially improve their energy performance. In this sense, the use of ventilated façades can often have a positive contribution to this objective. Their effect has to be quantified when calculating for the overall performance of the building, Infield et al, [2006].

In this paper, the cooling capacity of ventilated façade is analyzed. In some countries, summer over-heating is a big problem in building energy balances. Ventilated façades are a powerful tool when applied to building design, especially in bioclimatic building design.

A ventilated façade is a complex, multi-layer structural solution that enables dry installation of the covering elements, taking into account all the essential construction, hygrometric, static, safety and aesthetic aspects of a building technology. It consists of external façade cladding, a sub-structure (generally made of aluminium profiles), anchored to the wall surface of the building, insulating material and an air gap between cladding and insulating material.

Ventilated façade and wall coverings were developed to protect buildings against the combined action of rain and wind by counterbalancing the effects of water beating on walls and keeping the building dry, with high-level aesthetic characteristics and good heat insulation and soundproofing.

1. External cladding materials: This layer defines the visual appearance of the building and it may be constructed of glass, marble, stone, alukobond, etc. The role of this layer is to provide protection against the sun, rain and other atmospheric influences. This layer also receives and transmits loads generated by wind to the supporting structure.
2. Air gap: The ventilated layer size must be a minimum of 3cm to a maximum of 10cm (in accordance with the recommendations of the façade manufacturer). The role of this layer is to take moisture out of the building (from internal, warmer space to outer, colder environment; that is why a vapour barrier is not necessary), as well as to eliminate water that may penetrate the external veneer. In summer, it prevents heat getting into the building (decreases the heating of the basic wall mass), while in winter it is an additional thermal insulator.
3. Insulating material. Thermal insulation material in a system of ventilated façades must be rigid and properly bonded in order to prevent tearing and dispersion of the material due to any stronger air flow in the ventilated layer. Higher density of insulation material also prevents the infiltration of cold air into the material.
4. Aluminium sub-structure. The substructure is mechanically tightened to a supporting wall, in accordance with instructions of the façade manufacturer. The role of the substructure is to support the external, façade covering and to transmit the load generated by wind to the bearing structure.

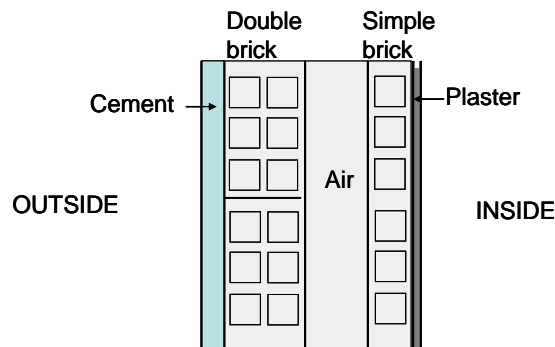


Figure 1. Standard wall considered for simulations

These materials combined in the ventilated façade must achieve some basic requirements in both summer and winter conditions Balocco, [2002, 2003]:

- Air permeability: it must reduce heat dispersion during winter season and guarantee passive cooling effect to the ambient inside by combining convective and heat transport between the outer and inner walls
- Watertight: it must guarantee no water infiltration due to rain, humidity or capillarity, no condensation on the surface into the wall mass
- Thermal performance: it must guarantee a global thermal insulation and then the indoor thermal comfort

The objective of this paper is the quantification of the improvement in the efficiency of thermal behaviour of buildings when this sort of façade is installed, especially in summer conditions. These improvements are related with the ventilation capacity of the additional structure to the shield mainly for saving cooling power in summer in warm countries. This effect is due to convection produced in the air gap of the façade, Kokogiannakis and Strachan [2007]; Gang, [1998]. This convection depends on the air movement inside the gap and the heat transmission in this motion Manz, [2003]; Yilmaz [2007]. These quantities are mathematically modelled by CFD techniques and experimented in laboratory prototypes to validate the results.

2. MATHEMATICAL MODEL OF THE FAÇADE

2.1 Modelled Geometry

A façade in a building is modelled in order to obtain the velocities profiles in the air gap and the temperature distribution across the air and the faces of the solid boundaries. The dimensions of the model are:

Length = 6 m.

Width = 2 m.

Height = 7 m.



Figure 2. Building with façade modelled with CFD technique

A detail of the air gap in the numerical model is shown in Figure 3.

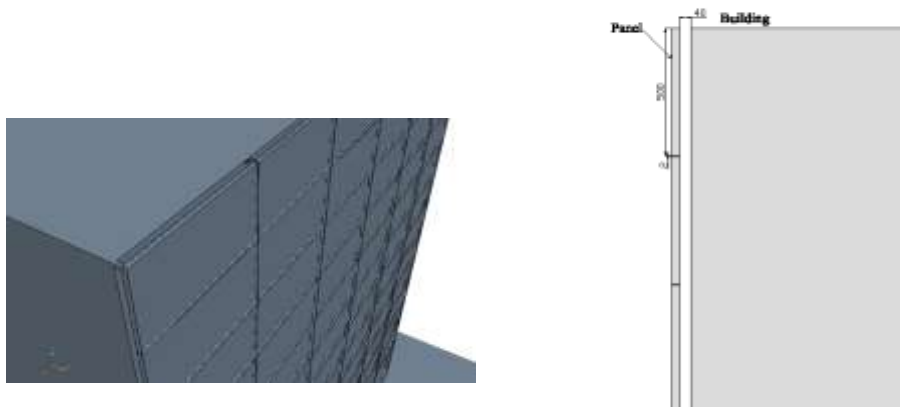


Figure 3. Detail of the geometry for CFD model. Sketch of dimensions considered in the numerical model.

2.2 CFD model for ventilated façade

Computational fluid dynamics (CFD) has been widely used for simulation of the air movement in and around commercial, educational, industrial and residential buildings and interaction between indoor and outdoor environments since 1970s. Nevertheless, this technique has not been used for modelling ventilated façades, and it has a proved capability to represent the flow and heat transfer in these sort of simulations. Engineering simulations have been widely used. Predicting behaviours and performance of constructive solutions is a capable tool to quantify these benefits. This simulation will enable engineers to perform what-if studies and to compare alternatives and different processes that otherwise would be impractical.

In this case, quantifying the difference of temperatures in a room when ventilated façade is used or not, is a possibility that enables designers to optimize their constructive solutions by simulation techniques and not by trial-and-error methodologies, which is one of the most important advantages of computational models.

In order to compare the results of the measurements and to visualize many other aspects, this case has been implemented within the software STARCCM+. As already indicated, the computational model solves numerically the governing laws of Fluid Dynamics. These equations, taking into account turbulent phenomena, are solved in a geometrical domain, given a number of suitable boundary conditions. In CFD the relevant velocity, pressure and temperature fields are calculated in a discrete manner at the nodes of a certain mesh or grid and they are represented along the mesh. The use of these computational tools to gain insight into the velocity and pressure fields of the modelled flows currently constitutes a powerful tool for researchers working in the field of environmental engineering. It can be said that CFD models have enabled us to understand a huge number of subtle intricacies of our world in a new way.

The way to think about solving the problem is to consider the total flow that enters or leaves a cell through its boundaries. Continuity implies that the net flow through the boundary of an element is equal to the flow produced by the internal sources. In fact, an equation of balance for the property in as small a volume as desired is solved. But, instead of carrying out a discretization of the differential equation, the process is reversed: each point is associated with a mean value obtained by integrating the differential equation, instead of an approximate value of the property at the point.

Typically, this produces a set of linear equations whose matrix of coefficients is sparse. Before solving this system, boundary conditions must be taken into account. Then, the subsequent system of equations is solved to obtain the distribution of the property in all the nodes. In this way, both pressures and velocities will be known, as well as the magnitudes derived from the solution of the equations (mainly turbulence properties).

The advantage of using these models resides in the fact that they are able to reproduce real problems of Fluid Mechanics to any degree of complexity. Furthermore, they are able to visualize hydrodynamic aspects impossible to measure or represent in a real case (i.e. stream lines) that have great importance in the comprehension of the studied phenomena. The conservation equations solved by the code are those of mass and momentum. The continuity or mass conservation equation solved by the software used in the dynamic study of the present problem is the following one:

$$\frac{\partial \rho}{\partial t} + \nabla \rho \bar{v} = S_m \quad (1)$$

Where ρ is the fluid density, \bar{v} is velocity and S_m represents the mass source contained in the volume of control. For other geometries, suitable coordinates, namely spherical or cylindrical, should be used. Also, the momentum takes the form:

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

Where p is the static pressure, g and F are the gravitational and outer forces defined on the control volume respectively, and $\bar{\tau}$ the stress tensor defined by:

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

Where μ is the eddy viscosity, I is the unit tensor and the third term accounts for the effect of the expansion of volume.

All conditions and properties are defined via STAR-CCM+ and solved using the coupled solver. The results are displayed via post-processing tools available in STAR-CCM+. The analysis includes the following heat transfer modes: Diffusion and natural convection in the surrounding fluid. The volume mesh in a simulation is the mathematical description of the space (or geometry) of the problem being solved.

The defined geometry is three-dimensional. This model is designed to work on three-dimensional meshes, and should only be activated if the mesh is indeed three-dimensional. The use of a one-cell-thick three-dimensional mesh is much less efficient than using a true two-dimensional mesh for two-dimensional and axisymmetric simulations.

The Steady model is used for all steady-state calculations. When this model is activated, the concept of a physical time-step is meaningless. The Motion models in STAR-CCM+ provide methods for computing and accessing information relating to the motion of the reference frame and/or mesh. The Stationary model is used for simulations in which there is no mesh motion and in which all reference frames are stationary. A stationary reference frame condition is, however, added to all regions in which the stationary model is activated. The specification in this condition is important for the creation of periodic interfaces and for subsequent transformations involving these interfaces. A segregated flow model is used. The Segregated Flow model solves the flow equations (one for each component of velocity, and one for pressure) in a segregated, or uncoupled, manner. The link between the momentum and continuity equations is achieved with a predictor-corrector approach. The complete formulation can be described as using a collocated variable arrangement (as opposed to staggered) and a Rhie-and-Chow-type pressure-velocity coupling combined with a SIMPLE-type algorithm. This model has its roots in constant-density flows. Although it is capable of handling mildly compressible flows and low Raleigh number natural convection, it is not suitable for shock-capturing, high Mach number and high Raleigh-number applications.

The Gravity model permits the inclusion of the buoyancy source terms in the momentum equations when using the Segregated Flow model or the Coupled Flow model. For Constant Density flows, the Boussinesq Model is used to approximate the buoyancy source term. A K-Epsilon turbulence model is used for representing turbulence. The turbulence models in are responsible for providing closure of the governing equations in turbulent flows. A K-Epsilon turbulence model is a two-equation model in which transport equations are solved for the turbulent kinetic energy and its dissipation rate. Various forms of the K-Epsilon model have been in use for several decades, and it has become the most widely used model for industrial applications. Since the inception of the K-Epsilon model, there have been countless attempts to improve it. The Standard K-Epsilon Model is a de facto standard version of the two-equation model that involves transport equations for the turbulent kinetic energy and its dissipation rate.

2.3 Computational Model

In this study, for quantification of the performance of the ventilated façade, a heat balance is proposed. The objective is the comparison of the temperature in the external face of the building wall with the presence of the exterior ventilated façade and without it. Figure 4 shows the relevant temperatures without the exterior ventilated façade (a) and with the simplified exterior panels (b). In this case, for the numerical model, the temperature of external air T_{air} has been taken to be 294°K. Temperature of interior wall (T_1) has been taken to be 304 °K. With this consideration, T_2 of the wall is modelled. It is desired to maintain 296°K in the room, which is T_{inside} .

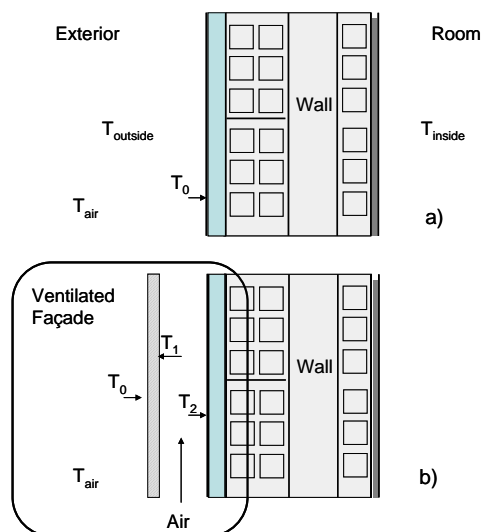


Figure 4. Temperatures considered in the model

The CFD model will simulate the velocity of air inside the gap, depending on the exterior wind around the building. Considering a velocity of air exterior and perpendicular to façade of 0.5 m/s, the velocities field are calculated by the CFD, as it is shown in Figures 5 and 6.

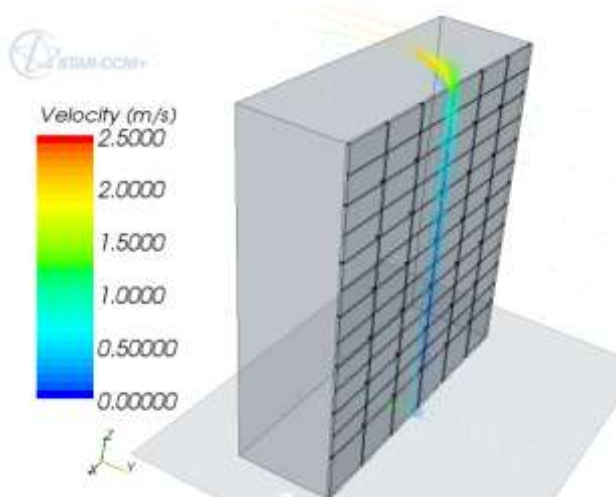


Figure 5. Velocity profile in medium line

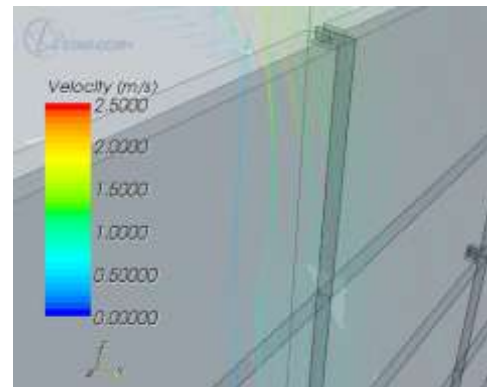


Figure 6. Detail of velocity profile

With the temperature boundary conditions, and the exterior velocity perpendicular to the wall, the numerical model determines the temperature T_2 , in this case dependant on the action of the façade. These temperatures in the whole plane of the building are represented in Figure 7.

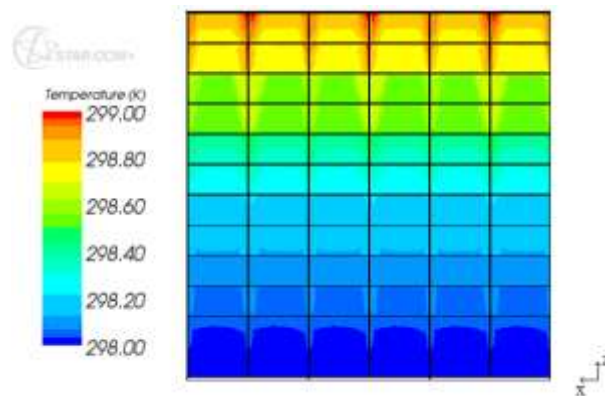


Figure 7. Distribution of temperatures T_2 in the building

Considering that the temperature of the internal side of the façade panel is constant: $T_1=304^{\circ}\text{K}$, this means that temperature has decreased in the air gap, as a consequence of the ventilation, as is described on Figure 8. The mean decrease along all the vertical height of temperature in the moving air, in this case is $4,482^{\circ}\text{K}$. When comparing with non ventilated façade, the external wall temperature will achieve 304°K or more.

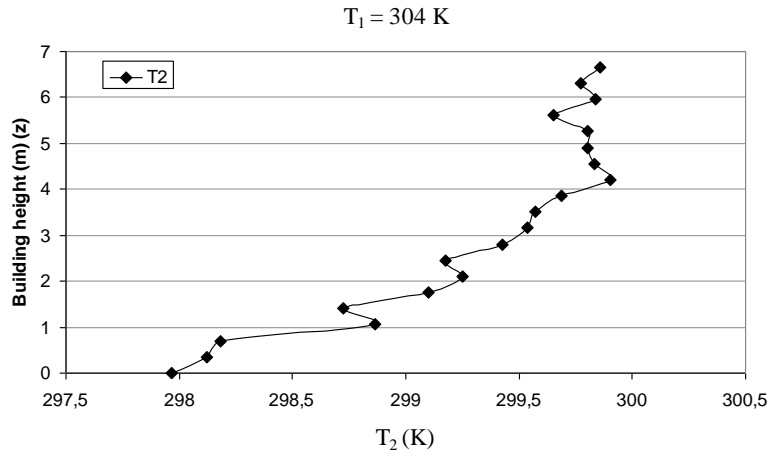


Figure 8. Central distribution of temperatures T_2 in the building

This decrease of temperature caused by the presence of the air gap will determine the temperature of the wall: T_2 . This temperature T_2 will mean a saving of cooling power to maintain the same temperature in the room.

Studies performed by Chiampi in [2003] showed that one of the more affecting factors to efficiency increase is the exterior temperature. In summer conditions the energy savings will increase remarkably as solar radiation increases: the bigger the solar radiation is, the more efficient ventilated façades turn to be from an energy point of view.

This aspect has been also simulated with the current CFD analysis. Figure 9 shows the different temperature variation inside the air gap when increasing T_1 and then simulating T_2 for each case. As it can be observed in this Figure 9 as panel temperature increases, the temperature drop in the gap is bigger ($T_1 - T_2$) and the cooling action of the façade is more effective.

Figure 9 represents in the x-axis the difference between $T_1 - T_2$ being T_1 increased 1 K in each case (moving to the right hand). Is written in the legend the colour each variation corresponds, showing only the value of T_1 .

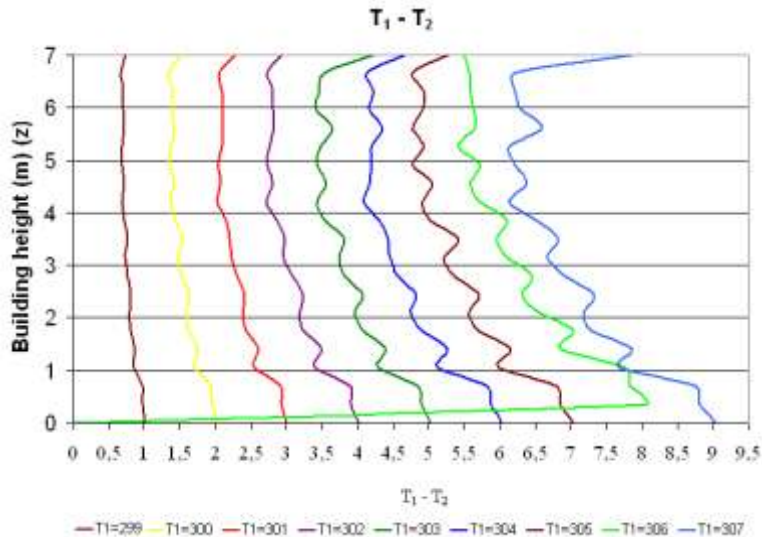


Figure 9. Variation of temperature of air inside the ventilated façade when considering different values for T_1

3. CONCLUSIONS

A numerical method has been illustrated in order to estimate the thermal performance of ventilated façades. A steady state calculation has been proposed to estimate the temperature evolution, together with the resolution of the fluid mechanics conditions by means of CFD techniques. To illustrate the effectiveness of the ventilated façade, a variation in the external temperature has been proposed in order to model the velocity of the air in the gap of the façade, and the cooling effect caused by the convective effect of the air. Consequently, the model shows an acceleration with bigger temperatures, increasing the cooling energy saved by the structure of the façade. In summer conditions the energy savings will increase markedly as solar radiation increases: the bigger the solar radiation is, the more efficient ventilated façades turn to be from an energy point of view.

A simplified CFD model for the ventilated façade has been presented. This simple model is useful to understand heat transfer mechanism in the air gap inside ventilated façades. It must be improved in future simulations with a complex heat transfer model, including all the radiation effects and the heat transmission inside the room and with external thermal variation conditions in order to include all the real effects affecting the façade behaviour. Future research will be addressed in this direction and in the experimental validations of the use of this CFD modelling with in situ measurements.

ACKNOWLEDGEMENTS

This study was carried out by the researchers of UPV involved in the projects DANAIDES (Ref. DPI2007-63424), supported by the Ministry of Science and Education of Spain.

REFERENCES

- Balocco, C. A simple model to study ventilated façades energy performance. *Energy and Buildings*, Vol 34, N 4. Pp 469-475. 2002
- Balocco, C. A non dimensional analysis of a ventilated double façade energy performance. *Energy and Buildings*, Vol 36, N 1. Pp 35-40. 2003
- Ciampi, M.; Leccese, F.; Tuoni, G. Ventilated façades energy performance in summer cooling of buildings. *Solar Energy* N. 75. Pp 491–502. 2003.
- Gang, G. A parametric study of Tromble walls for passive cooling of buildings, *Energy and Buildings*, Vol 27, N 1. Pp 37-43. 1998.
- Infield, D. Eicker, U., Fux, V., Mei L., Schumacher, J. *Building and Environment*. N 41. Pp. 893–901. 2006.
- Manz, H. Numerical simulation of heat transfer by natural convection in cavities on façade elements. *Energy and Buildings*, Vol 35, N 3. Pp 305-311. 2003.
- Kokogiannakis, G.; Strachan, P.. Modelling of double ventilated façades according to the CEN Standard 13790 method and detailed simulation. 547 2nd PALENC Conference and 28th AIVC Conference on Building Low Energy Cooling and Advanced Ventilation Technologies in the 21st Century. Crete island, Greece. 2007.
- Yilmaz, Z. Evaluation of energy efficient design strategies for different climatic zones: Comparison of thermal performance of buildings in temperate-humid and hot-dry climate. *Energy and Buildings*, Vol 39, N 3. Pp 306-316. 2007.