Transient Simulation of a Naturally Ventilated Façade in a Mediterranean climate

C. Balocco*
Dipartimento di Energetica "Sergio Stecco", Università degli Studi di Firenze
*Corresponding author: via Santa Marta 3 50139 Firenze Italy, carla.balocco@unifi.it

Abstract: Energy performances and building energy quality can be achieved by high performance envelopes. In this paper the thermal performance of a naturally ventilated façade, applied to a traditional wall, widely used in existing buildings, was investigated by a transient simulation. A three dimensional model of the façade, applied to the South-facing wall of an existent building was investigated in Florence and Treviso (Italy). The CFD Comsol simulation shows the ability of the software for modelling natural ventilation flows, driven by thermal buoyancy and stack effects in the system where the flow is constrained within a vertical channel. Results are also presented in dimensionless form based on parameters either known or checkable. Comparison obtained by simulating the system in two different climatic zones showed that thermal performance of the ventilated façade cannot be generalized and is particularly effective in reducing cooling loads, especially at average low latitudes, where the amount of total solar radiation striking the building walls is very important.

Keywords: natural ventilation, external building façade, CFD, simulation, cooling

1. Introduction

Recent studies have discussed the possibilities of improving energy performance of old existent buildings by their sustainable restoration and /or retrofitting, using naturally ventilated façades.

The naturally ventilated façade is a special technique for cladding building walls, based on a cladding system made up of slabs or panels detached from the building walls. In particular, the slabs dry-mounted on a metal substructure are fixed to the building load-bearing structure.

The insulating cladding is mounted inside the air cavity in adherence to the building wall and the air cavity between the cladding slabs and the insulating panel is ventilated by natural convection ("chimney effect").

In contrast to forced convection, buoyancy-driven (natural) convection components are more difficult to model and then it is more difficult to predict their performance. The basic simulation problem is to control and then take into account every parameter that describes energy performance of this complex thermodynamic system. All these parameters are connected together and contemporary to the outside and inside variable climatic conditions. In particular the complexity of the CFD (Computational Fluid Dynamics) codes for energy simulation depends also on difficulties associated with proper definition of boundary conditions and the quality and resolution of the mesh. A ventilated façade is a multifunctional thermodynamic system used to combine both outdoor aspects with building passive behaviour.

Several studies have highlighted some basic parameters of the energy performances of the ventilated façades:

- the channel width, that affects natural convection and air movement inside the air cavity; usually a reduced wall frictional resistance corresponds to the increasing air-gap width and the increasing air flow rate through the channel; this last effect increases with decreasing of the ratio between width and height of the channel [2],;
- the mass flow rate in the channel that increases with the cavity width and the stack effect increasing; it was also shown [3,5] that this effect has a maximum when the cavity width is 0.2-0.3 m; results of this research agree with others in literature [6];
- geometry and mean rugosity of the channel affect uniformity and continuity of the heat flux, the surface temperature variation along the channel and the pressure losses.

The prediction of the performance of a naturally ventilated façade is not straightforward because the approach has to take into account the primary climatic factors, temperature distribution inside the air cavity, the air flows and velocity profile that result from many simultaneous and connected thermal, conductive, convective, radiative and fluid flow processes.

The aim of this paper is to highlight the importance of CDF simulation to predict and study the energy performance of naturally ventilated façade systems, when experimental measurements cannot be carried out due to high costs and time needed. Furthermore the contribution of a naturally ventilated façade to the cooling energy savings of the building is highlighted.

2. The model studied

The ventilated façade studied is made up of a brick cladding layer bonded to the perimeter walls of a building by a special “dry-mounted” fixing structure (it is a mechanical assembly according to [9]) with an air cavity of 0.15 m thickness. This façade was applied to the South-facing wall of an existent building considered as located in Florence and then Treviso (Italy).

A wall with high thermal mass was chosen as a case study for investigation of the possible solar design strategies at the average latitudes like those of Florence (43°) and Treviso (45°). In the system studied the air inlet is between the two brick covering panels and at the bottom of the channel. Thermophysical properties of the different materials are provided in Table1.

Climatic data of the hottest day provided by the standard year of both locations were used and the external air temperature was corrected taking into account the incident solar radiation on the South covering panel.

3. The simulation

To investigate the development of the ventilation flow inside the system studied, a time dependent simulation based on general heat transfer and the incompressible Navier-Stokes on non-isothermal air flow, was performed.

Two transient simulations of the three dimensional models using hourly climatic data of the hottest day provided by the standard year of both locations [1], were performed using Comsol Multiphysics [4]. After many attempts, a good quality of the mesh was obtained by 100 3172 degrees of freedom with 262604 tetrahedral elements. The linear system solver “PARDISO” was used. The initial conditions for transient computation were obtained by running the simulation for several days before, assuming, for the initial indoor climatic conditions, a uniform internal air temperature of 20°C and 50% of relative humidity, as usually suggested. The sub-domain setting equations used for the general heat transfer:

air

$$\rho C_p \frac{\delta T}{\delta t} + \nabla \cdot (-k \nabla T) = Q + q_s T - \rho C_p u \cdot \nabla T$$

walls

$$\rho C_p \frac{\delta T}{\delta t} + \nabla \cdot (-k \nabla T) = Q + q_s T$$

T = temperature, H = enthalpy

$$H = C_p \frac{T}{y} + \frac{p_a}{\rho}$$

The sub-domain settings equations used for the incompressible Navier-Stokes model:

$$\rho \frac{\delta u}{\delta t} + \rho(u\nabla)u = \nabla\left[-p \cdot I + \eta \cdot \left(\nabla u + (\nabla u)^T\right)\right] + F$$

where $\nabla \cdot u =: 0$

The boundary conditions used for the general heat transfer:

a) “temperature” linked to the corrected outside air temperature hourly values for the brick external covering panel;
b) “convective flux” at the inlet of the channel and at the grating between the two brick panel;
c) “temperature” linked to the indoor constant air temperature for the surface of the wall facing the internal ambient.

The boundary conditions used for the incompressible Navier-Stokes model:

a) “outlet-pressure” for all the ventilation openings, fixed at the initial atmospheric pressure (1.013 kPa);
b) “symmetry” for all the other surfaces of the ventilation openings.

The input data used were as follows:

a) thermo-physical properties of all the different building materials reported in Table 1;
b) all boundaries of the computational domain, except the ventilation openings were modelled as no-slip boundaries;
c) hourly corrected external air temperature was used as boundary conditions for all the ventilation

openings;

d) the air thermal conductivity and heat capacity inside the fixing structure were considered constant;

e) referring to the incompressible Navier-Stokes model, in the air sub-domain, density and dynamic viscosity were considered as function of the air temperature and the volume force due to the buoyancy and function of air density following the Boussinesq approximation [4].

Table 1. Thermal properties of the layers of the ventilated façade

Strata from outside to the inside ambient	s (m)	λ ($Wm^{-1}K^{-1}$)	ρ (kgm^{-3})	c ($Jkg^{-1}K^{-1}$)
brick covering panel	0.035	0.5	1000	840
Air cavity	0.15	0.024	1.2	1000
Polistyrene	0.04	0.037	20	120
Solid brick wall	0.27	0.148	1600	850
Internal plaster	0.01	0.17	696	1089

4. Results and discussion

Referring to the dimensional analysis based on Buckingham theorem [7,8] some of the dimensionless numbers with explicit physical meaning defined in a recent work [2] were applied for a parametric study of the façade. N1 that expresses the average Nusselt number, N2 Reynolds number, N3 Prandt number, N4 Grashof number were calculated for the systems studied, using the transient simulation results:

$$N1 = \frac{qD_e}{\lambda_{aou}\Delta T}; \qquad N2 = \frac{m}{\mu_{aou}D_e};$$

$$N3 = \frac{cp\mu_{aou}}{\lambda_{aou}}; \qquad N4 = \frac{g\beta\Delta T D_e^3\rho_{ach}^2}{\mu_{ach}^2};$$

$$N5 = \beta \cdot \Delta T \frac{H}{D_e}$$

The product between two non-dimensional groups, the aspect ratio (H/D_e), and thermal expansion coefficient β (calculated with the mean value between the air temperature in the channel and that one of the surface of the wall towards the channel) multiplied with temperature difference ΔT between the external and internal air temperature, is strictly connected to temperature distribution and flow regimes inside the cavity. This is referred to the N5 non-dimensional number.

Results show that the naturally ventilated façade located in Treviso is more efficient than that one located in Florence. This is mainly due to the lower temperature differences, between the external covering brick panel and the external air temperature, for Florence in comparison with those obtained for Treviso. As a matter of fact comparing Fig. 1 with Fig.2, it can be seen that temperature distribution obtained by transient simulation results with a slice representation, is based on lower values for Treviso location compared to Florence. This affects the average convective coefficient evaluated by the all simulation results: for Florence location its value is 6 W m^{-2} K^{-1}, and for Treviso its value is 9 W m^{-2} K^{-1}.

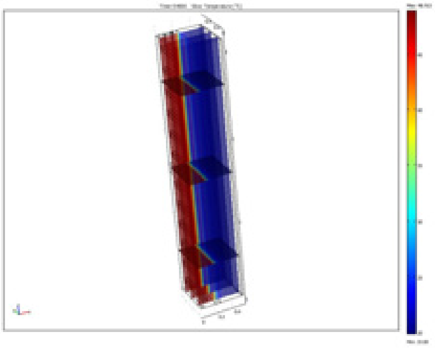

Figure 1. Temperature distribution – slice representation – h 15, Florence.

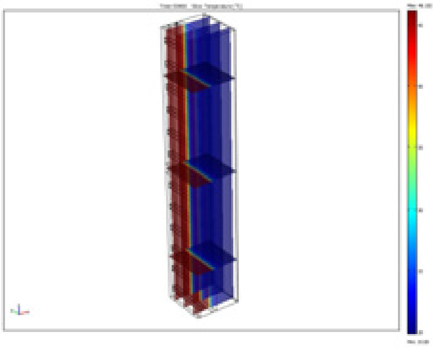

Figure 2. Temperature distribution – slice representation –h 15, Treviso.

Furthermore, this result is particularly evident comparing the air velocity field obtained for the two locations by transient simulation results at h 15 (Figs. 3,4). In particular, comparing Fig. 5 with Fig. 6, where the heat flux outgoing from the air channel is a function of the mean air velocity value distribution inside, a result obtained using the all transient simulation results, it can be deduced: for the Florence location the values are scattered around a regression line with a lower value of the correlation coefficient R^2.

The correlation coefficient obtained for the Florence location is 0.51 and for Treviso is 0.97. This is particular evident comparing the heat transfer throughout the ventilated façade as a function of the non-dimensional number N5, obtained using transient simulation results for the façade located in Florence and the one located in Treviso (Fig.s 7,8).

For the Florence location the correlation coefficient is 0.62, compared to that one for Treviso which is 0.82.

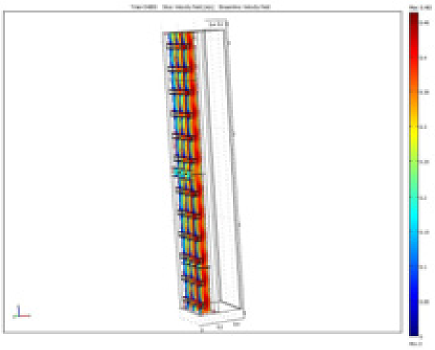

Figure 3. Velocity field - slice and stream lines representation, h. 15 Florence.

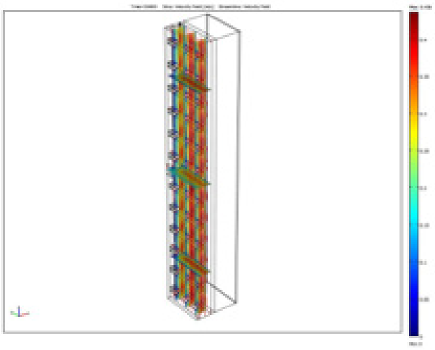

Figure 4. Velocity field - slice and stream lines representation, h. 15 Treviso.

The basic physical mechanisms underlying the complex fluid-dynamics, due to buoyancy-induced non-isothermal flow and heat transfer, were investigated evaluating the Richardson number with the combination between the calculated Reynolds and Grashof using transient simulation results. The changes in the vertical gradient of temperature inside the air channel, contribute to the evolution of the Richardson number profile. Results obtained for the average Richardson number for the two locations show the better performances of the façade in Treviso: for Florence, where the external air temperature is $19.9 < Te < 37.6$ the Richardson number is $0.02 < Rc < 1.20$. For the ventilated façade located in Treviso, where $22.7 < Te < 33.2$, Richardson is $0.49 < Rc < 1.24$. This means that local vertical turbulence phenomena due to the "dry-mounted" fixing structure, are stronger in the façade located in Florence, but at the same time for both the locations the dominant flow regime is low speed natural convection.

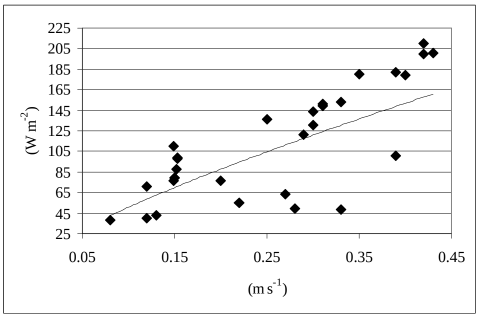

Figure 5. Heat flux outgoing from the air channel as a function of the mean air velocity value distribution inside – Florence.

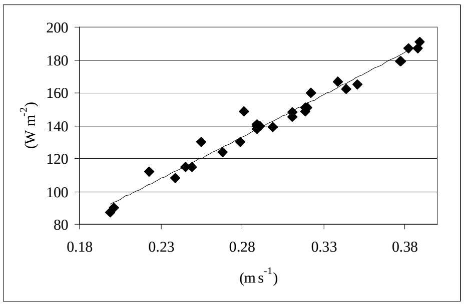

Figure 6. Heat flux outgoing from the air channel as a function of the mean air velocity value distribution inside – Treviso.

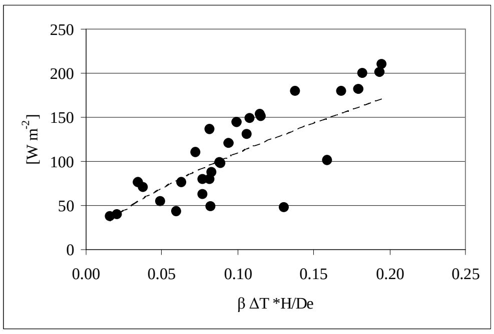

Figure 7. Heat transfer throughout the ventilated façade as a function of the non-dimensional number N5 – Florence.

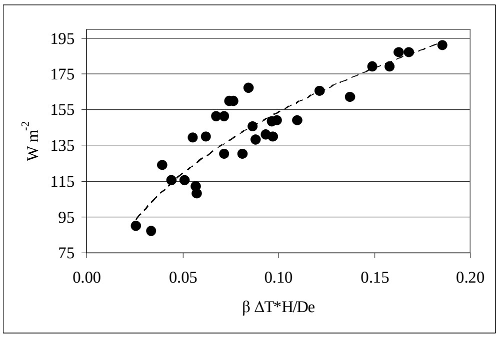

Figure 8. Heat transfer throughout the ventilated façade as a function of the non-dimensional number N5 – Treviso.

5. Conclusions

This study found that the three dimensional CFD-FEM transient simulation, using the Incompressible Navier-Stokes model combined with General Heat Transfer model, performed by Comsol Multhiphysics, is very efficient for simulating buoyancy driven natural ventilation.

Three-dimensional simulation is more realistic than two-dimensional approximation and it may be used to test the Rayleigh and Nusselt values variation with the flow thermal stratification. Furthermore, for the average latitude with rapid variations of the external climatic parameters, unsteady state analysis of the 3D models must be applied. The results obtained by using non-dimensional numbers calculated using the all transient simulation results, might be used to address building sector operators in the development of new design solutions for a climate with high summer temperatures and solar radiation fluctuations. Results of thermal-physics performances of the naturally ventilated façades studied, show that a correct and careful façade design, supported by CFD simulation, can improve cooling energy saving of buildings and at the same time can increase thermal comfort inside. This action would be very important to promote Italian building stock renovation according to the requirements of European Directive 2002/91.

6. References

1. Alabisio M., Sidri R., Gestione archivio Anno Tipo, ENEL CRTN, Milano (1985).
2. Balocco C., A non-dimensional analysis of a ventilated double façade energy performance, *Energy and Buildings*, **Volume 36**, Issue 1, Pages 35-40 (2004)
3. Ciampi M., Leccese F., Tuoni G., Ventilated facades energy performance in summer cooling of buildings, *Solar Energy*, **Volume 75**, Issue 6, Pages 491-502 (2003)
4. Comsol, Comsol Multiphysics 3.5a, COMSOL Lab.
5. Haase M., Marques da Silva F., Amato A., Simulation of ventilated facades in hot and humid climates, *Energy and Buildings*, **Volume 41**, Pages 361-373 (2009)
6. Kim Yu-Min, Kim Soo-Yopung, Shin Sung-Woo, Contribution of natural ventilation in a double skin envelope to heating load reduction in winter, *Building and Environment*, **Volume 44**, Issue 11, Pages 2236-2244 (2009)
7. Langhaar H. L., Dimensional Analysis and Theory of Models, Wiley Ed. (1951)
8. Olsen H.H., Buckingham's pi-theorem, *Mathematical Modelling*, Pages 9-15 (2001)
9. UNI 11018 – 2003, Cladding and anchoring systems for back ventilated external enclosures of buildings - Instructions for the design, installation and maintenance - Ceramic and stone cladding. Pages 1-86 (2003)

9. Acknowledgements

The author is grateful for the support of the COMSOL Centre in Brescia (Italy) concerning the use of the large scale computer in its Computing Centre.