CFD TRANSIENT ANALYSIS OF NIGHT COOLING STRATEGY APPLIED TO SCHOOL BUILDING

M. Simonetti[1†], G. V. Fracastoro[1] and M. Perino[1]

[1]*Department of Energetics, Turin Politechnic,Italy*

ABSTRACT

Application of night cooling to educational buildings looks very promising in mild climates. Night cooling strategy performance in a school building to be realised in Italy during 2008 has been studied by means of a commercial CFD code.

Due to the symmetry of the building, first a 2D numerical model consisting of a vertical section through the three-storey building has been developed. The CFD model includes the solid zone describing the concrete structure. A conjugate heat transfer transient simulation has been implemented to evaluate the natural ventilation flow and the cooling power during a typical mid-season and summer night cooling cycle .

A similar 3D model has been used to obtain a more detailed description of the air distribution in the room and to test the results of different turbulence modeling approaches: Reynolds Average Navier Stokes (RANS) and Large Eddy Simulation (LES).

Some critical aspects of turbulence modeling in buoyancy multi-scale model have been detected and are critically analyzed.

Results obtained predict cooling energy saving up to 50% and will be compared with experimental results obtained from the monitoring of the building, scheduled starting from year 2009.

KEYWORDS

Night cooling, Ventilation, Natural, CFD

DESCRIPTION OF THE SYSTEM

Night cooling is a building management strategy using low-temperature night air to cool the structural elements of the building itself, allowing them to store the thermal loads released by occupants, electric devices, lighting equipment or transmitted through the envelope the day after.

A three-storeys building under construction in Northern Italy, near Imola, has been analyzed as a case study.

The three-storeys building has a central atrium, enclosing the staircase, designed also to foster natural ventilation (1). Rooms are connected to the atrium by a ventilation grille and on the top of atrium some exhaust vents are provided.

Rooms have all-year mechanical air-conditioning, with radiant heating/cooling and ventilation. Free-cooling is provided through earth-air heat exchangers.

A night cooling system with natural ventilation is also designed to reduce daily thermal load. Air inlets consist of mechanically operated tilting windows.

The building is planned for construction within 2 years. A monitoring system will be installed, allowing continuous acquisition of slabs and air temperatures.

[†] Corresponding Author:

E-mail address: marco.simonetti@polito.it

2D MODEL

The CFD simulations have been performed using Fluent, a well known commercial code which is considered suitable for the analysis of these types of systems. The CFD transient model simulates the evolution from initial conditions with 28°C indoor air and slabs temperature and 18°C outdoor air temperature. A 10°C daily temperature excursion is considered typical for mid-seasons in Imola.

The temperature difference is the driving force for natural ventilation and is fully calculated with buoyancy terms in Navier-Stokes equations and ideal-gas model.

Solid zones are included. This is what is called a conjugate heat transfer model, that is, a model in which conduction is coupled with convection and radiation heat transfer.

The starting mesh is composed of nearly 124 000 triangular cells, with areas ranging from $3cm^2$ to $1mm^2$ for the fluid zone and structured rectangular cells, with 1cm steps used to model the 35cm slab thickness, for solid zones.

The three inlet vents, one for each level, are modelled as pressure inlet boundary condition, with gauge pressure zero. Conventionally, Fluent shows gauge pressures, that is, the difference between local pressure and local reference pressure (2). The latter is calculated as:

$$p(z) = p(0) - \rho g z$$

where

p(z) is the local reference pressure value

z is the local vertical coordinate

p(0) is the zero level pressure (absolute reference value)

ρ is the local air density

g is the gravitational acceleration

On the outlet vents, pressure outlet conditions, with zero gauge pressure are applied.

A non-stationary segregated solver is used.

Turbulence is modelled with $\kappa-\omega$ so as to achieve closure of RANS (Reynolds Average Navier Stokes) equations. Radiation heat transfer is taken into account with DTRM (Discrete TRansfer Model). SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithm is used to solve the equations. Derivatives are approximated with 2nd order up-wind scheme (3).

It has been demonstrated that a non-stationary segregated solver with adequate time step gives results very similar to those of a coupled solver, in a simple natural ventilation problem (4). In the same study, different turbulence approaches (RANS, LES) give similar flow rates predictions.

For the present case some simulations has been performed with $\kappa-\varepsilon$, $\kappa-\omega$ turbulence models (5 and 6) and with LES with dynamic sub-grid scale modelling (7), obtaining differences in the range of 5-10% in term of air flow rates. Since the building is modelled with a simplified geometry and only in a typical meteorological condition, expected range is of the same order of magnitude of the errors introduced by the simplifications. Hence, $\kappa-\omega$ is adopted for this work, considering it the best compromise between precision and computational time costs.

RESULTS

Two cases have been analysed. In the first one the same inlet areas are adopted at the three levels; in the second one local pressure losses have been imposed so as to balance the air flow rates on each level.

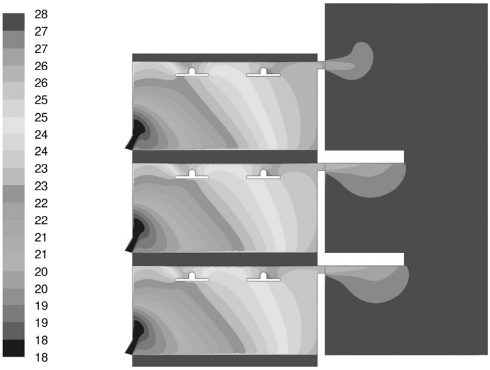

Figure 1. Temperature maps at time=60s [°C]

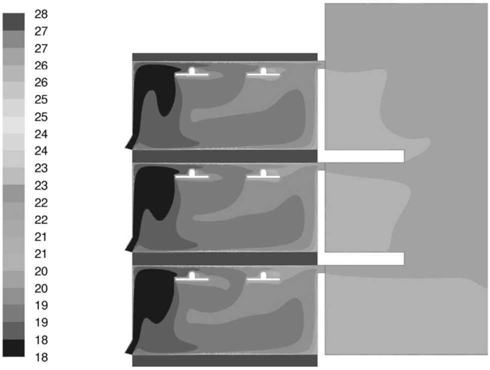

Figure 2. Temperature maps at time=600s [°C]

Cooling effects on structures are derived from application of the 1st principle of Thermodynamics to open systems, calculating enthalpy flow rates. Tilting inlet vents and room grille outlet vents are used as control surfaces.

Figure 4 represents the enthalpy flow rates at each level as a function of time, in kW per meter of depth.

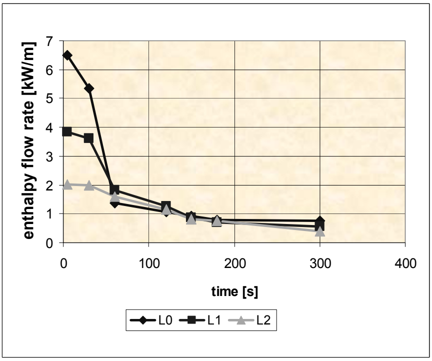

Figure 3. Enthalpy flow rate as a function of time [kW/m] for each level – unbalanced case

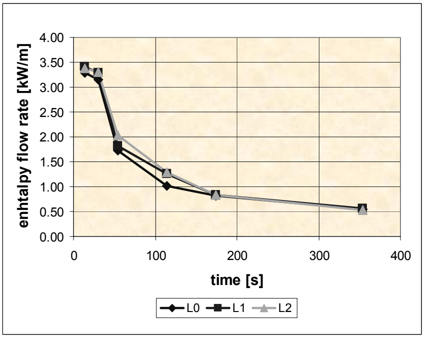

Figure 4. Enthalpy flows rate as a function of time [kW/m] for each level –balanced case

At start pressure differences at the in-out interfaces are very high, due to high density differences between indoor and outdoor air, the latter being at a temperature 10 °C lower. Hence large air flows start to develop (Figure 4 and 5). During the first minute enthalpy flows are very high, but they are not only created by heat transfer with the slabs and walls. There is also a non-negligible effect of thermal capacity term, as initially hot indoor air is gradually exhausted.

As soon as indoor air temperature decreases, being replaced with outdoor, cold air, air flow rates diminish (8).

After 2-3 minutes from start, enthalpy flows show a quasi-stationary decay rate. From this moment on, system dynamics are those of a purely natural convection problem, without the initial surplus of buoyancy effect due to in-out air temperature difference. Air flow rates depend only on slab heat transfer, and this one depends on air flows on its turn.

Balance between these two coupled effects results in an enthalpy flow diminishing as a function of time as quickly as slabs surface temperatures decrease, i.e. as quickly as slabs themselves are cooled.

In the unbalanced case (Figure 3) one can realize that enthalpy flows present very different values in the very first seconds, but quickly converge to a similar trend. That is, the system has a tendency to automatically balance different level heat transfer rates.

This feature can be explained by the commonly used butterfly diagram, in which total pressure versus level are represented. The problem is simplified by assuming a uniform indoor air temperature, hence hydrostatic pressure gradient is represented by a linear function. In real physics, this function is not-linear, but the rationale does not seems to change. As seen in figure 6, the effect of the local pressure losses introduced is to raise the level of the neutral plane, while slopes of the diagrams do not change, for they are only-temperature dependent. Hence, the net pressure head at higher level increases, while at lower levels a significant part of the static pressure head is lost in local frictions.

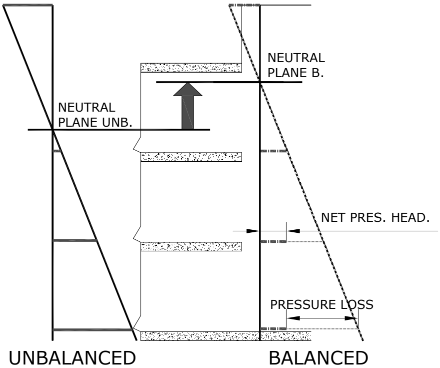

Figure 6. Total pressure (static+dynamic) versus level in unbalanced (left) and balanced (right) case, with indoor air uniform temperature assumption

3D MODEL

The three dimensional model includes the volume of a mid-level room and a portion of the central atrium. The atrium is essentially a stack, whose description is simplified, not considering topological details. Since air velocities in the atrium are very low and associated pressure losses are negligible compared to the hydrostatic gradient, this simplification should not affect the overall flow rates value.
The first run mesh is made up of nearly 750 000 hexa cells, with volumes ranging from 25 cm^3 to 43 000 cm^3.
One tilting windows inlet vent is modelled as pressure inlet boundary condition, with zero gauge pressure (see 2D model). The outlet vents are considered as pressure outlets with zero gauge pressure.
As for the 2D model, a non-stationary segregated solver is used, turbulence is modelled with $\kappa-\omega$,
DTRM is used to deal with radiation heat transfer, a SIMPLE algorithm is used to solve the equations, and derivatives are approximated with 2nd order up-wind scheme.

RESULTS

In the following, temperature coloured maps on a transversal vertical section in the middle of the room and atrium are reported for different time steps. In the same figures surface temperatures of the slab are also mapped.

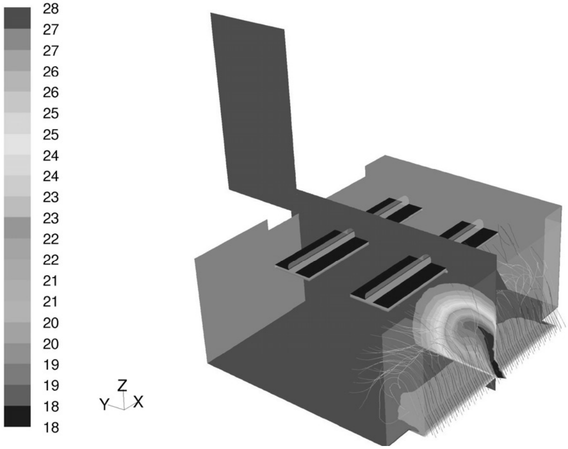

Figure 7. Temperature maps [C°] and streamlines   t = 5s

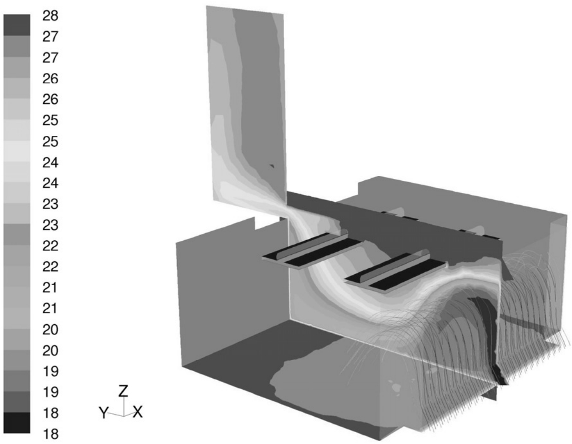

Figure 8. Temperature maps [C°] and streamlines   t = 35s

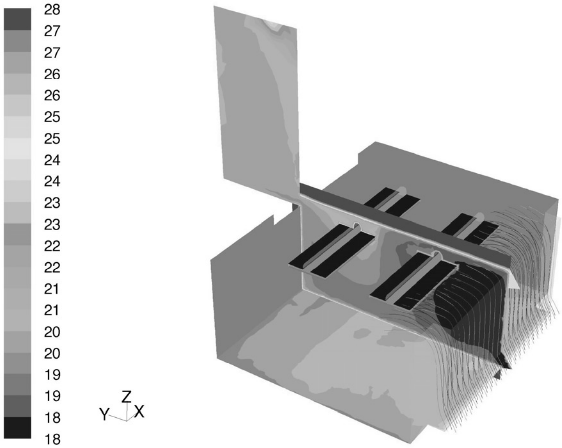

Figure 9. Temperature maps [C°] and streamlines   t = 1h

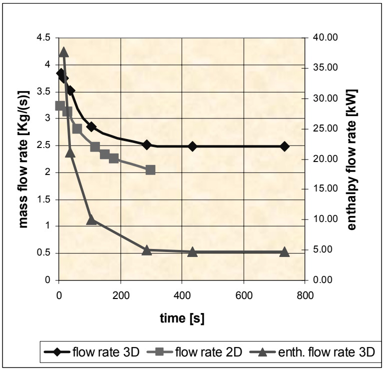

Figure 10. Mass (2D and 3D models) and enthalpy flow rates evolution in the first minutes.

Figure 10 shows that air flow rates differ between 2D and 3D models. One can explain these differences remembering that 2D models uses stack-effect onto three levels, that is the total pressure static gradient for mid-level is influenced and limited by the upper-stage air flow. On the other hand, the 3D model includes heat capacity and heat transfer with the vertical wall, neglected in the 2D one.

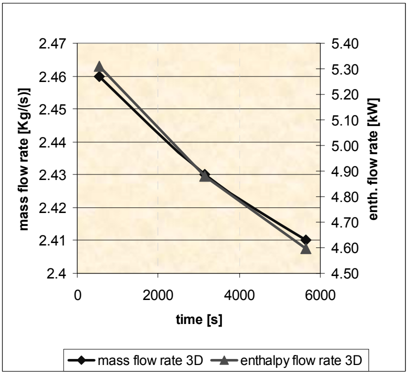

Figure 11. Mass and enthalpy flow rates evolution (3D model) in the longer term

In some minutes from start (figure 11) heat and mass flow rates decay with an almost linear rate, by about 8% per hour.

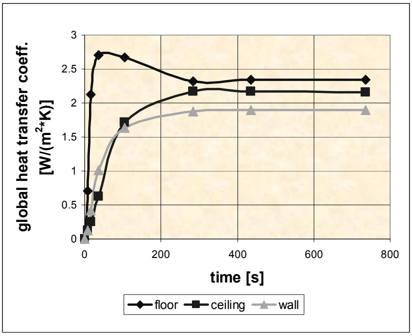

Figure 12. Surface heat transfer coefficients on the floor, ceiling and walls in the first minutes.

In figure 12 average surface heat transfer coefficients on floor, walls and ceiling are reported. As seen in figures 7 and 8 air flux entering from tilting windows, with initial velocity pointing at the ceiling, is bent to the floor by negative buoyancy. Due to this and to the very high air flow rates, during the first minutes (time < 250 s) high convective heat transfer appears on the floor. As soon as surface temperature of the floor lowers, heat is transferred by radiation to the floor from vertical walls and ceiling.
Once the flow stabilizes, heat transfer on all surfaces shows a similar value. All the surfaces take part in the cooling process, a part of them exchanging heat mainly by convection due to air motion, while another part exposed to quiet air acts as a "heat sink", reloading by radiation heat transfer surfaces cooled by convection.

Practical considerations and conclusions

The system is planned to operate mainly during mid season, because the two central summer months are vacation time in Italy.
Thermal energy dissipated during a typical mid season night cooling from 10pm to 5am sums up to

93MJ per room. On the other hand one can estimate that during daytime (8am-2pm) some 98MJ would be released inside the room by internal loads (persons, lighting, electrical equipment, etc.) and solar radiation. Hence the global loads, excluding ventilation, may almost totally be removed only by night cooling.

One could estimate a maximum ideal heat storage capacity, that is clearly the heat capacity of the building structure times an ideally uniform temperature difference. In this paper a 10°C temperature variation as been assumed. The mass of a single room structural elements mass is nearly 57tons, considering wall equally divided between adjacent rooms. The ideal storage capacity sums up to

$$Q_{\max} = m_c c_p \Delta T = 57 \cdot 10^3 [kg] \cdot 800 \left[{}^{J}\!/_{kg \cdot K} \right] \cdot 10[K] = 456[MJ]$$

Hence, the effective heat capacity represents nearly 23% of the maximum ideal one. In other words, only a little portion of the structure is involved in the dynamics, because the thermal wave doesn't reach the deepest layers of the slab. The actual heat capacity could be stored in a 6cm concrete layer. The remaining capacity seems useless for the night cooling strategy. Nonetheless it could play a part on weekly or longer temperature fluctuations, not investigated in this work.

Intensive monitoring of the building will be held starting from the end of 2008.

ACKNOWLEDGEMENTS

This work is part of a research program in collaboration with Mario Grosso, 2nd Faculty of Architecture, Polytechnic of Turin.

Support from Fluent Italy is gratefully acknowledged.

REFERENCES

1. Awbi, H. B. 1991. Ventilation of Buildings. London: Chapman & Hall.
2. Fluent.inc. Fluent 6.2 User's Guide
3. Ferziger, J.; and M. Peric. 1997. Computational Methods for fluid Dynamics. Springer.
4. Simonetti M. 2007 Applicazioni della CFD al progetto dei sistemi di climatizzazione naturale (Applications of CFD to the design of natural air conditioning system), Ph.D. Thesis in Energetics, Polytechnic of Turin
5. Nielsen. P.V. 1998. The selection of turbulence models for prediction of room airflow. ASHRAE Transactions. 104(1B):1119-1127.
6. Schälin, A. and P. V. Nielsen. 2003. Impact of Turbulence Anisotropy near Walls in Room Air Flow. Indoor Air
7. Germano, M., Piomelli, U., Moin, P., Cabot, WH., 1991 A dynamic subgrid-scale eddy viscosity model. Phys. Fluids A 3:1760-65
8. Fracastoro, G.V. Mutani, G , Perino, M. 2002 Experimental and theoretical analysis of natural ventilation by windows opening, Energy and Buildings, vol. 34, issue 8, pp. 817-827, september 2002.