ENVIRONMENTALLY FRIENDLY CITIES, Proceedings of PLEA'98, Lisbon, Portugal, June 1998
Pages 495-498 © 1998 James & James Science Publishers Ltd



# Simulation of the cooling effect of the night time natural ventilation: a 3D numerical application to the " Maison Ronde " of Mario Botta

CHRISTIAN MARENNE*, DOMINIQUE GROLEAU AND FRANCK RAYMOND

Laboratoire CERMA UMR 1563, Ecole d'Architecture de Nantes
Rue Massenet
44000 Nantes, France
marenne@cerma.archi.fr

*Abstract*

The present research applies the N3S 3D finite element CFD code to the air flow simulation on a well-known dwelling building located in South of Switzerland, the " Maison Ronde " of the architect Mario Botta. The summer night refreshing effect is examined when crossing ventilation due to the wind creates indoor air motion and for cooling walls and ceilings surfaces.

To be realistic, the simulation takes simultaneously into account the three main aspects of the problem:

- determining boundaries conditions. To avoid to set artificial or non realistic conditions at openings, the 3D mesh of the house is immersed in a numerical " wind tunnel ", at the frontier of which the wind profile is imposed. The incoming air flow is therefore the result of the outdoor aerodynamic effect of the building geometry and of the location of opened windows;

- simulating the air flow with thermal conditions. The N3S code uses a thermal boundary layer at the walls that enable to compute the mixed convection effect induced by the temperature gradient between the wall and the surrounding air;

- calculating the cooling effect of the air flow on the wall structure. A thermal model is integrated into the CFD code to model the transient behavior of the mass inertia walls.

This paper presents firstly the technical aspects of the implementation of the cooling effect in a summer night ventilation situation and secondly its application to the "Maison Ronde" The simulation evaluates the transient evolution of the air flow, internal air and wall surface temperatures from realistic initial conditions corresponding to the beginning of the night. Obtained results are analyzed in relation to the architectural specificities of the house to explain its thermal behavior and appreciate the night cooling effect.

## METHODS FOR SIMULATING THE COOLING EFFECT ON STRUCTURE

Natural ventilation can play an important role in the control of indoor temperature in summer, preventing overheating and promoting cooling of the structure of buildings. However, highly variable parameters like wind conditions at the openings, indoor air motion and heat exchanges between air and walls make its prediction very difficult and complex. Face to that situation, various ways and methods are used to investigate and understand that phenomenon where thermal conditions and air movement are strongly connected. The first ones are mainly intuitive and empiric; they may characterize the way architects conscious of environmental problems take the cooling process into account, establishing generally with blue and red arrows an hypothetical and optimal air motion that insure a pleasant and economic summer night comfort. The second ones are based on simplified methods that enable the evaluation of the flow rate and air flow direction between rooms at different pressure and temperature. Among these ones, zonal methods [1] which apply mean conditions in different zones of a building are particularly interesting at a scale where the local values and detailed data are not main objectives of the calculation. The third approaches make use of sophisticated models that generally resolve numerically the physical equations involved in the phenomenon.

The present research takes place in this third perspective like several works already carried out at the CERMA laboratory [2][3]. It applies the N3S [4] 3D finite element CFD code to the air flow simulation on a well-known dwelling building located in South of Switzerland, the "Maison Ronde" of the famous architect Mario Botta (Figures 1-2). The summer night refreshing effect is examined when crossing ventilation due to the wind is involved for creating indoor air motion and for cooling walls and ceilings surfaces.

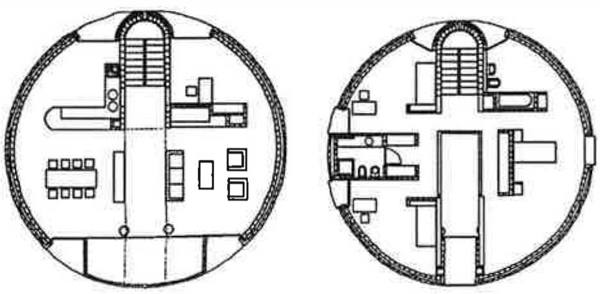

Figure 1. Living room and bedrooms layout

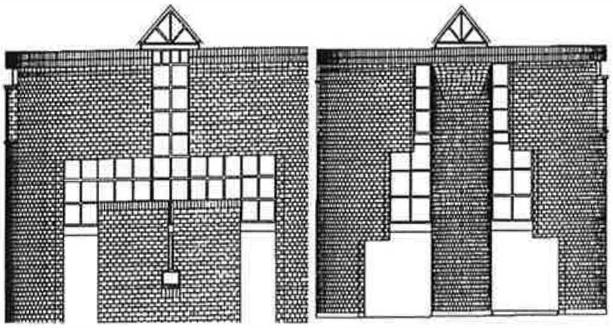

Figure 2. South and North elevations

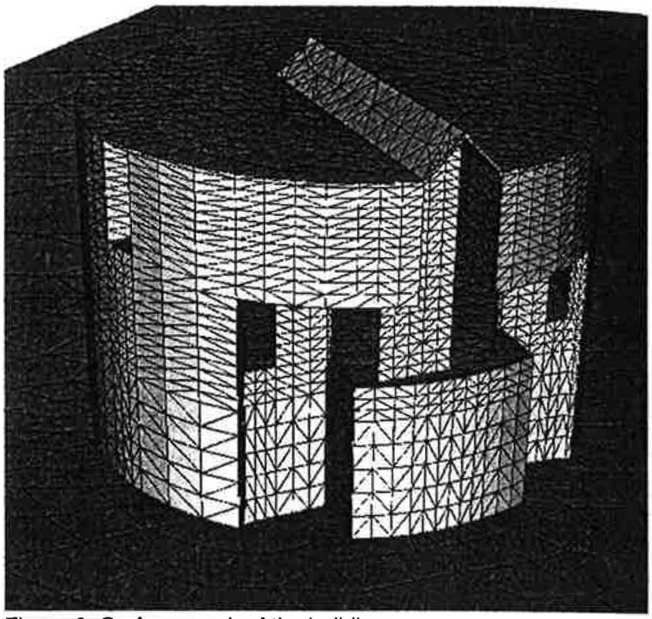

Figure 3. Surface mesh of the building

The problem presents two main difficulties: the first one concerns the correct management of the air motion inside a building taking into account thermal loads and heat exchange at the walls surfaces; the second one is the application of these numerical procedures to a very complex 3D inside building geometry.

These constraints correspond to our will to deal with realistic conditions and to underline, if possible, particularly sensitive architectural and structural parameters on the passive cooling mechanism.

To be realistic, the simulation takes simultaneously into account three main aspects of the problem: modeling the house and determining boundaries conditions, resolving air flow simulation with thermal conditions and finally calculating the cooling effect of the air flow into the structure of the wall and inside the building.

## MODELING THE BUILDING AND DETERMINING BOUNDARY CONDITIONS

The first step is to carry out the volumetric mesh of the "Maison Ronde", relied on the external shape of the building, but integrating also geometric characteristics of the inside layout, with bookcase, parapet and central open space. To apply different boundary conditions, walls of different materials and openings need to be precisely modeled. The house is then put into a numerical wind tunnel at entrance of which a wind profile is imposed. The main advantage of this kind of simulation is that the air and temperature flows through the openings are computed and not imposed arbitrary or according to some simplified hypotheses [5]. The effect of the outdoor building geometry on the indoor flow pattern is then directly taken into account during the simulation.

The 3D mesh of the whole domain contains approximately 120,000 nodes and 60,000 tetrahedra (Figure 3.).

The numerical wind tunnel is a rectangular box in center of which a 3D building model is placed. The house is supposed to be located on a flat ground without natural or built surroundings.

The aerodynamic and thermal inflow conditions are derived from the prevailing wind direction, magnitude and temperature in the "Maison Ronde" location during the summer (wind coming from the south-west direction with a mean velocity of 3 m/s at a height of 10 m ; temperature decreasing linearly from 303 K in the day time to 283 K in the night time). The incoming wind profile is supposed to follow a power law characteristic of an atmospheric boundary layer.

## INTRODUCING A WALL THERMAL MODEL IN THE N3S CFD CODE

The air and temperature fields around and inside the house are computed using the commercial N3S CFD code [6]. The unsteady Reynolds Averaged Navier-Stokes (RANS) equations are solved together with the incompressibility condition. The turbulence of the flow is represented by the standard k-ε model with wall functions while the energy conservation equation is derived under the Boussinesq approximation. The resulting equations are solved using a finite element discretization on an unstructured mesh of the well adapted to complex geometries.

The thermal boundary conditions at the building walls are provided by a thermal model elaborated in this research. It is intended to be simple, not time consuming (it is based on a one dimensional assumption) and general enough to be quickly introduced in any CFD code. The materials that this model can take into account are solid walls with negligible or important inertia and glass windows. The set of equations describing the heat exchange between the solid and fluid parts of the domain are:

$$-h_o(T_{fo} - T_{wo}) = \lambda\left(\frac{\delta T}{\delta x}\right)_o$$

at the outdoor fluid/solid interface

$$h_i(T_{fi} - T_{wi}) = \lambda\left(\frac{\delta T}{\delta x}\right)_i$$

at the indoor fluid/solid interface

$$\frac{\delta T}{\delta t} = a\frac{\delta^2 T}{\delta x^2}$$

inside the wall, where ($h_o$, $h_i$) are the outdoor and indoor heat exchange coefficients ($W/(m^2.K)$), $\lambda$ is the wall conductivity (W/(m.K)), $a$ is the wall diffusivity ($m^2/s$) and ($T_{wo}, T_{wi}$), ($T_{fo}, T_{fi}$) are the wall and fluid outdoor and indoor temperatures (K). The figure 4 shows the wall thermal model notations:

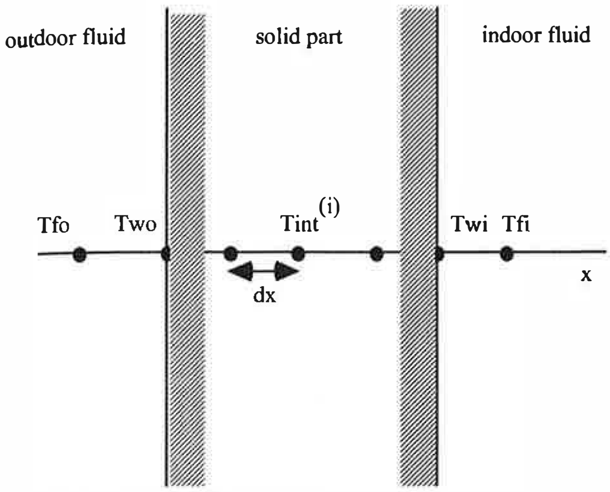

Figure 4. Thermal model notations

Those equations are discretized using a finite difference method: the scheme is fully implicit in order to avoid numerical instabilities when using large time steps. Each wall point neighbor fluid node number and position is found and stored ; the wall-fluid heat exchange coefficient is then computed according to the local wind velocity and temperature. The model has been implemented by means of the set of N3S code user's subroutine and carefully validated against analytical steady and unsteady solutions.

In the night time ventilation simulation, th ee kind of walls are modeled for the "Maison Ronde" building: the concrete walls with strong thermal inertia corresponding mainly to the external skin of the geometry; the inside concrete walls that separate inside rooms, and the closed windows that are the only thermal resistive solid parts without inertia.

## SIMULATING THE COOLING EFFECT BY NIGHTIME VENTILATION

To understand and follow the analysis of the cooling effect in the building, the simulation is carried out in seve al steps.

### *SIMULATING THE AERODYNAMIC FLOW*

A first simulation is realized considering an homogeneous temperature, enabling to observe the aerodynamic air flow outside and inside the building (indoor mean velocity, air flow at openings, level of indoor ventilation, main air patterns), and to establish a reference situation before simulating the cooling effect on the walls of the building (Figure 5). The openings of windows for air inlet and outlet are selected in order to provide maximum aeration of the building. The wind direction does not correspond to an optimized ventilation situation for that building, but constitutes a realistic climatic data for this site.

### *SIMULATING THE COOLING EFFECT*

The second simulation of the air cooling in the building is realized respecting certain theoretical conditions in order to appreciate the time changing conditions. The transient situation starts at an initial time where all the domain (outdoor and indoor air, inside and outside wall surfaces) is supposed to be at a the constant daytime temperature of 303 K. The initial air velocity field are those that are computed in the previous aerodynamic simulation. The changing conditions are introduced by a sudden modification of air temperature at the entrance of the wind tunnel.

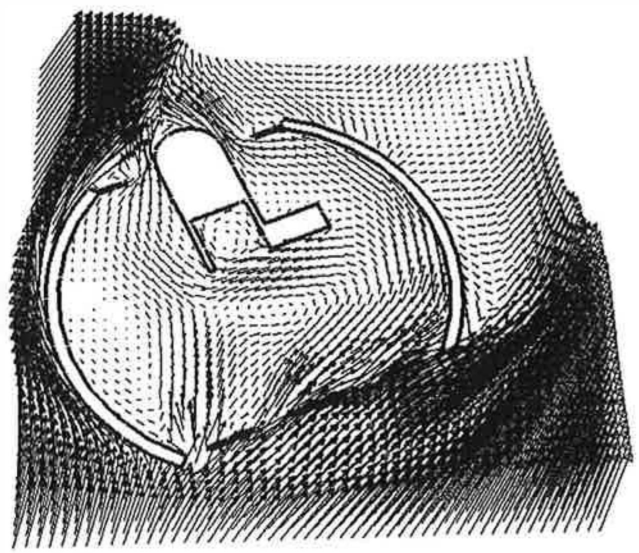
Figure 5. Display of the 3D velocity vectors on the first floor at 1.80m high

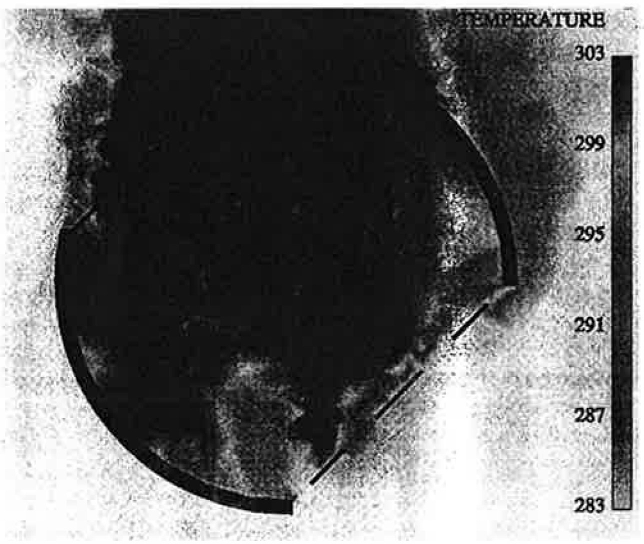

Figure 6. Temperature field – first floor

Figures 6-7 show the temperature field on the first and second floor of the building at a height of 1.50 m above the floor. This situation corresponds to the early stage of the entrance of the fresh air inside the house. The cooling effect is therefore weak and the inside temperature remains still close to the day time condition. On the first floor, the fresh air is mainly convected by the air flow: the right wall expe iences lower temperatures than the left part of the building according to the air flow pattern shown on Figure 5. A strong penetration of cool wind can also be observed at the south opening. A similar effect occurs at the second floor: cool air enters the building th ough the opening that is the most exposed to the wind and reaches the center of the building while the rest of the inside space remains hot.

Those first results allow us to consider the thermal-aerodynamic numerical model developed in this study as operational. The physical time co responding to this repa tition is of 1 mn and the simulation required a la ge computing time. Additional computations are under progress in order to investigate the long range thermal behavior of the house.

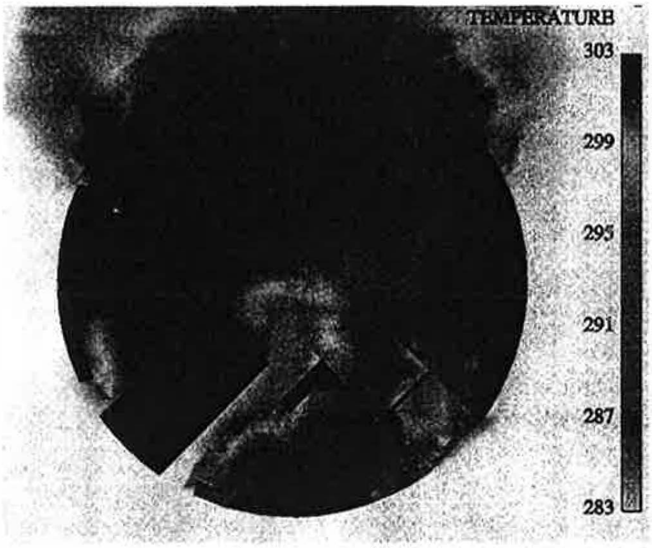


Figure 7. Temperature field – second floor

At regular time intervals, the results of the thermal-aerodynamic simulation will be analyzed and different means of 3D imaging post-processing of the rough data (air vectors, velocities, temperature, turbulent intensity) may be used to put in relief the following points:

- the modification of air pattern due to the temperature,
- the evolution of the air temperature outside and inside the building,
- the effect of the fresh circulating air in cooling the structure of walls.

A more detailed and architectural investigation should enable us to:

- differentiate behavior of cooling according to zones of the building and the flow pattern,
- the cooling response time of the building in relation with changing in air temperature condition,
- the identification of architectural dispositions that interfere greatly with the efficiency of the night time air cooling.

## CONCLUSIONS

Beyond the large computational time and resources required in this study, the obtained results highlight two important aspects:

- on an instrumental point of view, the conjunction of a wall thermal model and a CFD code allows the introduction of realistic thermal boundary conditions in numerical simula-tions. The detailed analysis of the cooling ventilation and its thermal effects on buildings is then greatly facilitated,
- on a practical point of view, the "Maison Ronde" simulation enables us to point out the key parameters (geometric, architectural or structural components) that play an important role in the efficiency of the passive cooling process to improve summer comfort.

## REFERENCES

1. J P. Smith Schneider, J.-J. Roux and J. Brau, Strategies for solving the air flow-thermal problem in multiroom buildings, Building and Environment, Vol. 30, No 2, pp 277-286 (1995).
2. Ch. Marenne, D. Groleau and M. Lefeuvre, Les mouvements d'air et la simulation thermique des ambiances: instrumentation d'une oeuvre architecturale de référence. La Maison de Ronde de Botta, La thermique des systèmes à température modérée, 2ème Colloque Inter-Universitaire Franco-Québécois, Montréal, May 23-25 (1995).
3. D. Groleau, K. Deguchi and Ch. Marenne, Evaluation of natural ventilation in traditional japanese dwellings and its possibilities of transposition, PLEA 97, Jan. 8-10, Kushiro, Japan (1997).
4. P. Chabard and G. Pot, An efficient finite element technique for computation of 3D turbulent flows, Emectricité de France (EDF), Département Laboratoire National d'Hydraulique, France (1989).
5. P. J. Jones and G.E. Whittle, Computational Fluid Dynamics for building air flow prediction - current status and capabilities, Building and Environment, Vol. 27, No 3, pp 321-338 (1992).
6. SIMULOG (société), Manuel Théorique N3S, version 3.1