NUMERICAL SIMULATION OF THE AIR COOLING BY NATURAL VENTILATION INSIDE THE "MAISON RONDE" OF BOTTA

F. Raymond, Ch. Marenne and D. Groleau
Laboratoire CERMA, UMR CNRS 1563, Ecole d'Architecture de Nantes
rue Massenet, 44300 Nantes-France
raymond@nantes.archi.fr

ABSTRACT

The present study applies the N3S CFD code to the air cooling simulation on an "architectural reference object", namely the "Maison Ronde" of Mario Botta. The summer night situation is examined when natural ventilation creates indoor air motion and cools the building structure. The transient behavior of the walls is represented by a thermal model coupled with the CFD code. The simulation evaluates the unsteady temperatures of the outdoor and indoor air flow together with those of the wall surfaces. Obtained results are analyzed according to the architectural specificities of the building in order to identify the key components that play a major role in the efficiency of the passive cooling process.

KEYWORDS

Numerical simulation, models coupling, passive cooling, natural ventilation.

INTRODUCTION

Natural ventilation can play an important role in the control of indoor temperature in summer, preventing indoor overheating and promoting cooling of buildings structures. However, highly variable parameters like wind conditions at the openings, indoor air motion and heat exchange between air and walls make its prediction complex. Three types of methods may be used to investigate those situations where temperature distribution and air flow are strongly connected. The first ones are mainly based on intuition and empirical information : they may characterize the way architects conscious of environmental problems take the cooling process into account. An optimal air flow pattern that insure a pleasant summer night comfort is guessed and generally represented by blue and red arrows. The second ones use simplified assumptions that enable the evaluation of the air flow rate and temperature fluxes between rooms at different pressure and temperature conditions. Among these, zonal methods (Schneider et al. (1995)) which apply mean conditions in different zones of a building are particularly interesting when working at a scale where local values are not the main objectives of the simulation. The third kind of methods make use of more sophisticated models (like Computational Fluid Dynamics) that solve numerically physical equations involved in the problem description and provide more detailed results.

The present research takes place in the last perspective like several works already carried out at the Cerma laboratory (Marenne et al. (1995), Groleau et al. (1997)). It applies the commercial finite element CFD code N3S (Chabart and Pot (1989)) to the simulation of the air flow and temperature distribution on the "Maison Ronde" of the famous architect Mario Botta. This well-known dwelling building located in the south part of Switzerland has been chosen as a "reference object" in order to highlight its architectural properties. The summer night refreshing effect is examined when crossing ventilation is used to create indoor air motion and to cool walls and ceiling surfaces. This problem presents two main difficulties : the first one concerns the coupling between the air flow and the wall temperature distribution, the second one is related to the application of these numerical procedures to a complex 3D inside building geometry. These constraints steam from our will to deal with realistic physical and geometrical conditions with the aim of underline particularly sensitive architectural and structural parameters on the passive cooling process.

GENERATION OF THE NUMERICAL MODEL

The first step of this study consists in providing a numerical model of the "Maison Ronde" and its environment as an input for the N3S code. This description integrates the external shape of the building but also the inside geometric layout including bookcase, parapet and central open space. The model is then put at the center of a numerical wind tunnel. The main advantage of this type of methodology is that the air and temperature flows through the building openings are computed and not imposed arbitrary or according simplified assumptions. The effect of the external building geometry on the indoor flow pattern is then more accurately taken into account in the simulation. The 3D mesh of the whole computational domain contains approximately 120000 nodes and 60000 tetrahedra : a close view of the external building model surface is presented on Figure 1. The aerodynamic and thermal inflow conditions are derived from the main meteorological conditions observed at the "Maison Ronde" location during the summer. The wind is supposed to come from the south-west direction with a mean velocity of 3 m/s at a height of 10 m and the temperature decreases 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.

DESCRIPTION OF THE WALL THERMAL MODEL

The air and temperature fields around and inside the building are computed by the N3S CFD code. The unsteady Reynolds averaged Navier-Stokes equations are solved together with the mass conservation condition. The flow turbulence is represented by the standard k-epsilon model with wall functions while the energy conservation equation is derived under the Boussinesq approximation (Chabart and Pot (1989)). The resulting equations are solved using a finite element discretization on an unstructured mesh well adapted to complex geometries.

The thermal boundary conditions at the building wall surfaces are provided by a model developed in this research. It is based on a one-dimensional assumption in order to keep it simple and general enough to be incorporated in any CFD code. It should be noticed that the wall thermal model application costs less than 1% of the total computing time. The set of equations describing the heat exchange between the solid and fluid parts of the domain (1) and inside the wall (2) are :

$$\pm h(T_{fluid} - T_{wall}) = \lambda(\frac{\delta T}{\delta x}) \tag{1}$$

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

where h is the heat exchange coefficient (W/m^{2}K), λ is the wall conductivity (W/mK), a is the wall diffusivity (m^2/s), t is the time (s) and (T_{fluid}, T_{wall}, T) are the fluid, wall surface and wall inside temperatures (K). Figure 2 shows the corresponding wall thermal model notations. Three kind of solid parts are used in this study : concrete walls with large thermal inertia corresponding to the external building skin, inside concrete walls that separate inside rooms and closed windows that are considered as thermal resistive parts without thermal inertia.

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 fluid / solid exchange coefficient is then computed according to the local wind velocity. The model has been implemented by means of the N3S user's subroutines and carefully validated by itself against analytical steady and unsteady solutions (Raymond (1998)).

SIMULATION OF A REFERENCE AIR FLOW PATTERN

A preliminary aerodynamic simulation is realized using a homogeneous temperature inflow condition, enabling to observe the air flow pattern outside and inside the building. This operation helps to define a reference situation before simulating the cooling effect. The openings for air inlet and outlet are selected in order to provide a strong crossing ventilation of the building.

The air flow pattern on the first floor (living rooms) is shown on Figure 3 : air penetrates the house by the balcony openings and then follows mainly two paths, dividing the first floor in two regions. The first path corresponds to relatively high velocities because air comes into the building directly from the more exposed opening and almost cross the building : the flow acceleration creates a large vortex that protects the west wall from being well ventilated. Because of the cylindrical external shape of the house, air penetrating by the other openings experiences a weaker velocity : the east wall is nevertheless well ventilated because it acts as a guide for the incoming fluid. Small vortices are also created at the center of the first floor near the central open space.

The situation is more complicated on the second floor (bedrooms) because of its more complex spatial organization (Figure 4). Air comes from the outside by the west opening and in a smaller proportion from the first floor by the central open space. An inside separating wall deflects the incoming flow and induces several paths to the outlet openings. The flow velocity is generally smaller than on the first floor, preventing the occurrence of uncomfortable sensation for people sleeping in the bedrooms.

INVESTIGATION OF THE PASSIVE COOLING EFFECT

The simulation of the air cooling of the building is realized under idealized inflow conditions in order to emphasize the effects of the time changing thermal conditions. The initial state corresponds to the velocity field presented in the previous section of this paper : the fluid and solid temperatures are supposed to be homogeneous at T = 303 K. The unsteady thermal inflow conditions are introduced by a sudden decrease of the air temperature to 283 K at the numerical wind tunnel entrance. The results presented in this section correspond to the early cooling process until a time of 400 s. The thermal behavior of the walls is monitored by selecting a few control points at specific locations inside the building. The cooling process of the walls can be decomposed in two steps : firstly cool air comes suddenly on the walls and the surface is quickly refreshed (time between 0 and 40 s), then the wall mass inertia compensate the temperature gradient between the fluid and the solid parts and the cooling effect is slower.

Figure 5 shows the temperature evolution of two points located on the west (point 1) and east wall (point 2) on the first floor. As expected from the analysis of the reference air flow pattern, the cooling effect is much more effective at point 2. After the initial sharp temperature decrease, the wall structure gives energy to the neighbor fluid and elevates its temperature quickly : this phenomenon is the cause of the oscillations observed at t=40 s. The temperature evolution at point 1 is smoother because cool air has to be mixed with hot fluid by a vortex before incoming on this surface. The cooling effect is less pronounced on the second floor because of the smaller fresh air amount coming from the outside and of the complicated air flow pattern. Air coming from the first floor through the central open space is still warm and therefore its contribution to the early cooling process remains small. The increase of the surface temperature at points 3 and 4 (figure 6) is caused by the response of the wall structure to a quick cooling, but oscillations are damped because of the vortices observed on the reference air flow pattern. Generally, the temperature is more homogeneous on the second floor than on the first floor but decreases slowly (points 5 and 6). This effect prevents the occurrence of sharp temperature gradients inside the sleeping rooms and thus improve the summer night comfort.

CONCLUSIONS

Beyond the large computational time and resources involved in this study, presented results highlights two important facts :

- on an instrumental point of view, the coupling of a CFD code with a wall thermal model allows the introduction of realistic thermal boundary conditions in numerical simulations. The detailed analysis of the air cooling and its effects on the building structure is therefore facilited,
- on a practical point of view, the choice of the "Maison Ronde" as an application case enables us to point out the key parameters (geometrical, architectural and structural components) that play a major role in the efficiency of the passive cooling process. It should be then possible to simplify the inside geometry in order to reduce the required computing time : this model reduction will help us to study the long range thermal behavior of the building structure and the differences in the wall cooling processes related to the surrounding air flow pattern.

REFERENCES

Chabart J.-P. and Pot G. (1989). An efficient finite element technique for computation of 3D turbulent flows, Research Report, Electricité de France (EDF), Département Laboratoire National d'Hydraulique, France.

Groleau D., Degushi K. and Marenne Ch. (1997). Evaluation of natural ventilation in traditional japanese dwellings and its possibilities of transposition. PLEA 97 Proceedings, **2**, 015-020.

Jones P.J. and Whittle G.E. (1992). Computational fluid dynamics for building air flow prediction - current status and capabilities. Building and Environment, **27 : 3**, 321-338.

Marenne Ch., Groleau D. and Lefeuvre M. (1995). 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.

Raymond F. (1998). Introduction d'un modèle de comportement thermique de paroi dans N3S. Research Report, Laboratoire Cerma.

Smith Schneider J.-P., Roux J.-J. and Brau J. (1995). Strategies for solving the air flow-thermal model in multiroom buildings. Building and Environment, **30 : 2**, 277-286.

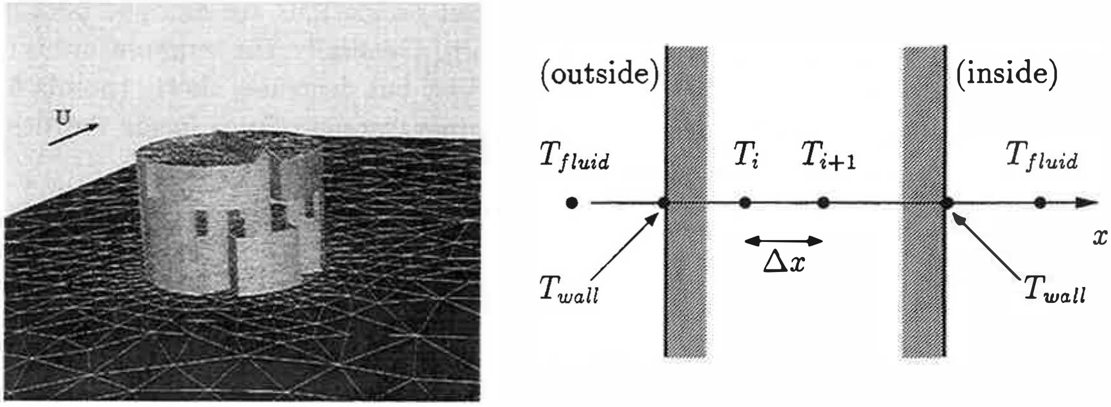

Figure 1-2 : Surface mesh of the building / Wall thermal model layout

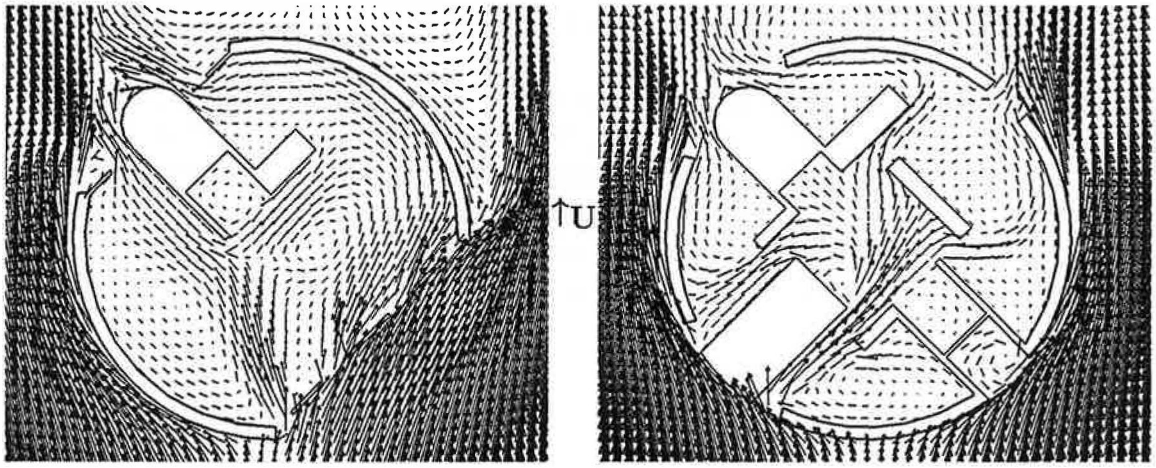

Figure 3-4 : Air flow pattern first / second floor (1.8m height)

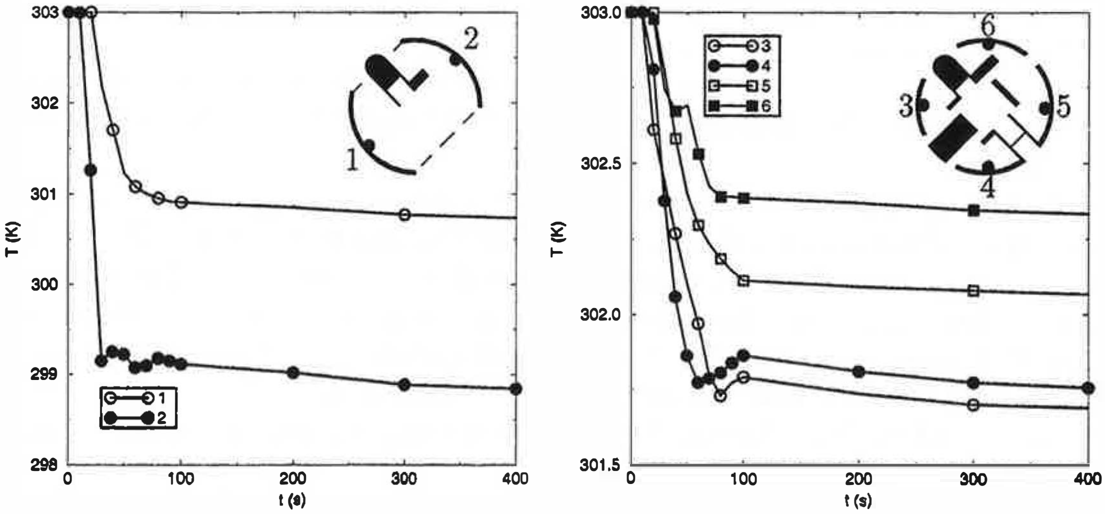

Figure 5-6 : Surface temperature evolution first / second floor