NUMERICAL SIMULATION OF TRANSIENT EFFECTS OF WINDOW OPENINGS

G V Fracastoro & M Perino
Dept. of Energy Technology
Politecnico di Torino
Corso Duca degli Abruzzi, 24 – 10129 Torino, Italy
fax: +39 11 564 4499
e-mail: fracastoro@polito.it, perino@athena.polito.it

ABSTRACT

The simulation of room airing (ventilation by means of door/window opening) by means of CFD techniques requires a specially skilled user, because a number of difficulties arise since the first stage of simulations development, when the user is asked to choose the calculation domain and the time step, and choices which in principle appear correct may frequently lead to meaningless results.
This work is centered on the 2D, transient analysis of a single side enclosure where the ventilation is only due to temperature differences. Wind effect has not been taken into consideration. Different runs have been performed varying: boundary conditions, window sizes and calculation domains. Field model results have been compared to lumped parameter and zone model analyses. A check on conservation principles has shown that CFD results are affected by noticeable inaccuracies for what concerns the prediction of both air temperature and ach's, which may be partially overcome re-scaling the time dependence of the phenomenon.

1 INTRODUCTION

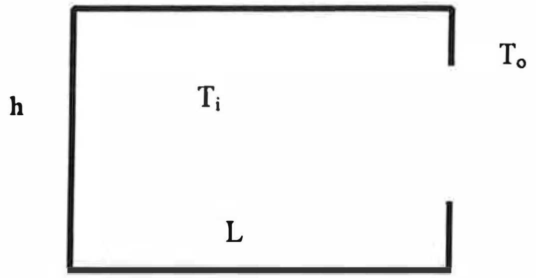
A detailed literature review developed during the fact-finding phase of the Annex 35-HybVent activity has pointed out the difficulties that arise when CFD technique is used to simulate naturally ventilated systems. This specially applies to airing (ventilation by means of door/window opening) (Schaelin et al, 1992, Elsayed, 1998), a simple action which produces qualitatively well known effects.
However, difficulties start since the first stage of simulations development, when the user has to choose the structure of the calculation domain. An apparently reasonable choice may, in fact, lead to surprisingly meaningless results under the physical point of view (e g, cold air entering the room through the upper part of the window and warm air exiting from below), while residuals values would suggest a successful simulation.
Furthermore, the use of simplified models and equations (see for examples Etheridge et al., 1996, Andersen, 1996, ASHRAE Handbook of Fundamentals, 1997, Agnoletto et al. 1981) is usually straightforward, but the user must provide one ore more "empirical" coefficients, whose value is not always known a-priori and may depend on the type of the opening as well as the temperature difference. Moreover, the phenomenon is, by its nature, unsteady and hence the value of temperature difference to be used in these formulas has to be forecasted as an average between initial and final conditions.
In this frame it has been decided to develop a CFD model and investigate the possibility to express the results in concise terms making use of non dimensional quantities such as Grashof number.

2 MODELS FEATURES

2.1 CFD Models

A two dimensional CFD transient analysis has been performed for a natural single side ventilated enclosure using a well known commercial software (FLUENT ®). Only thermal effects have been taken into account and therefore the wind speed has been assumed equal to zero.

In order to avoid difficulties in the description of the domain, the geometry of the room has been assumed very simple (see sketch below).



Two different CFD models have been implemented.

Both have been discretized using a non uniform grid made of 200 x 200 cells, but in the first one only the indoor environment is included (window 1), while in all the other models a strip 2 m wide of outdoor environment has been modelled.

A total of 5 runs have been performed varying window height and temperature differences. Table 1 reports the characteristics of the simulations.

The initial air temperature has been always taken equal to 20° C. The same value has been adopted for the wall temperature, considered constant.

Since the phenomenon is dominated by the buoyancy effect, the Grashof number has been identified as the relevant independent variable. It has been calculated as:

$$Gr = \frac{g \cdot \beta \cdot \Delta T \cdot H^3}{\nu^2}$$

Where ΔT = Temperature difference between wall and outdoors and H = window height.

Table 1 – Model features and boundary conditions.

Model	Outdoor env.	Window height [m]	ΔT [°C]	Grashof Number
Window 1	Not included	1.5	20	$1.17 \cdot 10^{10}$
Window 2	Included	1.5	20	$1.17 \cdot 10^{10}$
Window 3	Included	1.89	10	$1.13 \cdot 10^{10}$
Window 4	Included	1.5	10	$5.64 \cdot 10^{9}$
Window 5	Included	1.5	5	$2.77 \cdot 10^{9}$

The following assumptions were adopted:

- Turbulence model: standard k-ε
- Interpolation scheme: power-law
- Wall functions: standard log-law
- Transient analysis: variable time steps (values from 0.5 s up to 60 s).
- In the first 20-30 s of simulation, time steps larger than 0.5 s lead to numerical instability.
- Number of iterations per time-step: 1000
- Total number of simulated time-steps: about 100 (equivalent to a time span of about 600 s, for each of the simulated configurations for window 2 and 3, and about 150 s for window 4 and 5).
- Computational time: about 3 weeks (for each simulation).
- Hardware: HP Apollo 720 RISC WS (54 Mb RAM memory)

The solution phase is critical due to numerical instability problems and requires particular care. Moreover, as follow from the data listed above, it requires long time and resources.

2.2 Engineering Models

A single-zone model has been developed based on the formula reported by ASHRAE (1997) coupled with the conservation equation for energy:

$$h_p \cdot A_p \cdot (T_p - T_i) = \dot{m} \cdot c_p \cdot (T_i - T_o) + \rho_i \cdot V \cdot c_v \cdot \frac{\partial T_i}{\partial \tau}$$

$$\dot{m} = \rho_o \cdot A \cdot C_d \cdot \sqrt{\frac{g \cdot H \cdot (T_i - T_o)}{T_i}}$$

where the discharge coefficient C_d is given by:

$$C_d = 0.40 + 0.0045 \cdot |T_i - T_o|$$

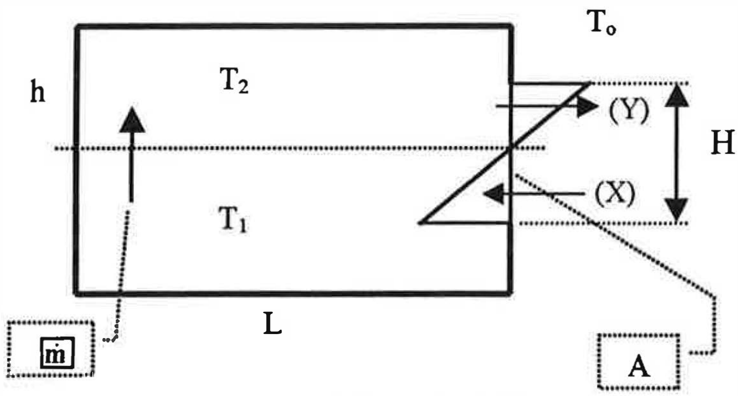
The two-zone model is described by the following equations

$$h_{p1} \cdot A_{p1} \cdot (T_{p1} - T_1) = \dot{m} \cdot c_p \cdot (T_1 - T_o) + \rho_1 \cdot V_1 \cdot c_v \cdot \frac{\partial T_1}{\partial \tau}$$

$$h_{p2} \cdot A_{p2} \cdot (T_{p2} - T_2) = \dot{m} \cdot c_p \cdot (T_2 - T_1) + \rho_2 \cdot V_2 \cdot c_v \cdot \frac{\partial T_2}{\partial \tau}$$

$$\dot{m} = A \cdot \sqrt{\frac{g \cdot H \cdot \left(\rho_o - \frac{\rho_1 + \rho_2}{2}\right) \rho_o \cdot \rho_2}{\beta_1 \cdot \rho_2 + \beta_2 \cdot \rho_o}}$$

The meaning of the symbols is shown in Figure 1 and in the following list:



A = half window surface
A_p = wall surfaces
h_p = film coefficient
c_v, c_p = heat capacities (constant volume/pressure)
T_p = wall temperature
T = air temperature
V = volume,
β = pressure loss coeff.
Subscripts
o,i,1,2 = outdoor, indoor, zone 1,2

Fig. 1 – Two-zone model – calculation scheme.

The value of $\beta_1 = \beta_2$ has been determined assuming the initial air flow rate to be equal to the initial flow rate of the single-zone model. The relation for the air mass flow rate has been derived integrating the energy conservation equation along the air flow path from outside (X) to outside (Y). It has been assumed that no air short-cut takes place between the indoor air "exhausted" from the window and the outdoor air entering the room. Furthermore, for the two zone model the air flows only from the lower zone (1) and the upper zone (2), without internal recirculation.

These two models have been solved numerically, discretizing the ODE's and employing an explicit time integration. The model is implemented by means of a spread-sheet software (Excel ®). No particular problems of numerical instabilities have been encountered. However, in order to achieve accurate solutions (with precise energy and mass balances) quite small time steps have to be used, specially in the first part of the simulation (0.1 s for the first 8 seconds, larger, and increasing, time steps for the following time).

Furthermore, in the case of single zone model, Simulink ® (a MatLab toolbox for dynamic system simulation) was also used for the model simulation. This tool, in fact, might show itself extremely useful in the HybVent system analysis, as it allows an easy coupling of different phenomena calculating quantities such as flow rates, pollutant concentration, temperatures, and introducing also the control strategies. At this stage of development only the flow rate model has been implemented. In all the tested cases, the computational time required for the solution (using a PC Pentium) is less than 1 s.

3 RESULTS

The use of engineering models is simple and straightforward. The only "innovation" introduced by the authors consists in the unsteady state application of formulas expressing the air mass flow rate, $\dot{m}$. The air flow rate, in fact, is evaluated at each integration time step, adopting for the calculation the previous value of air temperature.
The profiles of ach's and air mean temperature versus time (not shown here for brevity) obtained by means of single- and two-zone models[1] are quite similar. The use of other formulas (Etheridge, 1996) for $\dot{m}$ has produced little differences in the final results. In the same way results obtained by means of Excel software are identical to those obtained adopting the Simulink model.
Table 2 resumes the indoor air mean temperature and the ach's values when steady state conditions are practically reached (and the corresponding time required) determined by means of engineering models.
For what concerns the CFD analysis, as mentioned in the introduction, the choice of the geometrical domain has revealed to have a paramount effect in the reliability of the results. Actually, the results of model *window 1*, for which the outdoor environment has been modelled by means of proper boundary conditions (fixed pressure boundaries) are completely meaningless. After a few seconds when the air, as expected, flows from outdoors to indoors in the lower part of the window and vice versa in the upper part (although the neutral level appears to be strikingly low), there is an inversion of the air path and the warm air starts to flow from indoors to outdoors in the lower area of the window, contradicting the common experience. This simple example is a further evidence that CFD analyses choices which appear straightforward (specially for non expert users) may lead to wrong conclusions, even when the numerical indicators (residuals) assume satisfactory results. Being meaningless, the results related to model *window 1* will not be included in the following figures.
Results related to models *window 2* through *5,* are consistent with the expected air flow behaviour. The analysis of air velocity and mass flow rate profiles along the window height shows quite symmetrical trends that develop since the first seconds of simulation. In the first time steps the profile is slightly irregular (and the global mass balance of the rooms is not perfectly satisfied), but after about 2 seconds the curves are smooth and the mass balance is, practically, perfect. Figure 2 shows for the model *window 4*, as an example, the profiles of mass flow rate across the windows at different time steps.
In figure 3 the air changes per hour are plotted versus time for the different models. It must be underlined that the adopted model is 2-D, thus the room depth is assumed to be equal to 1 m. Consequently, the room volume V, adopted for the ach's calculation, is: V = 4.2 m x 2.7 m x 1m = 11.34 m^3.
From a physical point of view the system behaves as if it were an L x h enclosure of infinite width with a continuous "strip" window. The results are strictly applicable only to this type of window, but they may probably be extended to other configurations whether the edge effects of the window sides are negligible (i e, not too high values of ratio height to width). The effect of window to room width ratio is not known a priori and should be investigated.
In order to obtain the actual value of ach related to a particular window and room width, one must multiply the values shown in the following figures by the ratio of window width to room width. Figure 4 shows the air temperature (room mean value) profile versus time. In these charts 8 curves are plotted: four refer to CFD results, four refer to the two-zone model simulations. It is possible to see that there are large discrepancies between the results obtained by means of the two different classes of models. The analysis of the energy balance at each time step has revealed quite large errors in the case of CFD models. This appears unexpected; in fact, during the solution phase of all the models the residuals related to enthalpy were quite low. In figure 5 is possible to see the entity of power imbalance a t various time steps (curves with symbols – refer to the main axis on the left

[1] In the two-zone model the room mean temperature is determined as the simple average between zone 1 and 2 air temperatures.

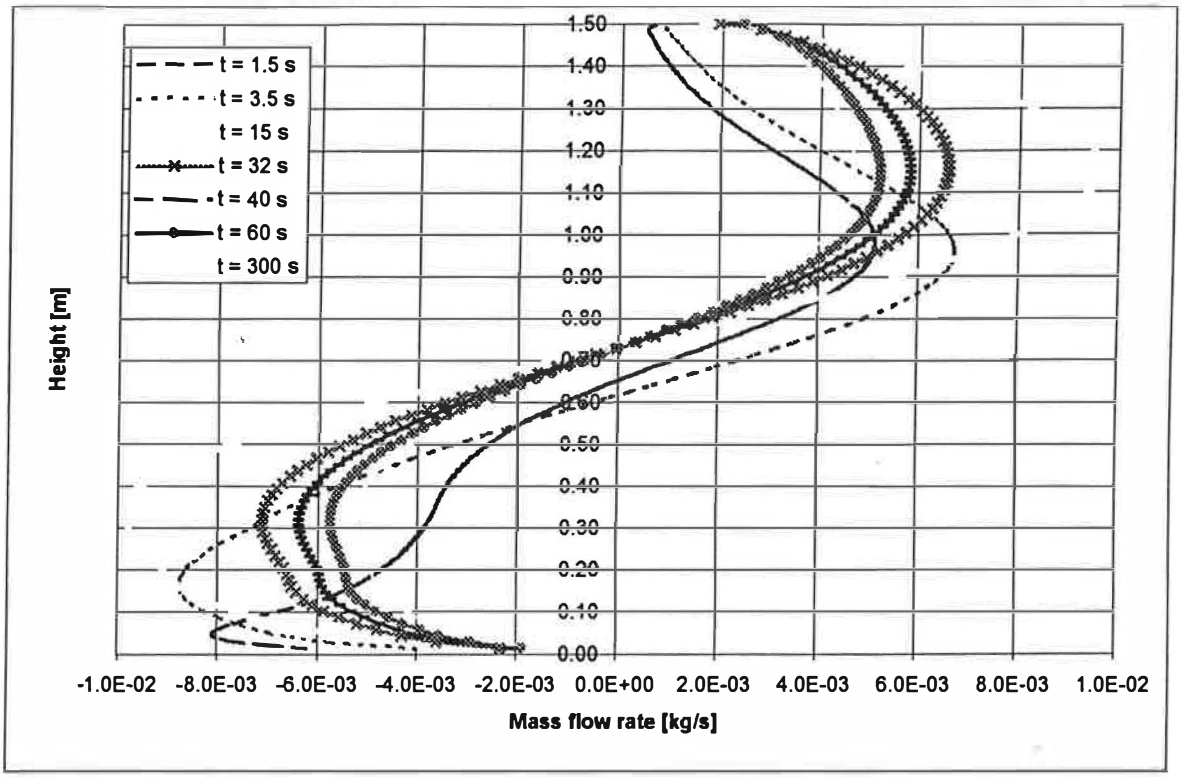


Figure 2 –Temperature profile along the window height for different time steps - CFD results – *Window 4.*

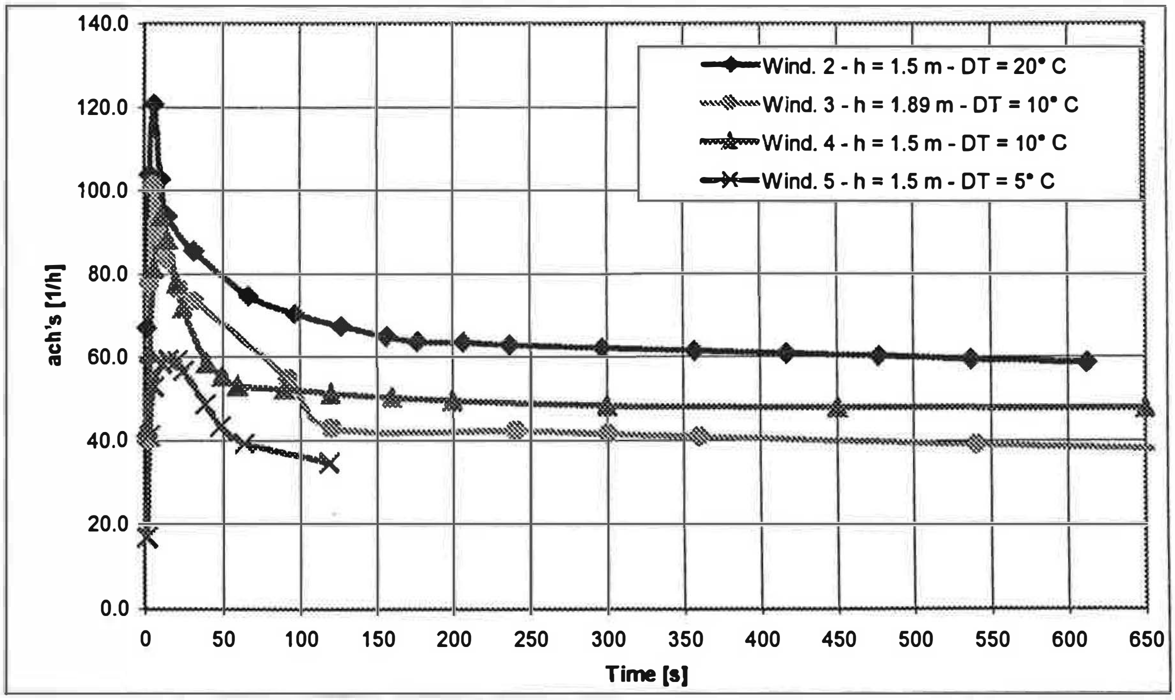
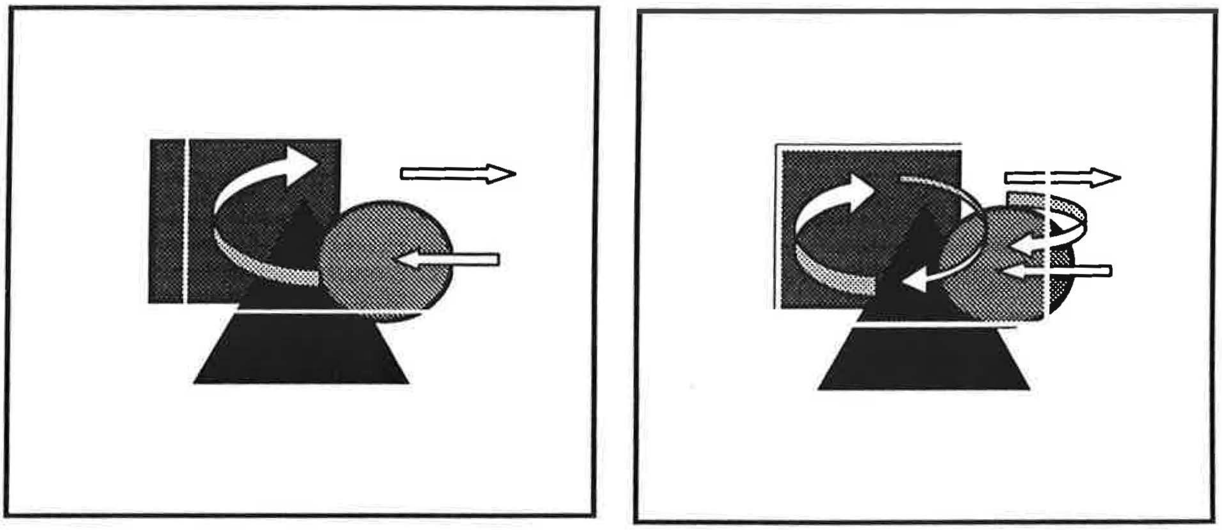


Figure 3 – CFD results – Time history of air change rate.

of the chart) and the relative error defined as: $E = \frac{(\text{Power imbalance}) - \dot{H}}{\dot{H}} \cdot 100$ (symbols - refer to the secondary axis on the right of the chart).

An off-line study of each term of the energy balance equation, performed by means of simplified analytical calculations, has pointed out that the components of the energy balance due to heat fluxes exchanged between walls and air and to enthalpy fluxes are apparently well predicted by the code. Instead, the time variations of internal energy (i e, temperature) seem to be largely underestimated. Everything happens as if the time step intervals adopted for the numerical solution procedure do not coincide with the actual time scale of the phenomenon. Starting from these remarks it has been decided to re-scale all the CFD numerical results on the basis of time steps, $\Delta\tau_r$, obtained imposing the energy balance at each time step: $\Delta\tau_r = \frac{\rho \cdot V \cdot c_v \cdot \Delta T}{\dot{Q} - \dot{H}}$.

Figures 6 to 9 show the results of such a procedure in terms of ach's and mean air temperature versus time. It is possible to see a general substantial improvement in the predictions, with a good agreement with profiles calculated by means of zonal models, particularly for what concerns *window 2, 4 and 5* models. In the case of *window 3* model, instead, the performances of the procedure seems to be worst. However, in this last simulation substantial re-circulation of warm exhaust air with fresh entering outdoor air occurs. Due to this mixing, the air actually entering the enclosure has a temperature slightly higher than the outdoor air. This phenomenon is clearly shown by the CFD simulation temperature fields of which figure 10 is an example (fig. 10a – *window 2* model, fig.10b *window 3* model. Time: about 22 s after the window opening).



(a) (b)

Figure 10 – Temperature field after 22 s and main air flow paths – Model window 2 and window 3

The lower temperature differences between indoor and outdoor induce a smaller air flow rate. It follows that the time profile of air mean temperature predicted by CFD calculation differs from that obtained by means of the two-zone model.
From a physical point of view this behavior is probably linked to the fact that in *window 3* model (that from a theoretical point of view should present the same ach's of *window 2*, since both cases have the same Gr number), the air velocities through the larger window are quite low and are influenced by the great clockwise vortex that takes place inside the room due to indoor thermal gradients (while for window 2 the initial vortex is destroyed by the stronger air current that flows in and out the room). Such kind of phenomenon could never be predicted by zonal models, as they assume a priori no re-circulation and mixing.

Table 2 – Steady state values – zone models

Model	n [1/h]	T [°C]	n [1/h]	T [°C]	τ [s]
	Single zone		Two zone		
Window 2	35.4	4.7	37.0	3.7	≈150
Window 3	34.8	12.5	37.1	11.9	≈150
Window 4	27.1	13.0	29.0	12.3	≈220
Window 5	20.8	16.8	22.6	16.4	≈220

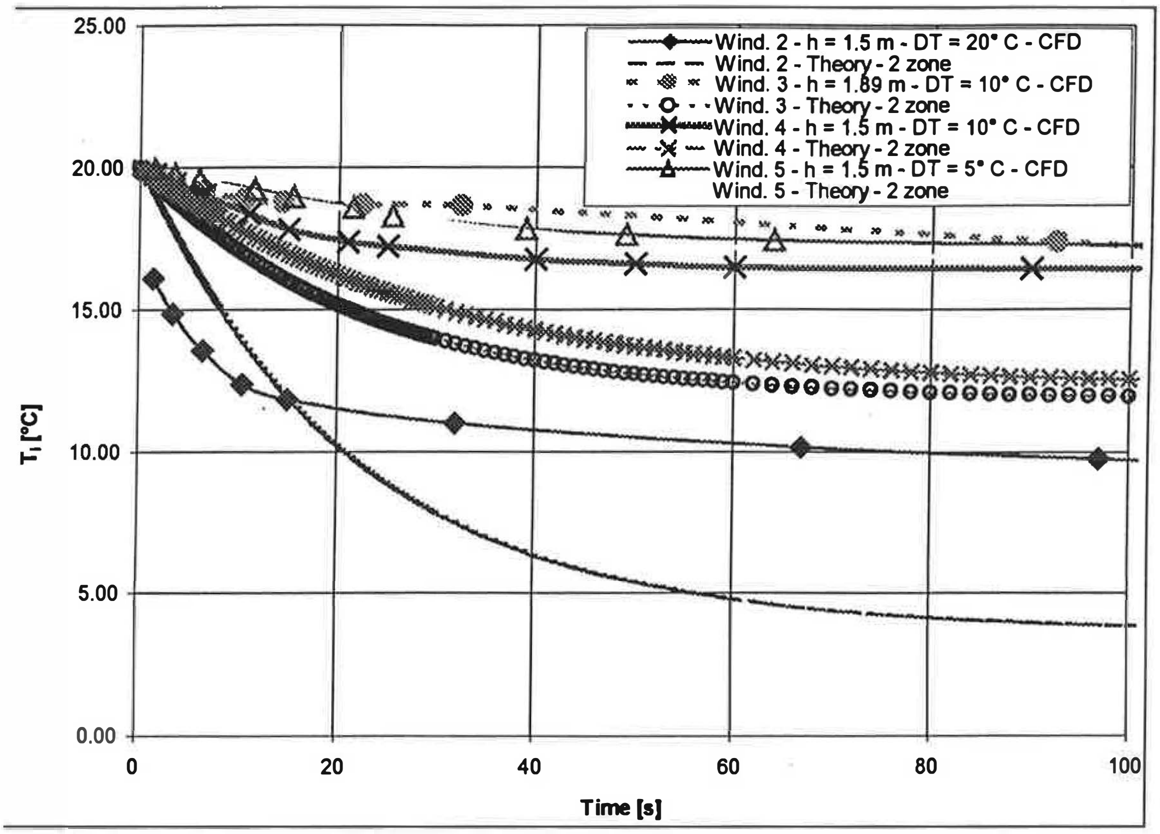


Figure 4 – Air temperature (room mean values) versus time (CFD and 2-zone model).

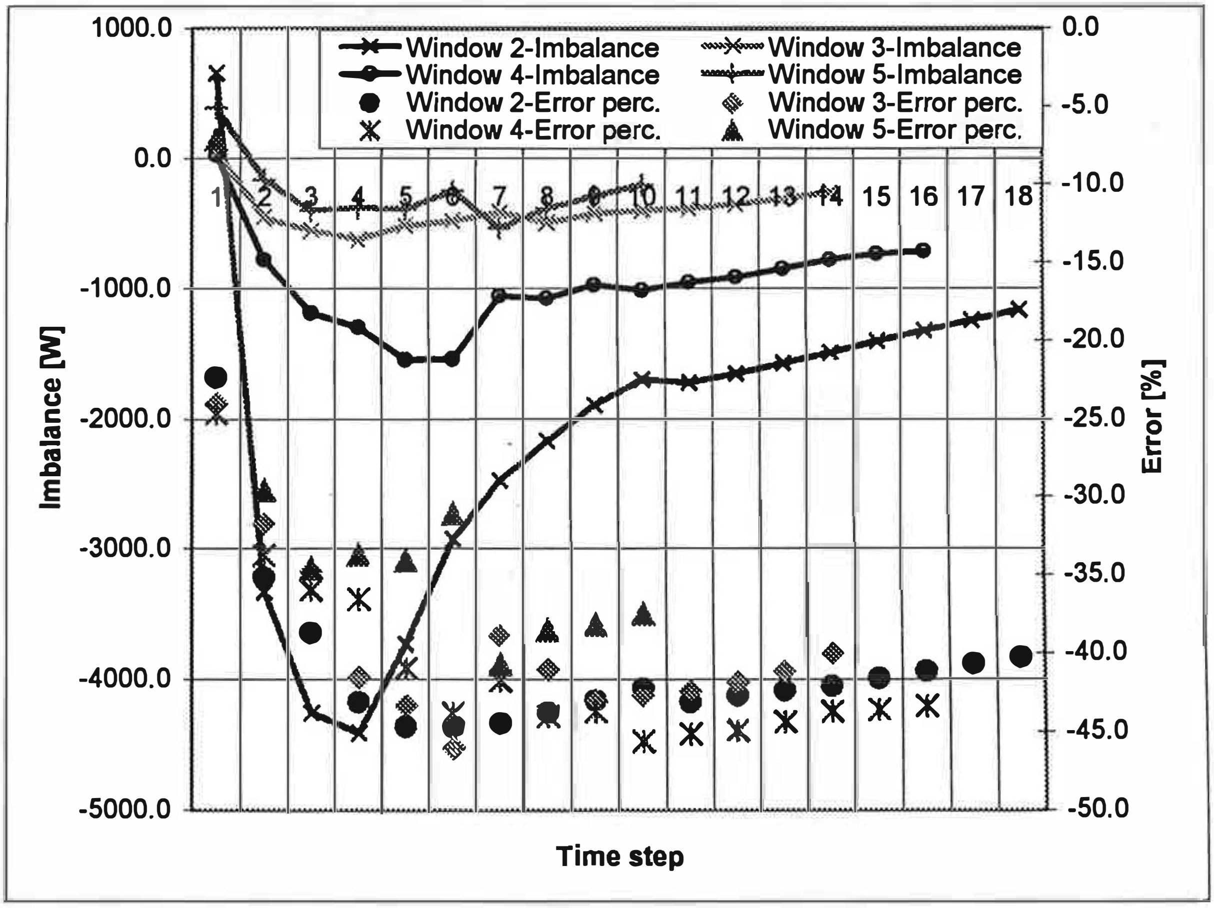


Figure 5 – Power imbalance and relative errors of CFD simulations.

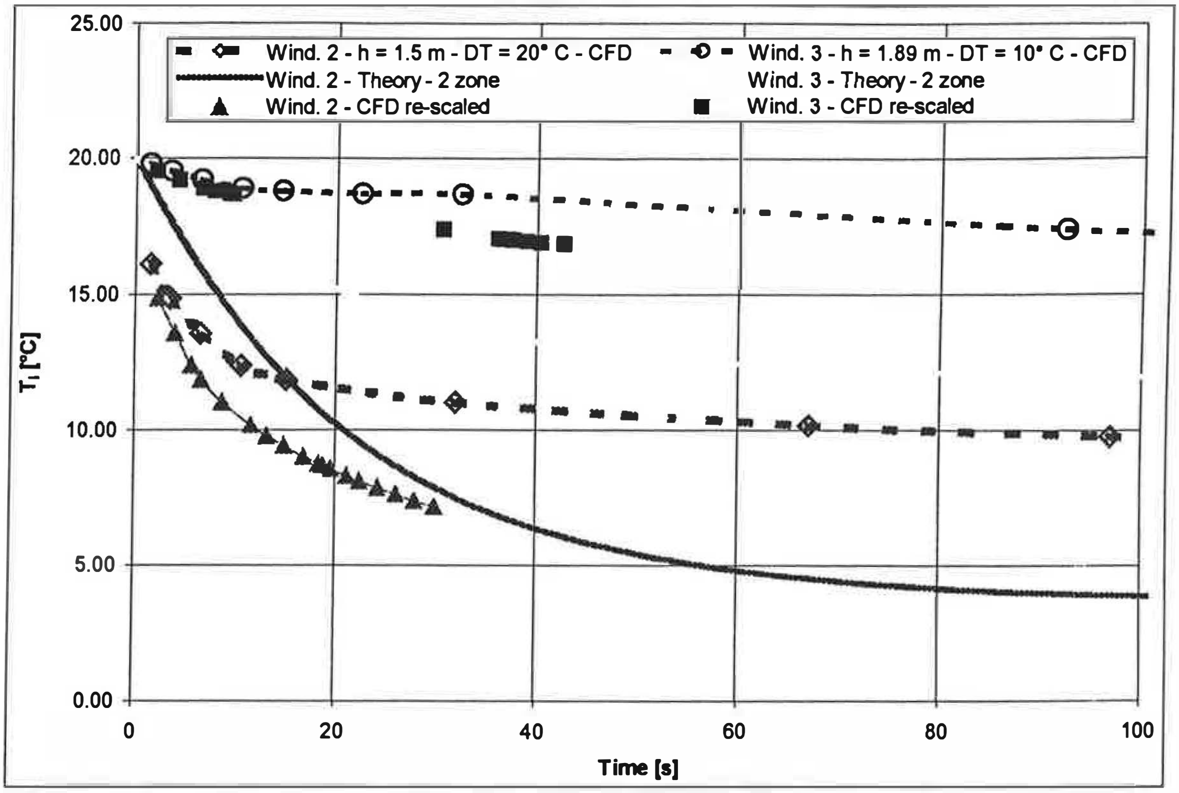


Figure 6 – Air temperature (room mean value) versus time (CFD, 2-zone and re-scaled values) – Models Window 2 and 3.

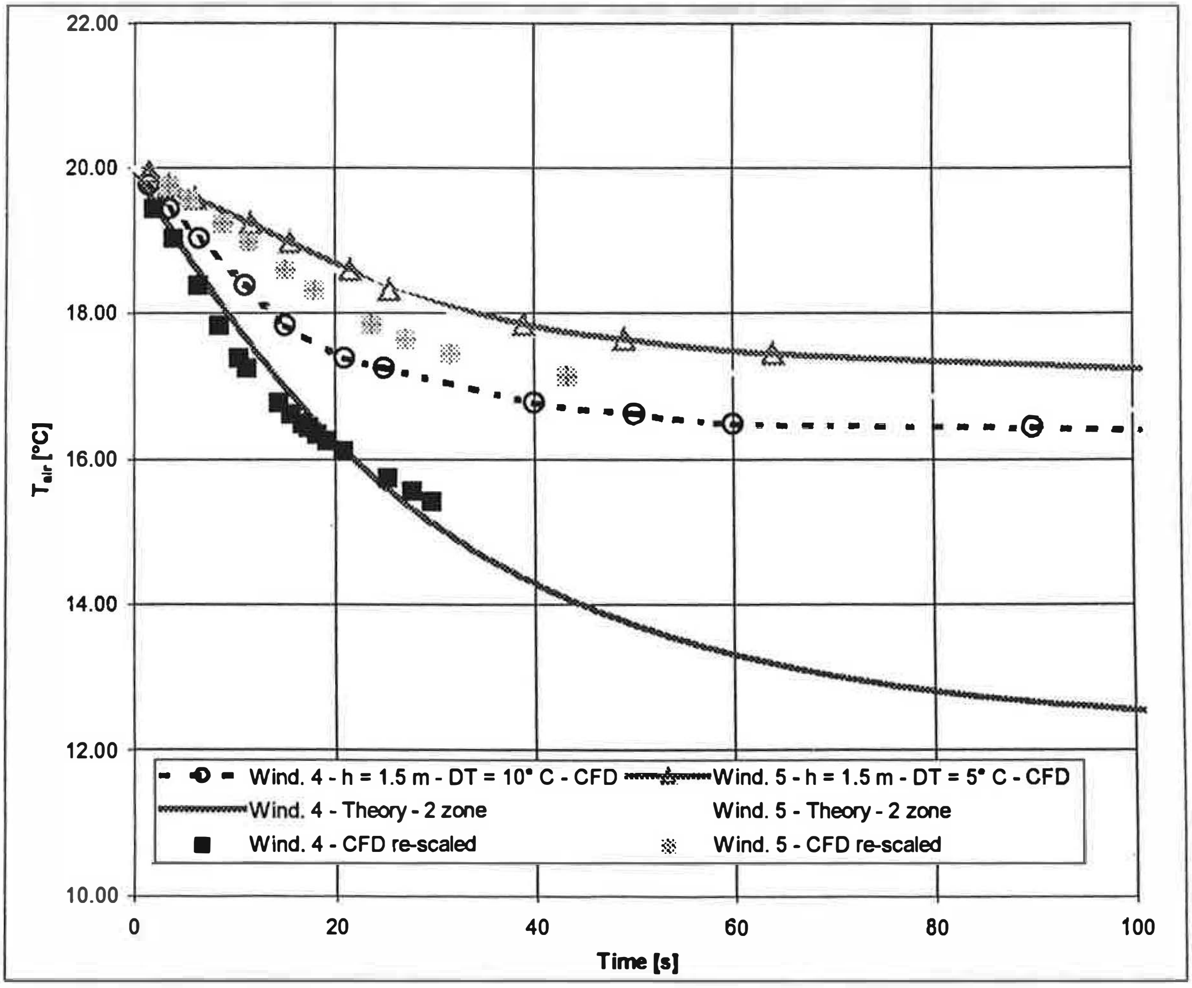


Figure 7 – Air temperature (room mean value) versus Time (CFD, 2-zone and re-scaled values) – Models Window 4 and 5.

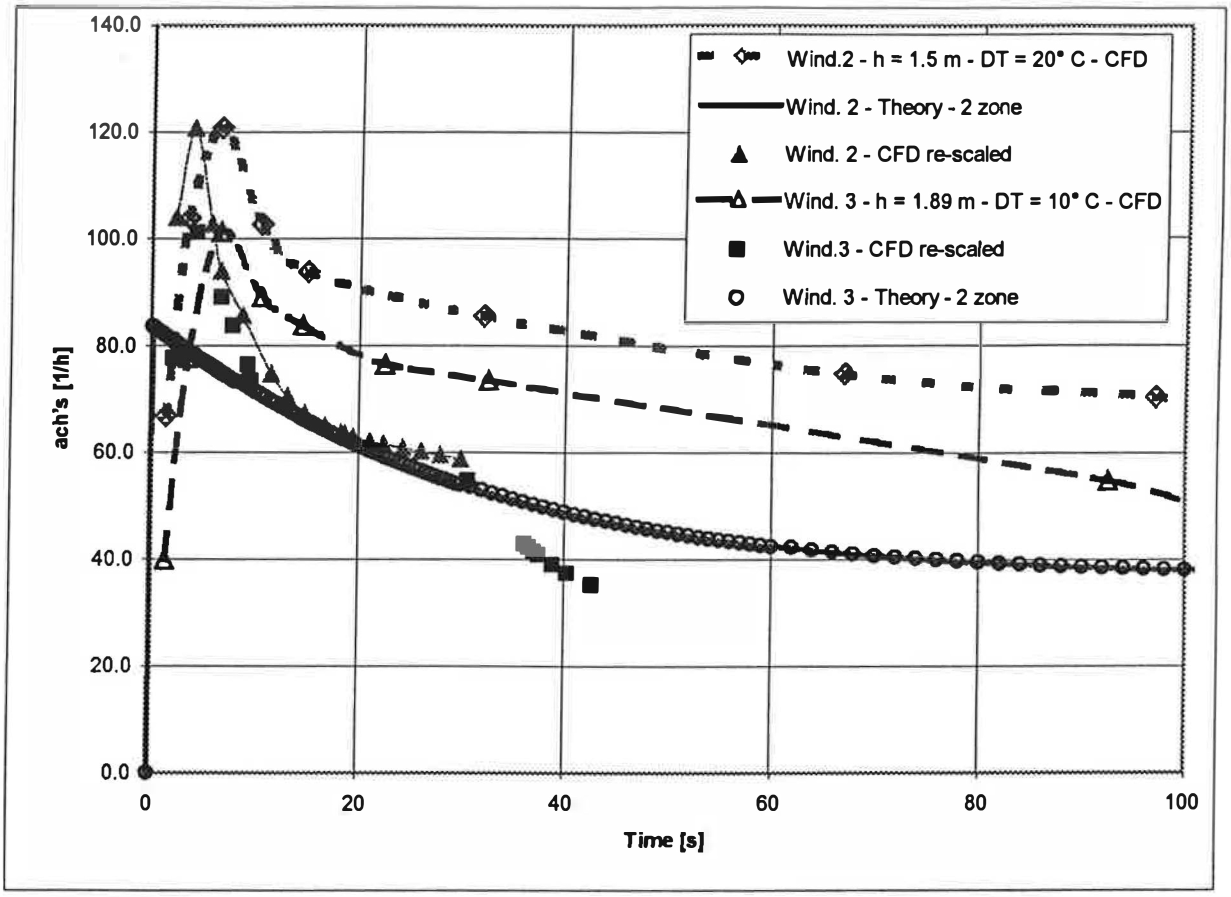


Figure 8 – Air changes per hour versus Time (CFD, 2-zone and re-scaled values) – Models Window 2 and 3.

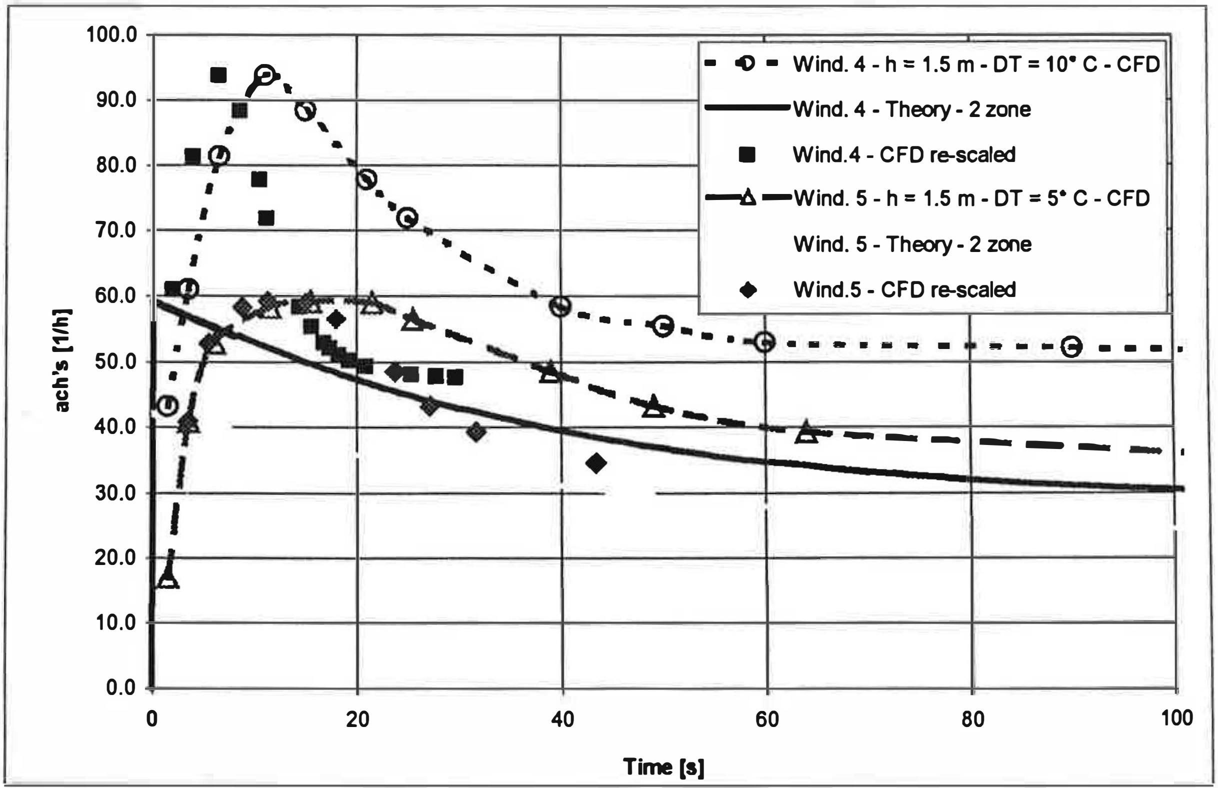


Figure 9 – Air changes per hour versus Time (CFD, 2-zone and re-scaled values) – Models Window 4 and 5.

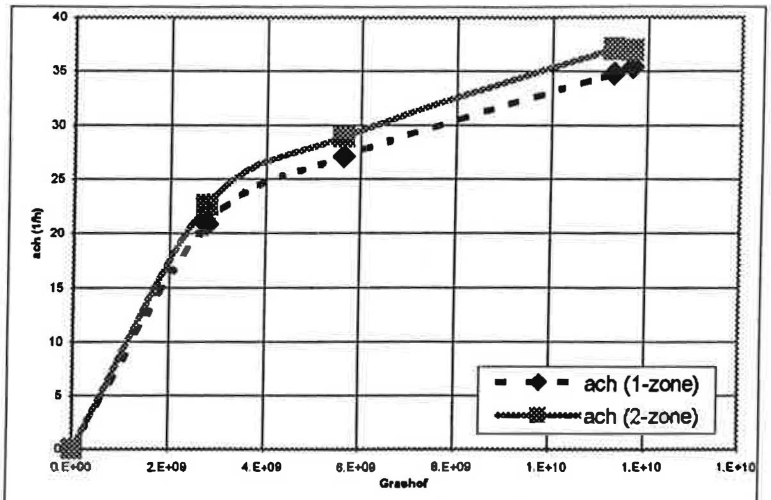


Fig. 11 –Steady-state ach's vs. Grashof number.

4 CONCLUSIONS

The work is, actually, still in progress. The aim of the paper was to verify the applicability of CFD analysis to a simple, yet physically complex phenomenon such as the consequences of opening a window under the mere effect of a temperature difference between indoors and outdoors.
It has been shown that CFD should be used very carefully, with a suitable choice of the calculation domain.
Furthermore, there is an apparent contradiction between the calculation time scale and energy conservation principles. This contradiction has been solved re-scaling the time history by forcing the solution to comply with the energy balance.
Simplified "analytical" models have also been developed, and their results have been compared to the CFD model results. After the time re-scaling, there is a fair agreement between CFD and engineering models, except for *window 3*, for which the CFD analysis revealed a certain degree of outdoor-indoor air recirculation.
A relationship between air changes per hour at steady state conditions and Grashof number has been derived (see figure 11). Both curves show a definite functional dependency upon Grashof number and could be used for first attempt prediction.

5 REFERENCES

Agnoletto L., E. Grava, La ventilazione naturale degli ambienti, Condizionamento dell'Aria, ottobre 1981.
Andersen K.T., Design of natural ventilation by thermal buoyancy – theory, possibilities and limitations, 5th Int. Conf. On air distribution in rooms, ROOMVENT '96, July 1996, Yokohama.
ASHRAE, Handbook of Fundamentals, Ch. 25, 1997.
Butera F., G. Cannistraro, M. Yaghoubi, A. Lauritano, Benessere termico e ventilazione naturale negli edifici, HTE Energie Alternative, anno 11, no. 59, maggio-giugno 1989.
Elsayed M., Infiltration Load in Cold Rooms, HVAC&R Research, Vol. 4 No. 2, April 1998.
Etheridge D., M. Sandberg, Building ventilation – Theory and Measurements, John Wiley & sons, Chichester, 1996, pp. 89-95,
Schaelin, A., Van der Maas J., Moser, A., Simulation of air flow through large openings in buildings, ASHRAE Transactions, part 2, 1992.

ACKNOWLEDGMENTS

The research activity presented in this work has been developed in the frame of "*Indoor Environment Engineering* "– a project co-funded by the Italian Ministry of University and Reasearch (MURST). The participation of the authors to IEA Annex 35 is supported by ENEA – Ente Nazionale Energie Alternative – Italy.