Air Distribution in Rooms, (ROOMVENT 2000)
Editor: H.B. Awbi

NUMERICAL SIMULATION OF TRANSIENT EFFECTS OF WINDOW OPENINGS

G V Fracastoro, G Mutani, M Perino

Department of Energy Technology, Politecnico di Torino
Corso Duca degli Abruzzi, 24 – 10129 Torino, Italy

ABSTRACT

This work is centered on the transient analysis of natural ventilation provided by a single side opening when only indoor-outdoor temperature differences are present (no wind). Using both simplified "engineering" models and a CFD commercial code (2D), different cases have been examined by varying indoor-outdoor temperature difference, window size, and including or not a heating appliance in the room. The CFD results reproduce correctly the phenomenon, but the time-scale seems to be inconsistent with global mass and energy conservation principles, in spite of the fact that numerical convergence is always achieved. Re-scaling the time allowed the results of the simulation to substantially agree with zonal model results. Furthermore, they allowed to determine the values, varying with time, of ventilation efficiency when a window is opened. The air changes at steady-state are also expressed as a function of Grashof number.

KEYWORDS
Natural ventilation, windows opening, CFD, stack effect.

SYMBOLOGY

A = half window surface
A_p = wall surfaces
h_p = film coefficient
c = specific heat
H = window height, enthalpy
$\dot{Q}$ = heat flux
T = temperature
V = room volume

β = compressibility factor
ξ = accidental pressure loss coefficient
ρ = density
ν = kinematic viscosity
τ = time
Subscripts
o,i, = outdoor, indoor
p,v = constant pressure, volume

INTRODUCTION

Opening a door or window is a simple and common action to improve the IAQ in a room, yet its consequences when a certain indoor-outdoor temperature difference is present and even if no

wind is present, are difficult to predict. Schaelin et al (1992) and Elsayed (1998) have already pointed out the difficulties which may arise when CFD techniques are used to simulate this phenomenon, which is actually complex, and strongly unsteady.
Difficulties start since the first stage of simulation, when the user has to choose the structure of the calculation domain. Confining the calculation domain to the heated space 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), even if the values of residuals would suggest a successful simulation.
Simplified models and equations (see for examples Etheridge and Sandberg, 1996, Andersen, 1996, ASHRAE Handbook of Fundamentals, 1997) are available for the calculation of air flow rates, 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, due to the unsteadiness of the phenomenon, the value of temperature difference to be used in these formulas has to be forecasted as an average between initial and final conditions.
The CFD results have been expressed as a function of Grashof number, $Gr = g \cdot \beta \cdot \Delta T \cdot H^3 / \nu^2$, where ΔT is the wall-outdoor temperature difference.

MODELS FEATURES

A naturally single-side ventilated enclosure 4.2 m (width) x 2.7 m (height) and 1 m (depth) has been taken in consideration. Only thermal effects have been taken into account. A total of 6 cases have been examined varying window height, temperature differences, and including or not a heating appliance in the room. Table 1 reports the main features of the simulations.
The initial air temperature has always been assumed equal to 20° C. The same value has been adopted for the wall temperatures, and considered constant.

TABLE 1
MAIN FEATURES AND BOUNDARY CONDITIONS OF SIMULATED CASES.

Case	Radiator	Window height [m]	$T_{i,0}-T_0$ [°C]	Grashof Number
1	No	1.5	20	$1.17 \cdot 10^{10}$
2	No	1.89	10	$1.13 \cdot 10^{10}$
3	No	1.5	10	$5.64 \cdot 10^{9}$
4	No	1.5	5	$2.77 \cdot 10^{9}$
5	No	1.5	2.5	$1.39 \cdot 10^{9}$
6	Yes	1.5	20	$1.32 \cdot 10^{10}$

CFD Models

A two-dimensional CFD transient analysis has been performed using a well known commercial software (FLUENT ®). The CFD model includes in the calculation domain a strip 2 m wide of outdoor environment, and is discretized using a non uniform grid made of 200 x 200 cells. The following assumptions were adopted: standard k-ε turbulence model, power-law interpolation scheme, standard log-law wall functions. A total number of about 100 time-steps varying from 0.5 s (in the first 20-30 s of simulation, to avoid numerical instability) up to 60 s were simulated, with 1000 iterations per time-step. The equivalent time span was about 600 s for all cases. The computational time was about 3 weeks on an HP Apollo 720 RISC WS (54 Mb RAM memory).

Engineering Models

a) A single-zone model has been developed, where the equation for mass flow rate $\dot{m}$, given by ASHRAE (1997), is coupled with the equation of conservation of energy for indoor air:

$$\dot{m} = \rho_o \cdot A \cdot C_d \cdot \sqrt{\frac{g \cdot H \cdot (T_i - T_o)}{T_i}} \tag{1}$$

$$h_p \cdot A_p \cdot (T_p - T_i) = \dot{m} \cdot c_p \cdot (T_i - T_o) + \rho_i \cdot V \cdot c_v \cdot \frac{\partial T_i}{\partial \tau} \tag{2}$$

where the discharge coefficient C_d is given by:

$$C_d = 0.40 + 0.0045 \cdot |T_i - T_o| \tag{3}$$

b) The two-zone model is described by the following equations

$$\begin{aligned} h_{p1} \cdot A_{p1} \cdot (T_{p1} - T_1) &= \dot{m} \cdot c_p \cdot (T_1 - T_o) + \rho_1 \cdot V_1 \cdot c_v \cdot \frac{\partial T_1}{\partial \tau} \\ h_{p2} \cdot A_{p2} \cdot (T_{p2} - T_2) &= \dot{m} \cdot c_p \cdot (T_2 - T_1) + \rho_2 \cdot V_2 \cdot c_v \cdot \frac{\partial T_2}{\partial \tau} \end{aligned} \tag{4}$$

$$\dot{m} = A \cdot \sqrt{\frac{g \cdot H \cdot \left(\rho_o - \frac{\rho_1 + \rho_2}{2}\right) \rho_o \cdot \rho_2}{\xi_1 \cdot \rho_2 + \xi_2 \cdot \rho_o}} \tag{5}$$

The value of $\xi_1 = \xi_2$ has been determined assuming the initial air flow rate to be equal to the initial flow rate of the single-zone model. The air is supposed to flow only from the lower zone (1) to the upper zone (2), without recirculation.

These two models have been solved numerically, discretizing the ODE's and employing an explicit time integration. In order to comply with energy and mass balances a time step of 0.1 s has been used for the first 8-10 seconds, increasing it with time.

RESULTS

The results are strictly applicable only to a continuous "strip" type of window, but they may probably be extended to cases where the ratio of window height to width is not too large.

The profiles of ach's (determined considering 1 m depth) and air mean temperature versus time (not shown here for the sake of brevity) obtained by means of the single- and two-zone models are quite similar. Table 2 sums up the indoor air mean temperature and the ach's values at steady state conditions (and the corresponding time to reach them).

TABLE 2
STEADY STATE AIR CHANGES AND TEMPERATURES

Case	n [1/h]	T [°C]	n [1/h]	T [°C]	$\tau(\infty)$[s]
	Single zone		Two zone		
1	35.4	4.7	37.0	3.7	≈150
2	34.8	12.5	37.1	11.9	≈150
3	27.1	13.0	29.0	12.3	≈220
4	20.8	16.8	22.6	16.4	≈220
5	15.9	18.6	17.5	18.4	≈200
6	36.2	5.2	-	-	≈200

The CFD results in terms of air flow patterns are consistent with conventional wisdom, as shown in Figure 1, where the profiles of mass flow rate across the window at different time steps are reported, as an example, for *case 3*.

On the opposite, the average room air temperature profiles versus time calculated by means of CFD are very different from those of the zonal models. Actually, the analysis of energy balance at each time step has revealed that CFD models predict correctly the enthalpy fluxes and convection

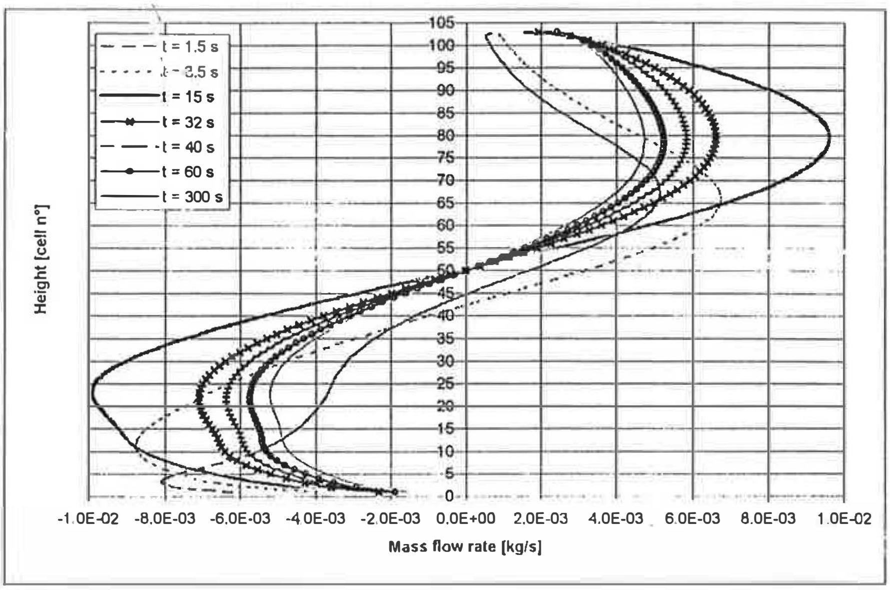

Figure 1 –Mass flow rates along the window height for different time steps– *Case 3.*

heat fluxes between walls and air, while the variation of temperature with time appears to be largely underestimated. The time step intervals adopted for the numerical solution procedure do not seem to fit with the physical time scale of the phenomenon. To overcome this problem the energy balance has been imposed at each time step, and the time-scale of the CFD results has been re-determined coherently from the following equation

$$\Delta\tau = \frac{\rho \cdot V \cdot c_v \cdot \Delta T}{\dot{Q} - \dot{H}}. \quad (6)$$

Figures 2 and 3 show the corrected CFD and the two-zone (2z) model results in terms of ach's and mean air temperature vs. time. The agreement is now fair for *cases 1, 3, 4 and 5*, while it is rather poor for *case 2*, probably because substantial re-circulation of warm exhaust air with cold air entering occurs, and lower indoor-outdoor temperature differences induce smaller air flows. Case 6 (same ΔT as case 1, but with radiator located adjacent to the wall opposite to the window) shows constantly lower ach's, but tending to the same value as case 1, and constantly higher room air temperatures, due to the higher equivalent surface temperature of the walls.
From the CFD results *ventilation efficiency* has been calculated for cases 4, 5 and 6. Its values start from one and tend to vary with time in a rather unpredictable way, decreasing for *case 4* increasing for *case 5* and maintaining perfect mixing conditions for *case 6*, as shown in table 3.

Table 3
Ventilation efficiency (-)

Time (s)	0	5	10	15	20	25
Case 4	1	0.98	0.96	0.91	0.74	0.61
Case 5	1	1.02	1.11	1.35	1.95	2.92
Case 6	1	-	-	0.98	1.02	1.01

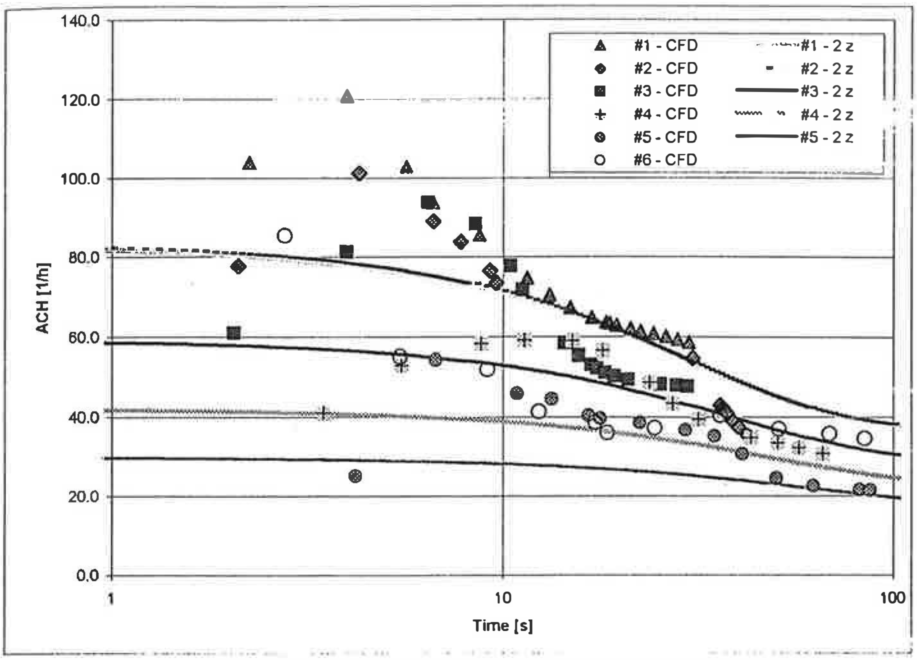

Figure 2 – Air changes per hour versus time (CFD re-scaled and 2-zone values)

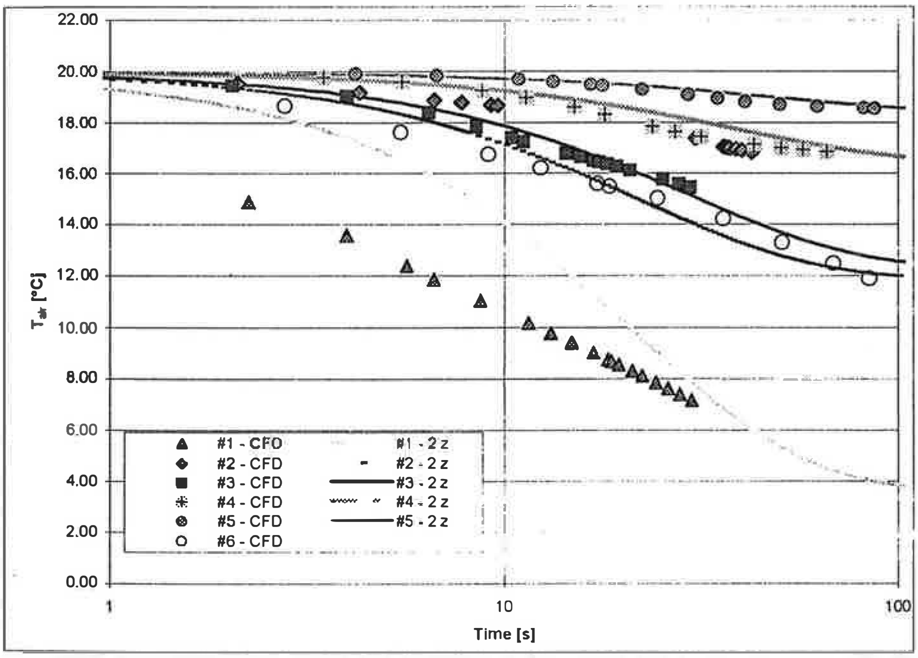

Figure 3 – Air temperature versus time (CFD re-scaled values and 2-zone)

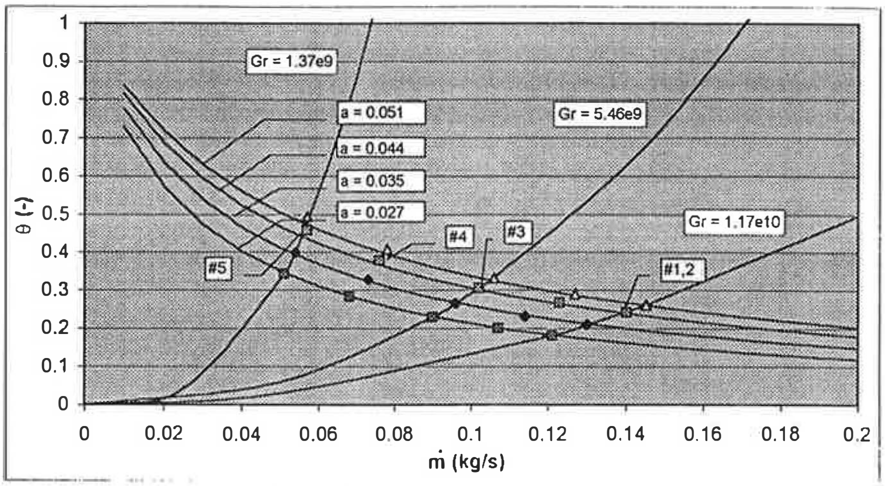

Figure 4 – Non-dimensional temperature as a function of mass flow rate and Gr at steady-state.

The relationship, derived from zonal models at steady state, between mass flow rate, non-dimensional temperature $\theta = (T_\infty - T_o)/(T_p - T_o)$, and Grashof number is shown in figure 4 ($a = hA_p/c_p$).

CONCLUSIONS

The aim of the paper was to verify the applicability of CFD transient 2D analysis to a simple, yet complex phenomenon such as the evolution of thermal and fluid dynamic fields following the opening of a window under the mere action of indoor - outdoor temperature difference.

Simplified "zonal" models have also been developed, to verify the time evolution of the phenomenon and the conditions at steady-state. A number of different configurations and boundary conditions have been simulated, including the presence of a radiator.

It has been shown that CFD should be used very carefully, with a suitable choice of the calculation domain. Furthermore, in order to comply with First Principle constraints, the time evolution had to be re-scaled. Once corrected, the CFD results fairly agree with engineering models. From CFD, the ventilation efficiency of airing by stack-effect has also been derived. Its evolution has shown controversial tendencies depending on the temperature difference.

References

- Andersen K.T., Design of natural ventilation by thermal buoyancy – theory, possibilities and limitations, 5th Int. Conf. ROOMVENT '96, July 1996, Yokohama.
- ASHRAE, Handbook of Fundamentals, Ch. 25, 1997.
- 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

This research activity has been developed in the frame of "*Indoor Environment Engineering*"– a National Project co-funded by the Italian Ministry of University and Research (MURST).