5855

BA-92-2-4

SIMULATION OF AIRFLOW THROUGH LARGE OPENINGS IN BUILDINGS

A. Schaelin, Ph.D.

J. van der Maas, Ph.D.

A. Moser, Ph.D.
Member ASHRAE

ABSTRACT

The coupling of indoor airflow to the outdoor flow through large openings in buildings that allow bidirectional flow is of considerable interest for many applications. In this work, the airflow pattern has been calculated for situations in which warm air leaves a room through the upper part of the large opening and rises as a thermal plume, with and without interaction with wind.

Both two- and three-dimensional CFD simulations of the velocity and temperature fields in a thermal plume agree well with the analytical models. These, in turn, are well supported by experiments. For single-sided ventilation without wind, the velocity profiles in the doorway predicted by CFD have features that are well known from experimental data but cannot be found in analytical models. With a normal incident steady wind on the opening, an increase of the convective flow is found in the three-dimensional case but not in the two-dimensional case.

At the boundaries of the computation domain corresponding to the free atmosphere, a zero pressure gradient boundary condition was found to be sufficient to avoid unrealistic recirculation. By proper placement of these boundaries around a heated room, good numerical results can be achieved for a computation domain extended only a little to the outside of the room.

INTRODUCTION

Interest in simulating the airflow through large openings in buildings, such as windows or doors, allowing for bidirectional flow, is increasing. Applications include the natural ventilation of office rooms and buildings or, in general, airflow around buildings interacting with airflow in rooms. In such a case, the indoor airflow is dominated by the convected flow through the opening because, for example, in a winter situation, warm air goes from the room to the outside and cold air enters the room through the same opening.

While some publications have shown calculations of the coupling of airflow through a door between two adjacent rooms (Jiang et al. 1992), little computational work has been published so far for the CFD modeling of airflow through large openings with open boundaries, particularly for free convection cases without wind. In such situations, the computational grid has to be extended from the room through the opening to the outdoor air.

A number of experimental studies have shown that the airflow through doors inside buildings is reasonably well understood and can be modeled both with simplified models using the temperature difference between rooms or with numerical methods (Pelletret et al. 1991; Haghighat et al. 1989; Jiang et al. 1992; van der Maas et al. 1992).

In contrast, studies of airflow through openings to the outdoor environment, where stack flow and wind interact, show that it is difficult to understand the flow rate (Bienfait et al. 1991). Parameters in models for single-sided ventilation are semi-empirical in nature and have large uncertainties. The usual assumptions for single-sided ventilation (Bienfait et al. 1991) are that (1) the effect of a steady, normally incident wind is an increase of the inside pressure but it does not affect the bidirectional stack flow; (2) variations of the average pressure coefficient over the opening cause two-way flow that interacts with the stack effect; and (3) pressure fluctuations increase the air change rate due to the compressibility of the air. To model single-sided ventilation numerically, the simplest cases should be treated first, and so in this paper we consider only the stack effect and the interaction with a steady, normally incident wind.

Wind tunnel and numerical studies usually concern isothermal airflow around buildings. In this paper, for the first time to the authors' knowledge, a numerical study of the interaction of wind with the bidirectional stack flow through a single opening in a heated cavity is presented.

Therefore, this work focuses on the simulation of airflow in a room with a heater inside and a large opening to the outdoors, with emphasis on the representation of the main flow without considering the details of near wall flow, heat transfer, etc. The boundary condition for the free atmosphere at the boundary of the calculation grid is discussed. The calculated flow field of the rising warm air plume without wind is compared with analytical predictions. The velocity field in the plane of the large opening is calculated in two and three dimensions and compared with simple analytical models based on the Bernoulli equation and with experiments. Finally, the calculation domain size is estimated, which is needed to yield reasonable results.

Alois Schaelin, Ph.D., is a research scientist and **Alfred Moser**, Ph.D., is a staff scientist at the Energy Systems Laboratory, Swiss Federal Institute of Technology, ETH-Zentrum, Zurich, Switzerland. **Jacobus van der Maas**, Ph.D., is a research scientist at the LESO-PB, Solar Energy and Building Physics Laboratory, EPFL, Swiss Federal Institute of Technology, Lausanne, Switzerland.

Numerical Performance

Two convergence criteria have been applied:

- The sum over all cells of the absolute values of the residuals of Equation 1 should be smaller than 0.01% of the main flux of the corresponding quantity Φ.
- The overall mass flow error, defined as $|\dot{M}_{in} - \dot{M}_{out}| / \dot{M}_{in}$ where $\dot{M}_{in}$ and $\dot{M}_{out}$ are mass inflow and outflow through the boundaries of the whole domain, should be typically $< 0.01\%$.

The final (converged) flow field, judged from the visual appearance, is usually achieved about 10 times faster than the above criteria are met. The sweep (i.e., one iteration of all equations for all grid cells) number needed depends on the cases. Problems with wind typically offer a convergence five to ten times faster than those without.[1]

SIMULATION RESULTS AND ANALYSIS

The final intent is the modeling of the two-way airflow through a large opening such as a door or a window. While the full problem would require treating time-dependent flow and temperature fields (stack effect plus fluctuating wind pressure), the scope of the present study is limited to (1) the modeling of the free convection case of a hot air plume flowing out of a large opening and (2) the mixed convection flow due to a steady wind normally incident on the opening of a heated room with the heater inside.

We consider the case of a room with a heater connected to the atmosphere by a door (Figure 1). Warm air leaves the room through the upper part of the opening and cold air enters through the lower part of the opening. The warm air rises outside as a thermal plume due to buoyancy.

As a first test of this large opening simulation method, the numerical calculation of such a plume alone without a building, compared with simple analytical models, is discussed in the next section. The following section expands this modeling to a room with a building, with and without wind, in two and three dimensions. The results are compared with a simple analytical model and experimental results at the opening cross section. The next section describes an investigation of the minimum calculation domain size necessary to yield proper results at the lowest computation time.

Free Plume Cases

For the thermal plume analysis, only free convection cases without wind were considered, with the geometry chosen as simple as possible so the plumes can be studied undisturbed by obstacles and other influences. The cases are two-dimensional Cartesian and axisymmetric and three-dimensional Cartesian.

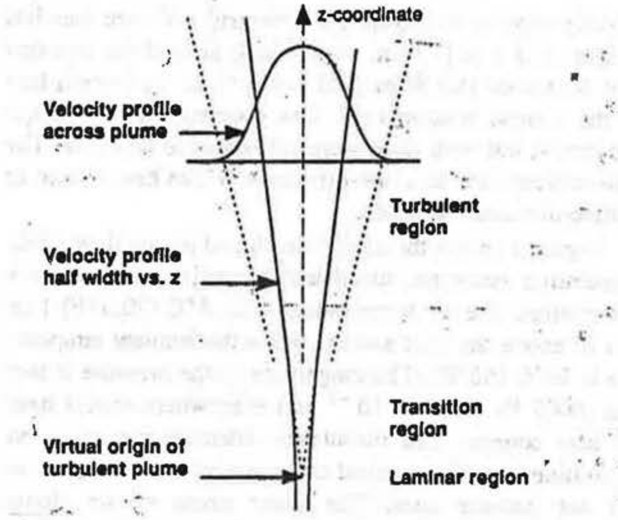

Figure 2 *Sketch of a buoyant plume from laminar to turbulent region. Shown are the meaning of the half-width of the profile across the plume of the vertical velocity component and the virtual origin of the linear spreading of the velocity profile half-width of the turbulent plume.*

Analytical Plume Laws Figure 2 shows a sketch of a plume with a cross section of the velocity profile. For the zone of fully developed turbulent flow at isothermal surroundings, analytical plume laws can be derived. These laws date back to 1941 (Schmidt) and are reviewed for different cases by Chen and Rodi (1980) and Rodi (1982) with a comparison to measurements. The dependences are shown in Table 1; the analytical predictions, in turn, are well supported by experimental data (Chen and Rodi 1980; Rodi 1982; Kofoed and Nielsen 1990).

Free Plume, Two-Dimensional Cartesian The grid used has 45 × 34 cells and spans 61 m × 100 m (200 ft × 328 ft), horizontally and vertically, respectively. The grid distribution is indicated by the distribution of the

TABLE 1
Power Law Dependences as Functions of Height z for the Profile of the Turbulent Plume

Turbulent Plume	Peak Velocity	Velocity Profile Half-Width	Peak Temperature
Two-dimensional plane	const	$\sim z$	$\sim z^{-1}$
Axisymmetric (three-dimensional)	$\sim z^{-1/3}$	$\sim z$	$\sim z^{-5/3}$

Note: The tilde (~) means "proportional to" (after Chen and Rodi 1980; Schmidt 1941).

---

[1] The simulations were done mostly on a computer with about 4 MFLOPs performance for the implemented partly vectorized CFD code. A two-dimensional 46 × 46 = 2,116 cells case without wind needs about 10,000 sweeps to converge in a time of 7.5 hours.

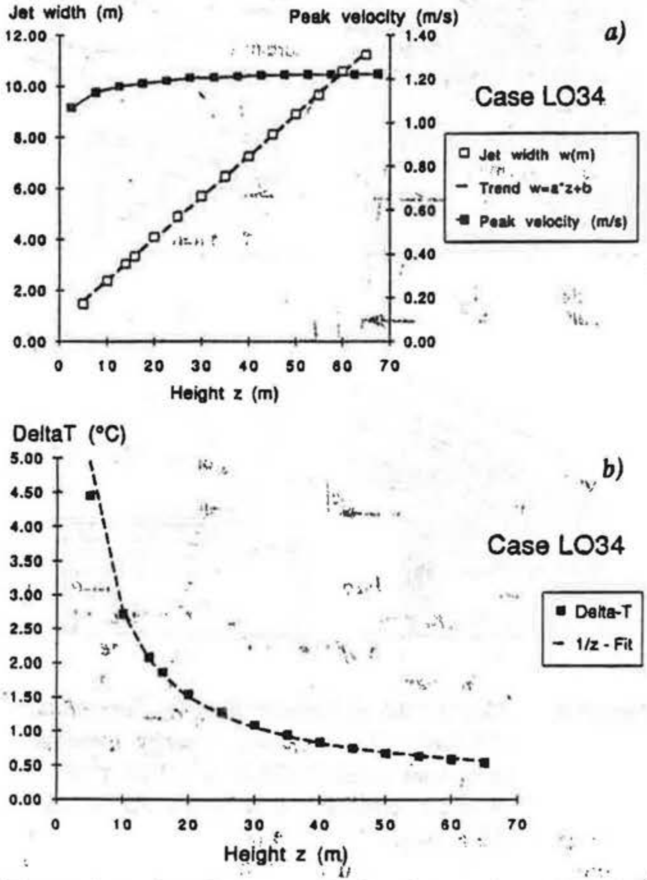

Figure 4 Case LO34. a) Peak value and profile half-width of the plume velocity as functions of height. b) Peak value of the plume temperature as a function of height.

(7.2 ft) high (see Figure 6a). A 0.6 m (2 ft) high heater at 50°C (122°F) in the corner opposite the door drives the airflow in the room. This geometry was inspired by preliminary wind tunnel experiments showing interesting effects. The grid has 46 × 46 cells and spans 34 m × 72.2 m (111 ft × 237 ft), horizontally and vertically, respectively. Figure 7 shows the flow field and contours of temperature, turbulent energy, and pressure.

Figure 4 shows the dependence of velocity peak value and profile half-width as functions of the height for the free convection case (no wind). The profiles are the same as in the free plume case described above. The plume is tilted in the first 2 m (7 ft) above the roof a little bit to the right, probably due to the fact that in the first 5 m (16 ft) more air is entrained from the left-hand side than from the right-hand side, and the correspondingly higher momentum from the left is responsible for the slight tilt. Apart from that, the different geometrical conditions in the lowest part of the plume do not seem to change the plume shape above as long as the plume buildup is not disturbed by other obstacles.

For the wind case, the wind speed was set uniformly to 1 m/s at the left boundary, without taking into account a wind profile near the ground that will be part of future

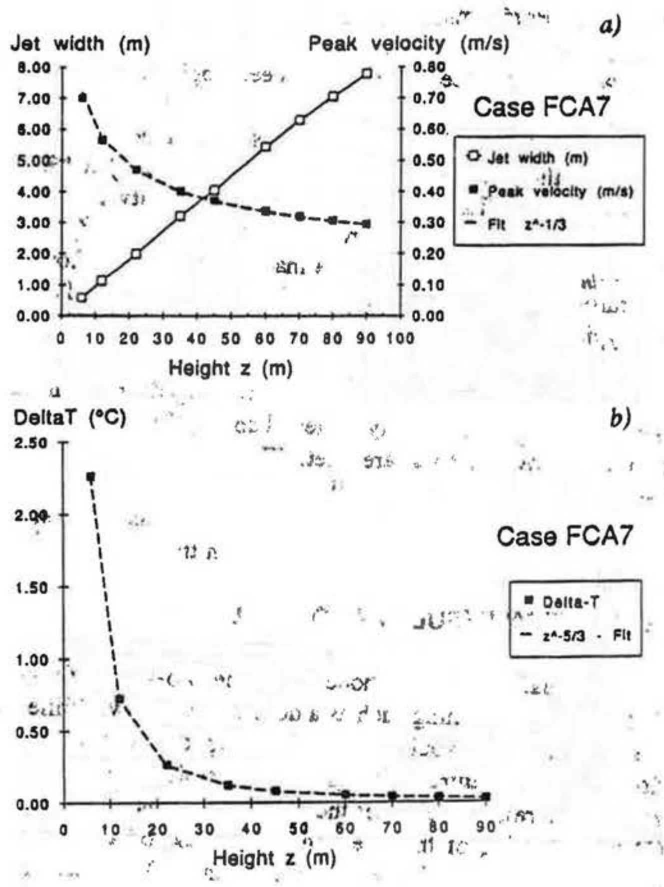

Figure 5 Case FCA7. a) Peak value and profile half-width of the plume velocity as functions of height. b) Peak value of the plume temperature as a function of height.

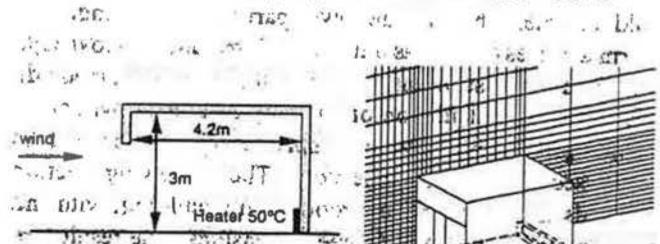

Figure 6 a) Two-dimensional large opening case. The grid has 46 × 46 cells and spans 34 m × 72.2 m (111 ft × 236 ft), horizontally and vertically, respectively. b) Three-dimensional large opening case. The grid has 41 × 31 × 31 cells and spans 34 m × 25 m × 18.7 m (111 ft × 82 ft × 61 ft) in x, y, and z, respectively. The shown grid resolution around the building also applies in the two-dimensional case.

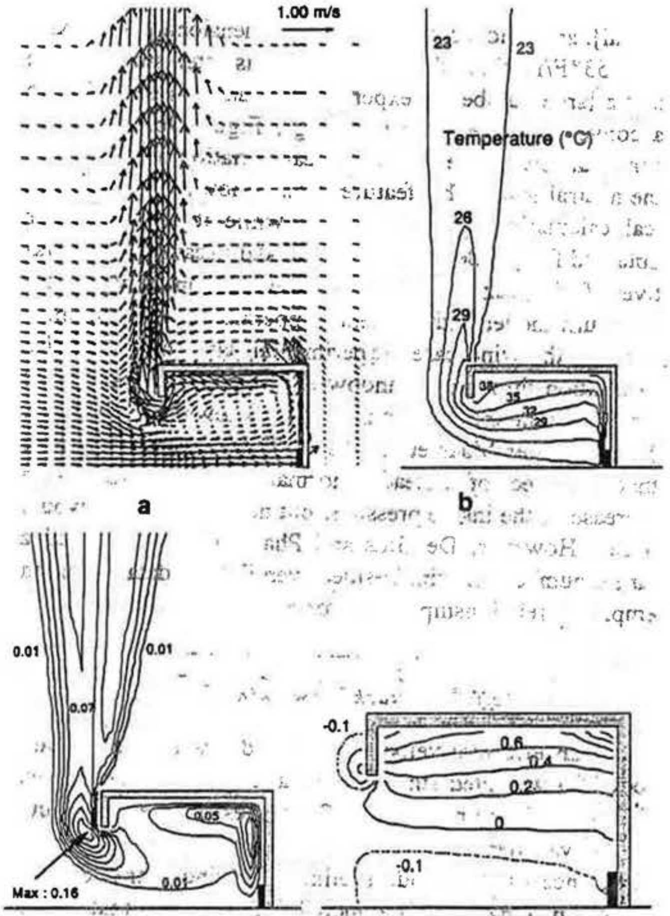

Figure 7 *Case LO34. a) Velocity field. b) Temperature contours. c) Turbulent energy contours; maximum value 0.16 m^2/s^2 (1.7 ft^2/s^2). d) Pressure contours, units Pa (1 Pa = 3.4 × 10^{-4} psi).*

investigations. Friction at the ground causes a more realistic profile after about 10 m (30 ft).

The wind case (Figure 8) shows that the thermal plume is blown away by the wind. The pressure contours in the surroundings of the house show a distribution similar to the pressure distribution around a closed cube in the cross-wind. The convective flow with wind turns out to be 10% smaller than in the free convection case.

Comparison with Experimental Results on Single-Sided Ventilation As long as there is no wind, the two-way stack flow through a single opening connecting two zones in a building is similar to the two-way flow through a window between a single zone and the atmosphere. A large number of experimental data on interzonal flow are available in the literature. However, for a comparison between experimental data and CFD calculations, a complete data set is needed, but only a few are available (Haghighat et al. 1989; Jiang et al. 1992).

The velocity profiles in the doorway obtained from CFD can be compared with velocities predicted by an analytical Bernoulli model. The latter is a common tool in the literature to interpret experimental data (e.g., Mahajan 1987; Pelletret et al. 1991), and it is also the type of algorithm used in multizone airflow models (van der Maas et al. 1992). While the Bernoulli algorithm gives the right order of magnitude of the velocities in the vertical mid-plane of the opening, data are often fitted using the discharge coefficient as a parameter. Even when the vertical temperature gradients in each zone are taken into account, fits of different experimental data sets and the Bernoulli model show a large spread in values.

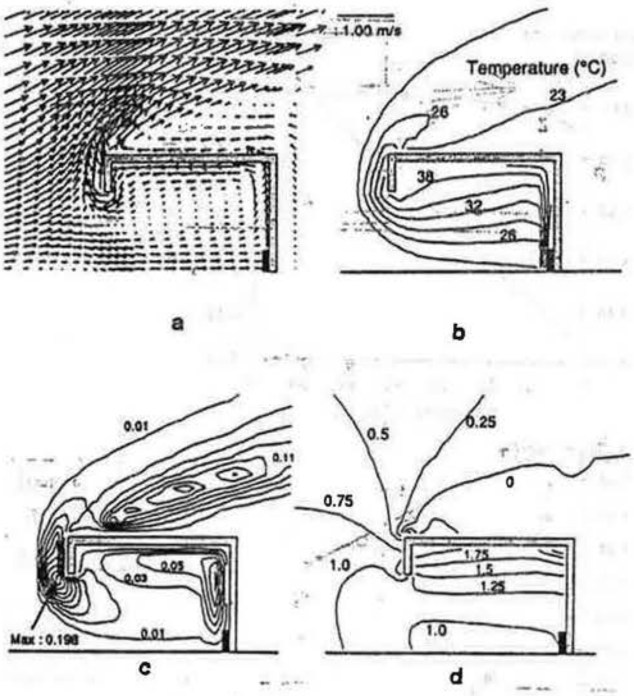

Figure 8 *Case LO42. a) Velocity field. b) Temperature contours. c) Turbulent energy contours; maximum value 0.198 m^2/s^2 (2.13 ft^2/s^2). d) Pressure contours, units Pa (1 Pa = 3.4 × 10^{-4} psi).*

Figure 9a shows the calculated velocity distribution in the door of the two-dimensional large-opening case, and Figure 9b shows in an overview the velocity profiles in the two- and three-dimensional cases, together with an experimental set of data to illustrate the agreement of the calculated shapes with the experimental ones.

It is interesting to compare, for the cases presented in Figure 6, the calculated velocity profiles in the opening with the prediction of the Bernoulli model (Figure 10). The Bernoulli velocity is calculated for zone temperature profiles $T_2(z)$ and $T_1(z)$ from

$$v(z) = C_d \sqrt{2g\,(T_2(z) - T_1(z))(z - z_n)/T} \qquad (4)$$

where C_d is the discharge coefficient, and z_n is the neutral level of the bidirectional flow. To calculate z_n, mass

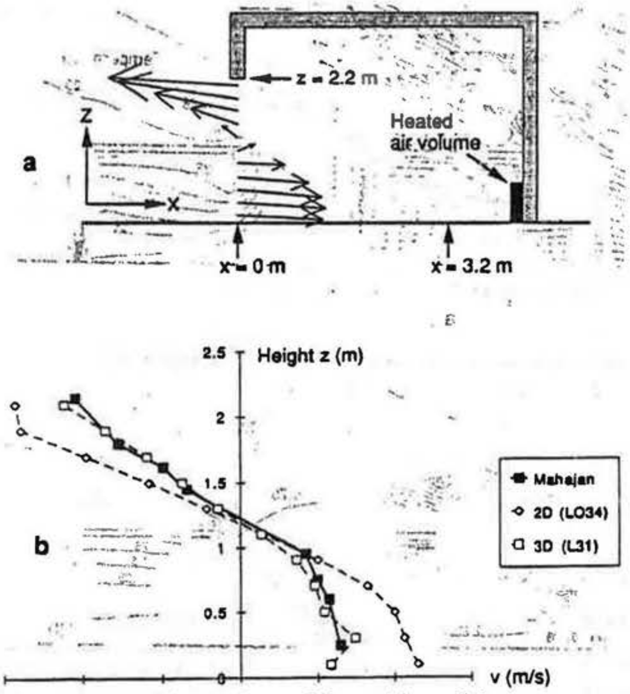

Figure 9 *a) Sketch of geometry with vector plot of the velocity field in the opening from the two-dimensional case LO34. b) Comparison of calculated two-dimensional and three-dimensional velocity fields in the opening with experimental data from Mahajan (1987); "2D" means "two-dimensional" and "3D" means "three-dimensional." The velocity values from Mahajan are scaled by a constant factor in order to compare the shape of the profiles.*

conservation is required, i.e., the incoming and outflowing mass have to be the same. A difficulty in applying the analytical model to the numerical data is the variation of the temperature profile over the cavity length (Figures 10a and 10c). This is a well-known problem for the interpretation of experimental data, often solved by experimenters by taking an average value for the temperatures.

The simplest case is where the inside temperature is taken to be uniform and constant and the discharge coefficient is given by the contraction coefficient for a sharp orifice, $C_d = 0.63$. The model then predicts a parabolic velocity profile for which the derivative $dv(z)/dz$ at the neutral level is infinite.

In Figures 10b and 10d the velocity profiles for the two- and three-dimensional calculations are compared with the Bernoulli model velocities. The apparent fit is artificial, however, because it was necessary to use a particular temperature profile (see Figures 9a and 9c) that does not correspond to the actual temperatures in the cavity (neutral level $z_n = 1.2$ m or 4 ft; discharge coefficient of $C_d = 0.7$; temperature gradients above z_n, 6 K/m or 3.2°F/ft [two-dimensional] and 3 K/m or 1.6°F/ft [three-dimensional], and below z_n, 0 K/m [two-dimensional] and 1 K/m or 0.53°F/ft [three-dimensional]). It is interesting to note that a large number of experimental data corresponding to a convectively heated cavity (e.g., Figure 9b, data from Mahajan 1987) give a nearly linear variation of $v(z)$ across the neutral level. This feature is also shown by the numerical calculation, presented here, while it hardly can be obtained from a Bernoulli-based model because the derivative of $dv(z)/dz$ at the neutral level is infinity for the Bernoulli model with a linear temperature stratification.

For the wind case, experimental data on single-sided ventilation through a window show that there is a wind effect, but the mechanism is not well known (Bienfait et al. 1991; van der Maas et al. 1992). The usual assumption is that the effect of a steady, normally incident wind is an increase of the inside pressure, but no flow increase would occur. However, De Gids and Phaff (1982) correlated a large number of single-sided ventilation data with an empirical relationship of the form

$$v_{eff} = \sqrt{v_{stack}^2 + C_w\, v_{wind}^2 + C_t} \qquad (5)$$

where an effective velocity, v_{eff}, is defined as the square root of a weighted sum of the stack and wind velocities. The wind fitting parameter, C_w, varies between 0.04 and 0.003 (van der Maas et al. 1992), and C_t represents the turbulence of the wind. Preliminary wind tunnel experiments on a heated scale model indeed show that under normally incident wind C_w was significantly different from zero. This feature is also exhibited by the three-dimensional results but not by the two-dimensional calculations. In the former case, a wind fitting coefficient of $C_w = 0.06$ is estimated for the case described below.

Large Opening, Three-Dimensional The room is 4 m (13.1 ft) long, 4 m (13.1 ft) wide, and 3 m (9.8 ft) high, with a door 2.2 m (7.2 ft) high and 1 m (3.3 ft) wide (see Figure 6b). A 0.6 m (2 ft) high and 3 m (9.8 ft) wide heater at 50°C (122°F) in the corner opposite the door drives the airflow in the room. The grid has 41 × 31 × 31 cells and spans 34 m × 25 m × 18.7 m (111 ft × 82 ft × 61 ft), x, y, z, respectively. The case is so far symmetrical with respect to a y plane through the middle of the room, but the full case was calculated for convenience with cases with general wind directions (not included in this paper). Figure 11 shows some cross sections of the flow field for the two cases, with and without wind of 1 m/s (3.3 fps).

The evaluation of the flow fields at the door now yields about 20% more convective flow in the wind case. From Equation 5, with values $v_{stack} = 0.4$ m/s, $v_{eff} = 1.2\, v_{stack}$, and $v_{wind} = 1$ m/s, a value of $C_w = 0.06$ is estimated.

It means that three-dimensional modeling is necessary to compare with the experiments in this case. The origin for the difference must be determined by further investigations with modified geometry. A possible reason could be that the heater width is longer than the door width.

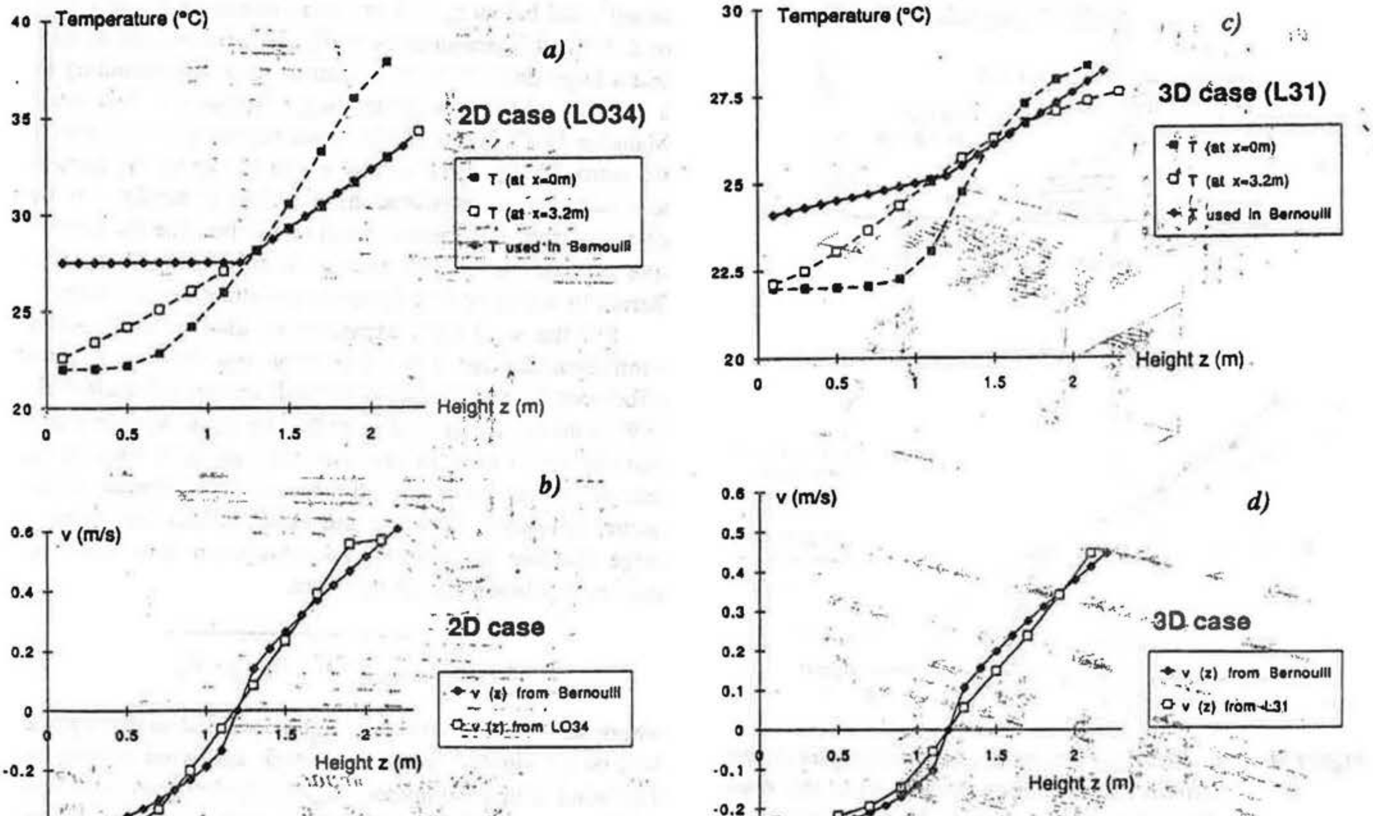

Figure 10 *Two-dimensional case: a) Calculated temperature profiles at two planes (see Figure 9a) and profile chosen for the Bernoulli equation 4. b) Calculated velocity profile in the opening and profile derived from the Bernoulli equation 4. c) and d) are as a) and b) but for the three-dimensional case. Note that in Figure 10 the z axis is horizontal and the velocity sign is changed, as opposed to Figure 9b; "2D" means "two-dimensional" and "3D" means "three-dimensional."*

Small Domain Calculation of Large Opening Cases

It is not desirable for a simulation application to have a huge calculation domain outside the zone of interest. Therefore, it was considered how small an outer zone can be chosen while still yielding the same results in the zone of interest. This investigation has been done up to now for the two-dimensional free convection large opening case of Figure 6a without wind plus several other cases.

Large Opening, Two-Dimensional In the large opening case described above, it was found that the calculation domain must be extended to the outside by about half the opening height in order to yield the same flow pattern in the room at the window opening and in the plume region inside the calculation domain ("same" means within about 2% of velocity direction and magnitude).

Figure 12 shows the flow pattern of the "reduced domain" case, LO38, compared with the corresponding part of case LO34. Case LO38 is two-dimensional, also, extends with 21 × 16 cells over 6.5 m × 3.2 m (21 ft × 10.5 ft), and is otherwise equal to case LO34. Also, pressure, temperature, and turbulence fields are almost equal in both cases.

Therefore, very good results for the airflow through a large opening can be obtained, compared to cases with a very enlarged calculation domain, without extending the computation domain very much. How much this extension should be depends a little bit on the problem. In the "reduced domain" case with wind, the area exterior to the door must be extended to almost 10 m outside the door so the same wind velocity profile near the bottom can build up as in the reference case LO42. Of course, a 1-m extra domain outside the door should be sufficient if the "real" wind velocity profile can be provided as the boundary condition (which is probably seldom the case).

Generally speaking, in wind cases, the necessary extra domain depends on the building height or on a knowledge of the local wind velocity profile. In free convection large opening cases, the length of the extra domain can be set as about half the opening height.

Large Opening, Three-Dimensional The same method was also applied to the three-dimensional cases with the same positive results.

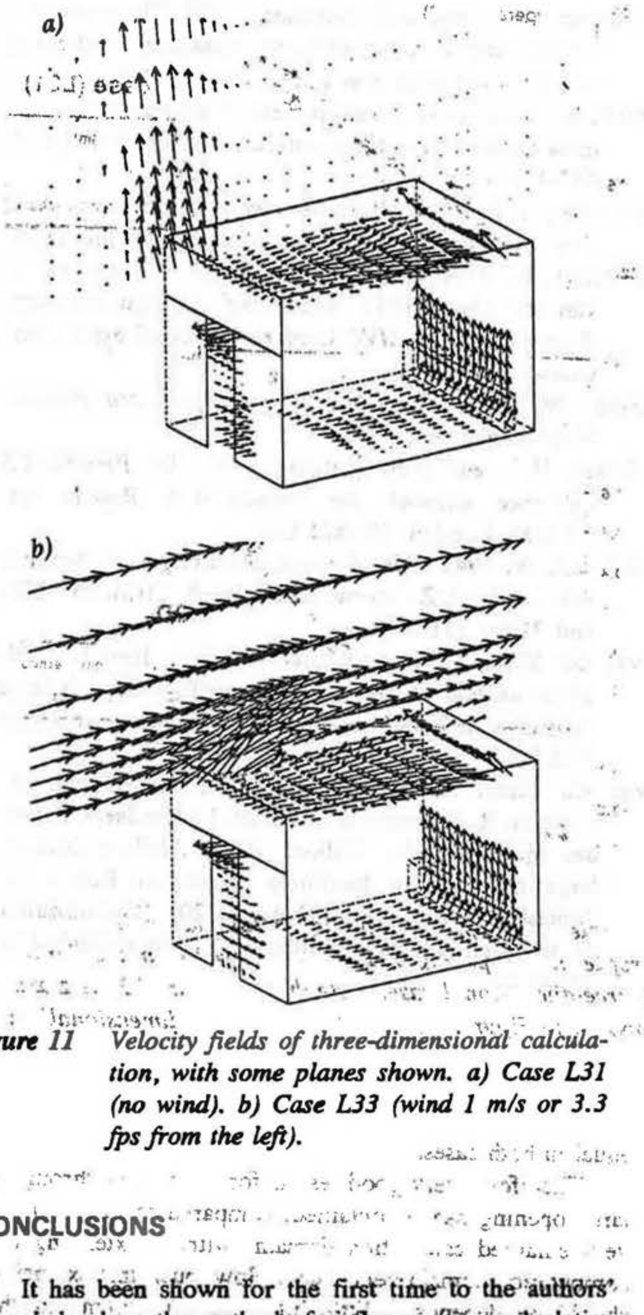

Figure 11 *Velocity fields of three-dimensional calculation, with some planes shown. a) Case L31 (no wind). b) Case L33 (wind 1 m/s or 3.3 fps from the left).*

CONCLUSIONS

It has been shown for the first time to the authors' knowledge how the method of airflow pattern calculation can be applied to the coupling of indoor airflow to the outdoor flow through large openings that allow bidirectional flow. In the described situations, warm air leaves a room through the upper part of a large opening and rises as a thermal plume, with and without external wind.

In particular, the following has been pointed out:

- CFD can be used for plume calculations. The modeling of a free plume generated by a heated point on a plane shows very good agreement with the analytical predictions for the peak velocity, peak temperature, and velocity profile half-width as functions of the height.
- CFD calculation of large openings in buildings shows very good agreement with experimental velocity profiles obtained from single-sided ventilation measurements. The numerical doorway velocity profiles are more realistic than those obtained by an analytical Bernoulli model. CFD reflects a higher complexity of real data and can therefore also be of great help in validating simplified models.
- Care must be taken, however, if two-dimensional approximations are used. In order to simulate the correct behavior of the throughput of air through a door or window as a function of wind speed, a three-dimensional calculation must be used with higher computation expenses.
- To save computation expenses, it is demonstrated in this paper that excellent results can be obtained for cases with large openings in buildings for a computation domain that is extended only a little to the outside of the room. In free convection cases, the length of the extra domain can be set to about half the opening height. In wind cases, the extra domain necessary depends on the building height or on knowledge of the local wind velocity profile.

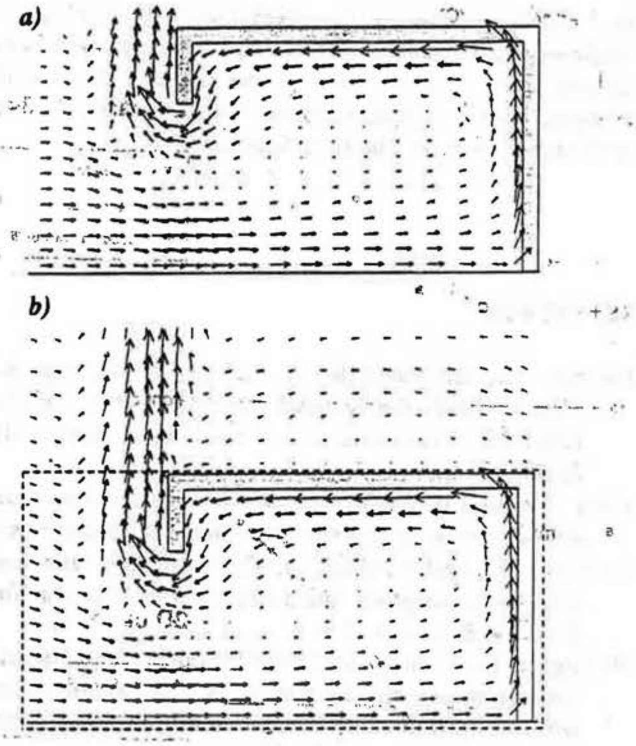

Figure 12 *Velocity field comparison of small and large domain calculation. a) Case LO38; the velocity arrows indicate the full calculation domain used. b) Case LO 34; zoomed view of Figure 7a; the dashed rectangle shows the calculation domain used in Figure 12a.*

ACKNOWLEDGMENTS

This work was financially supported by the Swiss Federal Office of Energy (BEW). The authors are grateful to J. Zhiang, now at the Department of Mechanical Engineering, Concordia University, Montreal, Quebec, Canada, and Q. Chen, now at the Institute of Applied Physics, TNO Industrial Research, Delft, Netherlands, for preliminary work and helpful discussion.

REFERENCES

Bienfait, D., L. Vandaele, J. van der Maas, and R. Walker. 1991. Single-sided ventilation. *Proc. of the 12th AIVC Conference*, 24-27 September, Ottawa, 1: 73-98.

Chen, C.J., and W. Rodi. 1980. *Vertical turbulent buoyant jets—A review of experimental data*. Pergamon Press.

De Gids, W., and H. Phaff. 1982. Ventilation rates and energy consumption due to open windows. *Air Infiltration Review* 4: 4-5.

Haghighat, F., Z. Jiang, and J.C.W. Wang. 1989. Natural convection and airflow pattern in a partitioned room with turbulent flow. *ASHRAE Transactions* 95(1): 600-610.

Jiang, Z., F. Haghighat, and J.C.W. Wang. 1992. Thermal comfort and indoor air quality in a partitioned enclosure under mixed convection. *J. of Building and Environment*, June.

Kofoed, P., and P.V. Nielsen. 1990. Thermal plumes in ventilated rooms. *Proceedings of ROOMVENT '90*, Oslo, Norway.

Launder, B.E., and D.B. Spalding. 1974. The numerical computation of turbulent flows. *Computer Methods in Applied Mechanics and Engineering* 3: 269-289.

Mahajan, B.M. 1987. Measurement of interzonal heat and mass transfer by natural convection. *Solar Energy* 38: 437-446.

Patankar, S.V. 1980. *Numerical heat transfer and fluid flow*. Washington, DC: Hemisphere Publishing. Corp.

Pelletret, R., F. Allard, F. Haghighat, G. Liebecq, and J. van der Maas. 1991. Modelling of large openings. *Proc. of the 12th AIVC Conference*, 24-27 September, Ottawa, 1: 99-110.

Rodi, W. 1982. *Turbulent buoyant jets and plumes*. Pergamon Press.

Rosten, H.I., and D.B. Spalding. 1987. *The PHOENICS reference manual*, for version 1.4. Report No. TR/200. London: CHAM Ltd.

Schmidt, W. 1941. Turbulente Ausbreitung eines Stromes erhitzter Luft. *Z. angew. Math. Mech.* 21(5): 264-278, and 21(6): 351-363.

van der Maas, J., C.A. Roulet, and J.A. Hertig. 1989. Some aspects of gravity-driven airflow through large apertures in buildings. *ASHRAE Transactions* 95(2): 573-583.

van der Maas, J. (ed.), F. Allard, F. Haghighat, G. Liebecq, R. Pelletret, D. Bienfait, L. Vandaele, J. van der Maas, and R. Walker. 1992. Airflow through large openings in buildings. Technical Report of Subtask 2 of the IEA/ECB-Annex 20, "Optimization of Airflow Patterns in Buildings." To be published in winter 1992.