Tamkang Journal of Science and Engineering, Vol. 6, No. 3, pp. 151-157 (2003)

Computational Fluid Dynamics Simulation of the Progress of Fire Smoke in Large Space, Building Atria

Cheng-hsin Chang[1], David Banks[2] and Robert N. Meroney[3]

[1]*Department of Civil Engineering*
Tamkang University
Tamsui, Taiwan 251, R.O.C.
E-mail: cc527330@mail.tku.edu.tw
[2]*Cermak Peterka Petersen (CPP) Inc.*
Fort Collins, CO 80523, U.S.A.
[3]*WEFL, Department of Civil Engineering*
Colorado State University
Fort Collins, CO 80523, U.S.A.

Abstract

Conventional fire and smoke control systems use pressure differences across small openings and cracks in physical barriers as a means to restrict smoke propagation from one space to another and water-spray curtains to diminish or eliminate fire and smoke. Most fire codes of Unite State of American depend upon the National Fire Protection Association (NFPA), guidebooks [1,2]. In turn these propose the use of simple zone models that solve conservation of mass and energy in a control-volume sense for each zone. One weakness of zone modeling is that momentum conservation is only captured through use of loss coefficient at openings. The strength of zone models is that they are very fast compared with computational fluid mechanics (CFD) based models. Atria, covered shopping malls, convention centers, airport terminals, sport arenas, and warehouses are examples of large spaces for which these conventional zone-model approaches are not always effective [3]. CFD, sometimes called “field-modeling” in the fire community to distinguish it form zone-modeling, has an unparalleled potential as an engineering estimator of fire consequence in atria since it permits specification of momentum conservation as well as much finer spatial and temporal resolution of the fire physics [4]. In addition CFD approaches provide a link between outside building weather conditions and fire and smoke development. This paper will discuss the results of calculations for an example building atrium based on zone (ASMET [5]) and field (FLUENT [6] and FDS [7,8] CFD)-based models.

Key Words**:** Computational Fluid Dynamics, Fire Smoke Control, Building Atria

1. Introduction

Since atria do not have the compartmentalization that has traditionally been a major contribution to fire protection, smoke management rather than elimination by water sprays is of particular importance. The three management methods available are smoke filling, gravity venting, and smoke exhaust. Issues include prestratification preventing smoke from reaching ceiling mounted detectors or vents, smoke detection and number and placement of exhaust vents. The Uniform Building Code (UBC) specifies constraints on smoke barriers, pressurization methods, and equipment characteristics, but the choice of equipment must depend upon the actual dynamics of the fire, kinematics of the smoke plume, and specific geometry of the atrium. The goal of smoke management is to provide for a tenable environment for the evacuation or relocation of occupants. Typically, the approach is to restrict any smoke spread to a plume rising from the fire and to a smoke layer just under the ceiling of the large space. The idea is to maintain a lower "smoke-free" layer for some specified time in which occupants can safely exit and fire fighters can see to contain and eliminate the fire.

Zone modeling predicts the vertical descent of a well-mixed smoke layer continuously supplied from a fire plume. The method assumes there is a large volume available in which turbulence and relatively small lateral velocities distribute the heated gases into a homogeneous mixture. The analytic relations used are based on laboratory and field scale fires for a limited range of volume configurations typically unimpeded by interior stairways or other architectural elements. The fire initial condition relations used in this study are those found in the 1988 California Building Code (CBC).

CFD provides a design technique to examine the relative merits of various exhaust schemes and alternative location of gravity vents or exhaust fans. The FLUENT CFD suite used provides an unstructured mesh building preprocessor and a control volume based solver containing a variety of turbulence model options that will be described in the presentation. The NIST-FDS program is also a control volume based solver with large eddy simulation (LES) turbulence models but uses a structured grid mesh.

2. Numerical Simulation

2.1 CFD Code Descriptions

ASMET (Atria Smoke Management Engineering Tools) consists of a set of equations and a zone fire model for analysis of smoke management systems for large spaces such as atria, shopping malls, arcades, sports arenas, exhibition halls and airplane hangers.

FLUENT is a very flexible and powerful modeling tool that permits solution over either structured or unstructured grids. The user can include heat exchangers, fans, etc. It can deal with steady or unsteady flow, laminar or turbulent, movable walls, deforming walls, etc. There are many different turbulent models to choose from including Kappa-epsilon and LES. The program allows the user to specify up to 20 separate chemical reactions (either heterogeneous or homogeneous in nature), solve for temperatures, radiation, combustion, and particle or spray combustion, etc. For these tests, the kappa-epsilon turbulence model was used, and steady state solutions were obtained.

NIST-FDS (NIST Fire Dynamics Simulator) is a computational fluid dynamics (CFD) model of fire driven fluid flow created by the US Building and Fire Research Laboratory which solves numerically a form of the Navier-Stokes equations appropriate for low-speed, thermally-driven flow with an emphasis on smoke and heat transport from fires. The formulation of the equations and the numerical algorithm are contained in the *Fire Dynamics Simulator* — *Technical Reference Guide 3*. FDS models turbulent buoyant motion by solving the full set of momentum and energy equations using a LES (Large Eddy Simulation) model for the time dependent motion. The use of the term "large eddy simulation" refers to the idea that the convective fluid motion should be simulated at the finest length and time scales allowed by a given computational grid. This program has been validated against experimental data for isolated plumes, heptane spray burner experiments, and even racks of containers stored in warehouse configuration. Results are presented via SMOKEVIEW, which visualizes FDS computed data by animating time dependent particle flow, slice contours and surface boundary contours.

Figures 1 and 2 show the sample geometries of the CFD codes, Fluent and NIST-FDS.

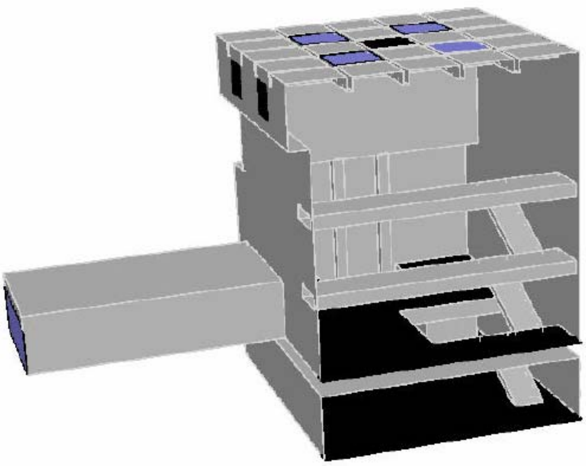

Figure 1. Sample fluent geometry, with south wall removed, blue surfaces are air intakes and exhausts.

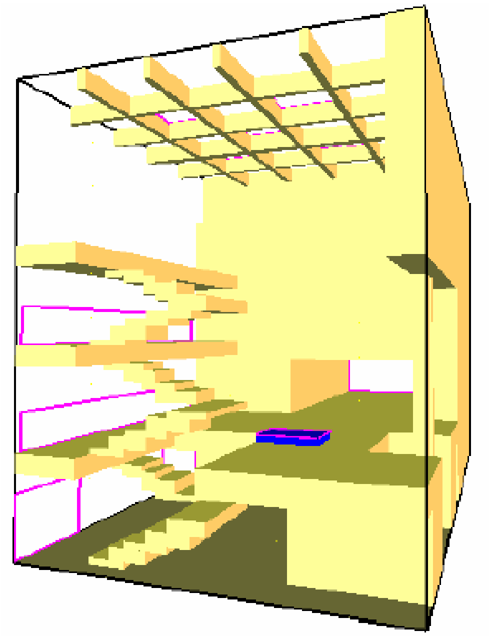

Figure 2. Sample FDS geometry, pink boxes indicate air inlets or outlets. Blue box in center is fire.

2.2 Building Case Study

A case study has been chosen for discussing that considers an actual atrium that includes an exposed interior staircase, suspended walkways, open lateral hallways and lobby, ceiling skylights and wall exhaust fans, and other architectural features which made the problem more complex than a simple box-shaped volume. These cases are summarized in Table 1. The CFD calculations were also compared to the predictions of the simple zones models found in the 1998 California Building Code (CBC) and related National Fire Protection Association (NFPA) guide books. As atrium size goes the case study was small (~17 m^3), but it was within the range typically evaluated as atrium spaces. Continuous fire sources of 5276 kW and 2100 kW were considered as suggested by the NFPA 92B guide. Fire sources were placed alternatively in the lobby, basement and first floor regions of the atrium. Steady state and temporal calculations were performed to determine fire kinematics, temperature distributions and smoke descent levels.

The goal of the smoke management system examined in this study is to maintain tenable conditions in the atrium during a design fire via the exhaust method. The essentially means that a smoke layer is allowed to build up beneath the atrium ceiling, but is exhausted at such a rate that the smoke layer stays at a constant level above any walking surfaces. Section 905.5 of the 1998 CBC governs the design of such systems, and stipulates that

- "The height of the lowest horizontal surface of the accumulating smoke layer shall be at least 10 feet above any walking surface within the smoke zone."
- "Provisions shall be made for natural or mechanical supply of outside air to make up an equal volume of the air exhausted at flow rates not to exceed 200 feet per minute toward the fire."

Table 1. The summary of CFD simulation cases

Case	Description of Venting Method	Power of Fire
1	200,000 cfm being removed from the North side wall via mechanical exhaust	5275 kW
2	200,000 cfm being removed from the ceiling wall via mechanical exhaust	2100 kW
3	1300 ft^2 natural venting opening in ceiling	5275 kW
4	1300 ft^2 natural venting opening in ceiling, 200,000 cfm being removed from the north side wall via mechanical exhaust	5275 kW
5	add the hanging porous curtains across the ceiling	5275 kW

In this situation, it is the final condition of the atrium smoke layer and the make up air flow rates which are of interest (rather than, for example, the rate of descent of the smoke layer, which is significant if the smoke is not going to be exhausted and a timed evacuation is expected. As a result, time dependent parameters such as the moment of activation of the exhaust vents or the growth of the fire were not simulated. In FDS, the fire size was quickly increased to the full, final value, and the smoke control system was assumed to begin operating immediately, so that the code could quickly reach a steady state. As a result, the final steady-state FDS results are emphasized in this paper, since the first 30-60 seconds of the simulation are not considered to realistically representing the initial stages of the fire. Fluent was also converged to a "steady-state" solution without regard to realistic fire growth conditions.

3. Results and Discussion

3.1 Simulation Case 1

Temperature contour produced by FDS for Case 1 are shown in Figure 3. There are five mechanical exhaust vents along the top of the north wall, of which two are visible. (The north wall was indicated to be the preferred exhaust vent location for mechanical venting during the initial consultation phase of this study.) Make up air is being supplied through two 125 ft^2 openings in the basement's south wall, a single 250 ft^2 opening in the level one south wall, and another 250 ft^2 opening in the west wall at the lobby entrance. The total of 750 ft^2 of make up air supply openings was based upon the anticipated need to supply a little less than 150,000 cfm of make up air (the air expands to 167,000 cfm of smoke as a result of the heat of the fire) while meeting the maximum flow speed requirement of 200 fps. This is in part because the CBC formulas used to calculate these volume flow rates, like those in NFPA92B, do not account for the diameter of the fire as recommended by Klote and described in NFPA 204. When the 16 ft. fire diameter is included in the calculations (through a virtual origin term), the smoke production rises to 208,000 cfm, and the make up air supply requirements rise to 180,000 cfm. Since this larger number is expected to more accurately reflect the smoke produced by the simulated fire, an exhaust rate of 200,000 cfm was used in the simulations.

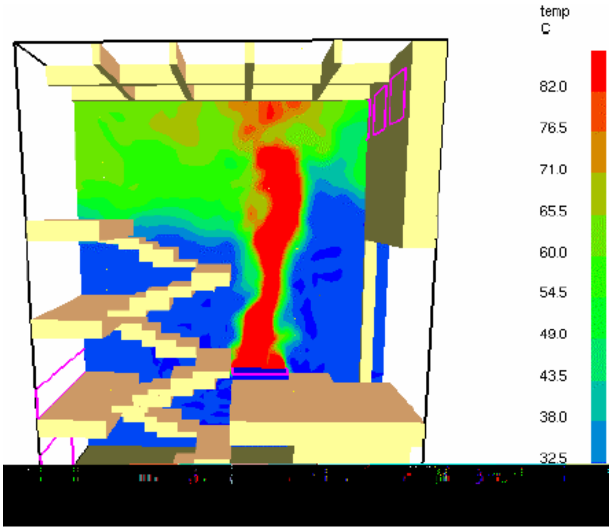

Figure 3. Temperature contour after 100 seconds of simulation time in FDS

The start of the smoke layer roughly corresponds to the height at which the temperature begins to rise rapidly from the ambient outdoor temperature (selected to be 27 °C, or 80 °F) to the smoke layer maximum temperature (in this case, 68 °C or 154 °F; the higher temperatures in the center of the room are considered plume temperatures). The precise height at which the smoke layer interface is considered to have begun can be debated. It has been suggested that the formula:

$$Tn = Cn(Tmax - Tb) + Tb$$

be used, where

Tn = interface height temperature

$Tmax$ = maximum temperature (at the ceiling)

Tb = temperature near bottom of atrium

Cn = interpolation constant (typically assumed to be 0.15 or 0.2)

In this case, Tn = 34 °C or 94 °F, which corresponds to the first level of color contour change, so the smoke layer interface is just below the walking surface of the level three walkway. On the north side of the atrium, the smoke layer interface is much higher, and relatively cool air (45 °C) can be seen to enter the exhaust vent. This phenomenon is referred to as “plugholing”, and it reduces the efficiency of the smoke removal system, since the exhaust system is not drawing all of its air from the hot smoke layer, but is taking some air directly from the cooler room air. As a result, increasing the flow rate into these vents will only have a limited benefit for increasing the smoke layer height, since this higher flow rate will exacerbate the plugholing problem. CFD runs performed for this study but not included in Table 1 have confirmed this.

3.2 Simulation Case 2

Since the velocity of the ceiling jet varies with the heat release rate or power of the fire, Case 2 was repeated using a 65 ft^2, 2100 kW fire. This is only expected to reduce the ceiling jet velocity 2 by 25%, however, and the simulation shows the continued presence of a warm southward circulation along the east wall, bringing smoke to the level 3 walkway (see Figure 4). Figure 5 shows pathlines of particles exiting SW ceiling vent, as predicted by Fluent.

3.3 Simulation Case 3

Case 3 uses an inlet next to the level 3 walkway to clear smoke from the walkway. The placement of an inlet near the smoke layer interface is in general not recommended however, because the mixing thus induced adds mass to the smoke layer. One further danger of this technique is that winds external to the building could induce a negative pressure on the south wall, which would pull smoke out of this opening. In fact, the pressures

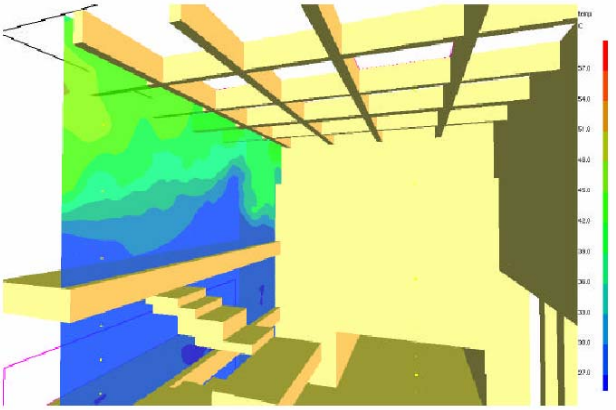

Figure 4. Temperature contours above the level 3 walkway (FDS)

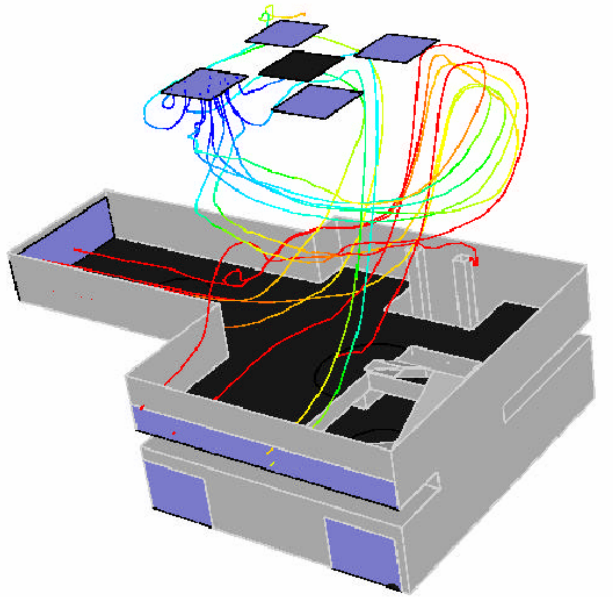

Figure 5. Pathlines of particles exiting SW ceiling vent, as predicted by Fluent

created by even mild (15 mph) winds, can easily dominate the forces induced by-buoyancy.

The pressures will in general increase the suction out of the ceiling for natural ventilation, but in this case, the potential exists for downdrafts as a result of the adjacent, taller structures to the north and East. External winds can also induce a crossflow between make up air inlets. For example, a wind from the south would induce a negative pressure on the west side, which would pull smoke into the lobby in all of the configurations tested in this study.

3.4 Simulation Case 4

In general, a combination of natural and mechanical venting offers considerable complications, as there is a potential for the mechaniccal exhausts to draw air directly from the natural vents. However, with CFD or physical modelling, such situations can be evaluated. Case 4 places 500 ft^2 of natural ventilation along the middle of the ceiling and 500 ft^2 of ventilation at the south end of the ceiling, above the level 3 walkway. Mechanical exhausts vented 300,000 cfm through five vents in the top of the North wall. For the first 60 seconds of the simulation, the openings above the level 3 walkway act as inlets, for t = 40 seconds. The flow coming down through these openings meets the southerly ceiling jet flow, and forces this flow down to the walkway surface. While this reduces the temperature above the level 3 walkway, it also causes increased smoke layer mixing and mass production. After 60 seconds, the south ceiling vents begin to act primarily as flow exits, and the east wall deflected ceiling jet follows its usual course toward the walkway, then flows out these openings, as shown in Figure 6. The openings in the middle row act intermittently as exits and sources; when they act as sources, this also increases the smoke mass. The net effect is that the temperature increase above the level 3 walkway is only 20 to 25 °F, but the walkway is smoke filled (see Figure 7).

While the solution at 40 seconds is not considered accurate (since, as noted in the introduction, the fire growth is not modeled realistically), it underlines the inherent instability of this flow situation. It is not unrealistic to presume that the flow out of the south ceiling vents could be reversed if conditions in the room changed slightly, such as a change in the fire strength or the inflow rates at the various lower levels make up air inlets. Regardless of the direction of the flow through these vents, however, the

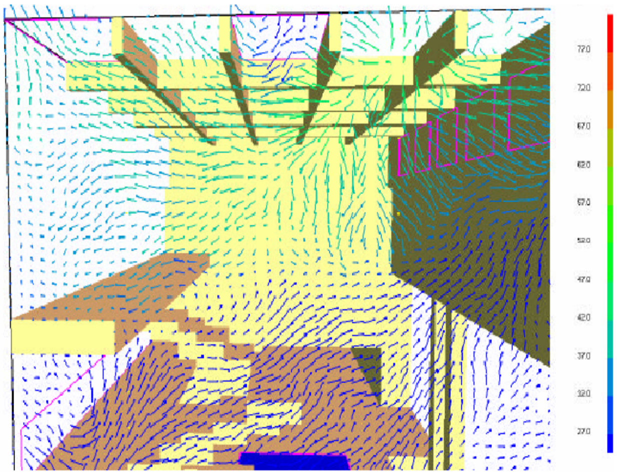

Figure 6. Flow vectors along east wall after 120 seconds of simulation (FDS), Case 4

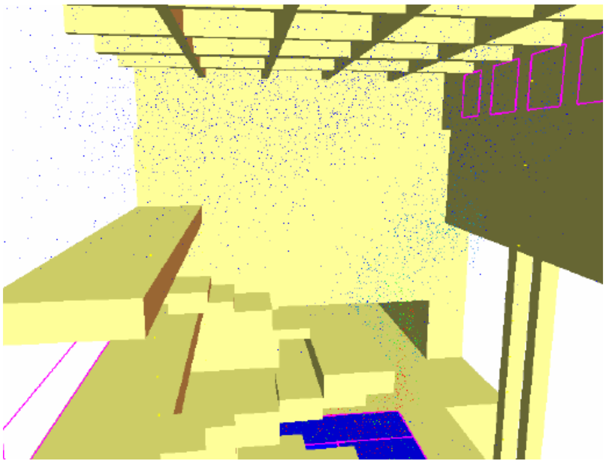

Figure 7. Case 4 smoke particles after 120 seconds of simulation (FDS)

configuration of Case 4 does not prevent smoke from accumulating near the surface of the level 3 walkway.

3.5 Simulation Case 5

Since none of the configurations tested succeeded (from case 1 to case 4) in keeping smoke 10 feet above the level 3 walkway. Case 5 consider add the hanging porous curtains across the ceiling. Figure 8 shows the mitigation devices across the ceiling. In case 5, smoke would descend significantly below safe levels due to impingement of the fire plume against the ceiling that produced lateral jets that were, in turn, deflected downward by the atrium sidewalls. ASMET, simple zonal model simulated the following consideration and condition setups, (1) the fire height 0.2 m; (2) room height 21.9 m; (3) floor area 252 m^2; (4) growth rate ultra-fast (0.187 kW/sec). The results were showed in Figure 9.

4. Conclusion

The summaries of case 1 to case 4 are:

1. None of the configurations tested succeeded in keeping smoke 10 feet above the level 3 walkway.
2. The ceiling jet, which results from the plume reaching the ceiling with velocities of 750 to 1000 fpm curves downward as it, reaches the walls.
3. This downward momentum establishes a lower limit to the smoke layer depth, which is primarily a function of the fire strength and the atrium dimensions.
4. Increasing the exhaust flow rate or the number of natural exhaust vents does not ameliorate the resulting smoke layer depth. The resulting smoke layer depth is not ameliorated by a reasonable decrease in the fire strength.

Figure 8. The mitigation devices across the ceiling

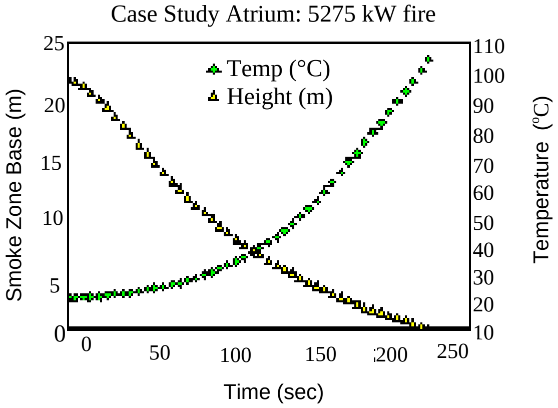

Figure 9. Temperature and Height of smoke by ASMET

The zone model calculations using ASMET suggests that an exhaust rate of 200,000 cfm would be adequate to limit smoke descent to regions 10 feet above any walking surface within the smoke zone. But both the FLUENT and FDS models demonstrate that smoke would descend significantly below safe levels due to impingement of the fire plume against the ceiling (Case 5) that produced lateral jets that were, in turn, deflected downward by the atrium side walls. Consideration of a wide variety of conventional inlet ventilation, ceiling skylight, and wall exhaust fan alternatives did not reveal a safe solution for this dilemma! Architectural changes to the ceiling region including the use of hanging to reduce lateral jetting can mitigate the problem.

Typically, the wind field outside the building is not considered when specifying fire hazard systems. Separate calculations that included the presence of a simple wind field impinging on the building exterior reveal that such conditions can significantly alter the trajectory of the fire plume and internal circulations; hence, simple zone models are not suitable for flow fields subject to external perturbations.

References

[1] *"NFPA 92B*, Smoke Management Systems in Malls, Atria, and Large Areas, 2000, "National Fire Protection Association, Quincy, MA, U.S.A. (2000).

[2] "NFPA 204M, Guide for Smoke and Heat Venting. 1998,*"* National Fire Protection Association, Quincy, MA, U.S.A. (1998).

[3] Klote, J. H. and Milke, J. A., "Smoke Management in Atria and Other Large Spaces," *Design of Smoke Management Systems*, ASHRAE, Atlanta, GA, U.S.A., pp. 101-134 (1992).

[4] Tieszen, S. R., "On the Fluid Mechanics of Fires," *Ann. Rev. Fluid Mech.* Vol. 33, pp. 67-92 (2001).

[5] Klote, J. H., "Method of Predicting Smoke Movement in Atria With Application to Smoke Management," *NISTIR 5516*, National Institute of Standards and Technology, Gaithersburg, MD, U.S.A. (ASMET: Atria Smoke Management Engineering Tools).

[6] "FLUENT 5 Users Guide," Fluent Incorporated, Lebanon, NH, U.S.A., see http://www.fluent.com (1998).

[7] Rehm, R. G., McGrattan, K. B., Baum, H. R. and Simiu, E., "An Efficient Large Eddy Simulation Algorithm for Computational Wind Engineering: Application to Surface Pressure Computations on a Single Building," *NISTIR 6371*, Building and Fire Research Laboratory, NIST, Gaithersburg, MD, U.S.A., Website: http://fire.nist.gov (FDS: Fire Dynamics Simulator).

[8] McGrattan, K. B., Baum, H. R. and Rehm, R.G., "Large Eddy Simulation of Smoke Movement," *ASHRAE Transactions*, Vol. 105, Part 1, Paper No. CH-99-1-4, 11 (1999).

Manuscript Received: Jan. 10, 2003
Accepted: Jun. 10, 2003