Computational Fluid Dynamics Simulation of the Progress of Fires in Building Atria

David Banks[1], Ph.D., Cheng-Hsin Chang[2], Ph.d & Robert N. Meroney[3], P.E. Ph.D.

Introduction:

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 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 consider a comparison of 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.

Principal Concepts:

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 pre-stratification 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 pre-processor and a control volume based solver containing a variety of turbulence model options

[1] Cermak Peterka Petersen, Inc., Fort Collins, CO
[2] Graduate Institute of Environmental Engineering, National Taiwan University, Taiwan
[3] Civil Engineering Department, Colorado State University, Fort Collins, CO

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.

Building Case Study:
A case study has been chosen for comparison 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. As atrium size goes the case study was small (~17 m cube), 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.

Results:
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 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 porous curtains 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 (2000), *NFPA 92B, Smoke Management Systems in Malls, Atria, and Large Areas.* 2000*, National* Fire Protection Association, Quincy, Mass.
2. NFPA (1998), *NFPA 204M Guide for Smoke and Heat Venting.* 1988, National Fire Protection Association, Quincy, Mass.
3. Klote, J.H. and Milke, J.A., "Smoke management in atria and other large spaces," Chapter 10 of *Design of Smoke Management Systems,* ASHRAE, Atlanta, GA, 1992, pp. 101-134.
4. Tieszen, S.R., "On the Fluid Mechanics of Fires," *Ann. Rev. Fluid Mech. 2001*, Vol. 33: pp. 67-92.
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. (ASMET: Atria Smoke Management Engineering Tools).
6. FLUENT 5 Users Guide, Fluent Incorporated, Lebanon, NH, 1998. See http://www.fluent.com
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. 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 1999*, Vol. 105, Part 1, Paper No. CH-99-1-4, 11 pp.