International Journal on Engineering Performance-Based Fire Codes, Volume 1, Number 3, p.183-187, 1999

EXPERIMENTAL AND NUMERICAL STUDY OF SMOKE CONDITIONS IN AN ATRIUM WITH MECHANICAL EXHAUST

G.V. Hadjisophocleous and G. Lougheed
Fire Risk Management Program, Institute for Research in Construction, National Research Council Canada
Ottawa, Ontario, Canada, K1A 0R6

ABSTRACT

Full-scale experiments and numerical modelling using Computational Fluid Dynamics techniques were employed to investigate atrium smoke exhaust in physical model studies. These investigations are part of a joint research project between the American Society of Heating Refrigerating and Air-Conditioning Engineers Inc. (ASHRAE) and the National Research Council of Canada (NRCC). The objective of these studies is to develop input to design guides for atrium smoke management systems. This paper presents initial results from this study.

The physical tests were done in a specifically constructed compartment equipped with a smoke exhaust system and instrumentation for measuring temperatures, CO_2 concentrations and velocities. Fire was modelled using propane burners capable of producing fires with different intensities and areas. The numerical simulations were done using Computational Fluid Dynamics (CFD) models. A comparison between the experimental and predicted temperatures and CO_2 concentrations indicated that the CFD model can predict the conditions in the room, as well as the depth of the hot layer.

1. INTRODUCTION

An atrium within a building is a large open space created by an opening, or series of openings, in floor assemblies, thus inter-connecting two or more storeys of a building. This design feature has gained considerable popularity, mainly because of its visual appeal. The sides of an atrium may be open to all floors, to some of the floors or closed to all or some of the floors by unrated or rated fire-resistant construction. As well, there may be two or more atria within a single building, all interconnected at the ground floor or on a number of floors.

An atrium, by interconnecting floor spaces, violates the concept of floor-to-floor compartmentation, which is intended to limit the spread of fire and smoke from the floor of fire origin to other storeys inside a building. With a fire on the floor of an atrium or in any space open to it, smoke can fill the atrium and connected floor spaces. Elevators, open stairs and egress routes that are within the atrium space can also become smoke-laden.

Protecting the occupants of a building from the adverse effects of smoke in the event of a fire is one of the primary objectives of any fire protection system design. Achieving this objective becomes more difficult when dealing with very large spaces, such as an atrium or an indoor sports arena, where a large number of occupants may be present and the compartment geometry may be complex. Because of these difficulties, model building codes place restrictions on the use of atrium spaces in buildings. Some of the requirements that are commonly applied in codes for buildings with atria include: the installation of automatic sprinklers throughout the building; limits on combustible materials on the floor of an atrium the installation of mechanical exhaust systems for use by firefighters and; the provision of smoke management systems to maintain tenable conditions in egress routes.

Atrium smoke management systems have become common in recent years and design information for these systems is provided in NFPA 92B [1] and Klote and Milke [2]. There are, however, a number of situations that may detrimentally impact the effectiveness of the smoke management system. These include obstructions in the smoke plume (Hansel and Morgan [3]) or the formation of a pre-stratification layer in the atrium (Klote [4]). In the former case, smoke may be directed to adjacent spaces or mixed with the air within the zone in which tenable conditions are required. In the latter case, the smoke produced by the fire may not reach the ceiling where it could be exhausted. Also, in this latter case, smoke build-up could occur at a height at which it can migrate into the communicating spaces.

Under some conditions, another phenomenon may impact the effectiveness of a smoke management system: air from the lower (cold) layer mixing with smoke in the upper layer as it is being exhausted by the smoke management system. This phenomenon reduces the effectiveness of the smoke management system (Hinckley [5]). As a result, the clear height in the atrium is reduced and people in some spaces may be exposed to smoke and toxic fire gases. This phenomenon is referred to as "plugholing" and investigations on this have been carried out using natural venting systems (Morgan and Gardiner [6], Spratt and Heselden [7]). To study the effects of plugholing on a mechanical exhaust system used for atrium smoke management, a joint research project was initiated by ASHRAE and NRCC in 1995. This project includes both physical and numerical modelling of an atrium smoke management system. The objective of the project is to develop methods with which designers can account for the mixing of cold air with the smoke exhaust. These methods will provide a basis for the design of cost-effective smoke management systems that will meet design expectations.

This paper presents the initial results of physical model studies of an atrium space with mechanical exhaust. It includes a comparison between experimental data and Computational Fluid Dynamics (CFD) model predictions of the conditions in the atrium space.

2. DESCRIPTION OF PHYSICAL MODEL

The experimental facility used for this study is a large compartment with dimensions of 9 m x 6 m x 5.5 m height. All interior wall surfaces of the compartment were insulated using 25 mm thick rock fibre insulation. Insulation was used for two reasons: first, to protect the walls of the facility so that high gas temperatures could be attained during the tests and, second, to provide a better boundary condition for the CFD runs.

A fan was used to supply fresh air into the compartment through openings in the floor around the walls. The openings were designed to maintain the velocity of the incoming air to less than 1 m/s for the maximum airflow expected which was between 2 and 4 m^3/s. These openings had a width of 0.1 m, and a total length of 22.8 m.

Thirty-two exhaust inlets with a diameter of 150 mm were located in the ceiling of the compartment. These inlets were used to extract the hot gases from the compartment during the tests. All exhaust ducts were connected to a central plenum. A 0.6 m diameter duct was used between the plenum and an exhaust fan. By using multiple exhaust inlets, smoke exhaust system parameters such as total area of exhaust inlet, velocity at the inlets and exhaust inlet location relative to the ceiling and the fire could be readily investigated.

The exhaust system included a two-speed fan with nominal capacities of 3 and 4 m^3/s. The actual volumetric flow rate produced by the fan in a test depended on a number of factors including smoke temperature and the number of exhaust inlets used. Therefore, the volumetric flow rate in the main duct was continuously measured throughout a test.

A square propane sand burner was used for the fire source. The burner was capable of simulating fires ranging from 15 kW to 1,000 kW with three possible fire areas: 0.145 m^2, 0.58 m^2 and 2.32 m^2. The heat release rate of the fire was determined using two methods. The first method computes the heat release rate from the volume flow rate of propane supplied to the burner. The second method was based on the oxygen depletion method using oxygen concentrations, temperature and, volume flow rate measured in the main exhaust duct.

The room was instrumented with thermocouples and pitot tubes for velocity measurements. In addition, gas inlets were located in the room for extracting gas samples to determine CO_2 concentrations at various locations.

Twelve CO_2 inlets were located at one of the room quarter points at various heights. The CO_2 inlets were connected to two CO_2 analyzers. A set of 19 thermocouples was located at the centre of the room over the propane burner and eight thermocouples were located along the vertical centreline. Six thermocouples were located at 250 mm intervals along a horizontal line at a height of 3 m and another five thermocouples were located at 250 mm intervals along a horizontal line at a height of 4.5 m. Four additional thermocouples were located along a horizontal line near the ceiling at 1 m intervals.

A second set of thermocouples was located below one of the duct inlets. These thermocouples were used to measure the gas temperatures around the exhaust inlets to determine whether fresh air was exhausted from the room. A thermocouple tree was located at the South West quarter point with 15 thermocouples. These thermocouples, together with the CO_2 measurements at the same locations, were used to determine the depth of the hot layer in the room.

The volume flow rate, temperature, CO, CO_2 and oxygen concentrations were measured in the main exhaust duct. These measurements were used to determine the heat release rate of the fire, as well as to calculate the exhaust rate of the ventilation system. A pitot tube and thermocouple, located at the centre of the duct, were used to determine the volumetric flow rate in the duct.

The tests described were conducted over an extended period of time (up to 1 h). The test procedure was as follows:

1. All systems, including the mechanical exhaust system and data acquisition system, were started;
2. The small burner was ignited and the propane flow rate adjusted to provide a low heat release rate fire;
3. All conditions in the test facility, except CO_2 concentrations, were monitored continuously using the data acquisition system;
4. The conditions in the test facility were allowed to stabilize for approximately 15 min producing a steady clear height with upper layer exhaust;
5. The CO_2 concentrations at various heights were measured. This data, along with the temperatures measured at the same heights, were used to determine the height of the smoke layer;
6. The heat release rate was increased and Steps 3-5 were repeated.

Using this test procedure, data could be acquired for several heat release rates under the same test conditions.

3. DESCRIPTION OF NUMERICAL (CFD) MODEL

For numerical modelling, this project used TASCflow, a computer model developed by Advanced Scientific Computing Ltd. [8]. TASCflow is a general three-dimensional Computational Fluid Dynamics model with capabilities in handling laminar and turbulent flows, incompressible and compressible gases, multi-component fluids, porous media, Lagrangian particle tracking, reacting combusting flows, conjugate heat transfer, surface-to-surface radiation, rotating frames of reference and subsonic, transonic and supersonic flows. The grid generation features of TASCflow include the ability to handle non-orthogonal boundary fitted grids, grid embedding and grid attaching.

The fire was modelled using a flamelet model [9,10] and radiation exchange between the hot gases and the surroundings was modelled using the diffusion radiation model of TASCflow [8] with a gas absorption coefficient of 0.15. Turbulent flow was specified using the k-ε model, and all turbulent walls used the log-law treatment. All walls of the room were treated as adiabatic. The boundaries at the outflow openings were defined as mass flow boundaries with a total mass flow rate corresponding to the flow rate used in the experiments. The experimental propane mass flow rate was defined at the fire surface. All fresh air inlets were defined using a constant static pressure.

One quarter of the test room described previously was chosen as the computational domain due to the symmetric flow characteristics observed in both the experimental tests and preliminary model simulations. The whole computational domain was divided into a grid of 21 x 31 x 21 control volumes. Additional grid points were embedded around the fire source and the exhaust inlets to enable better resolution of the solution in these areas. To simulate the small-scale test facility, the total number of grid points for the simulation was 22,496.

4. CFD MODEL RESULTS

The CFD model TASCflow was used to simulate a number of the physical model tests. The primary purpose of the numerical simulations was to determine whether the model could accurately simulate these experiments. Heat release rate and smoke exhaust rate, used for the numerical simulations, were obtained from the experimental data. The experimental heat release rate was used to determine the amount of propane that was consumed in the model and the experimental exhaust rate was defined at the exhaust inlets.

Fig. 1 shows a comparison between the experimental and numerical upper layer temperatures. Overall, the agreement between the two is good with the model slightly over-predicting the temperature. Fig. 2 shows a comparison between the experimental and numerical CO_2 concentrations. As with the temperature results, this correlation shows that the model can predict CO_2 concentrations in the upper layer.

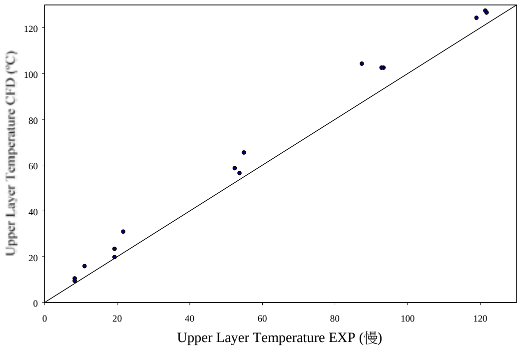

Fig. 1: Correlation of experimental and numerical upper layer temperatures

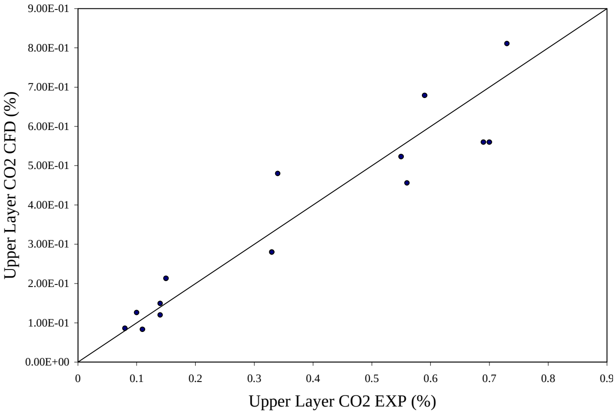

Fig. 2: Correlation of experimental and numerical upper layer CO_2 concentrations

Fig. 3 shows a comparison between the calculated upper layer and exhaust temperatures. Under effective operating conditions, the exhaust system is expected to draw gases from the hot layer only so the two temperatures are expected to be comparable. However, if the exhaust system draws in air from the cold lower layer, then the temperature in the duct will be lower than the temperature of the upper layer. As seen from Fig. 3, the duct temperature for a number of cases is lower than the temperature of the upper layer indicating that cold air is probably drawn into the exhaust system.

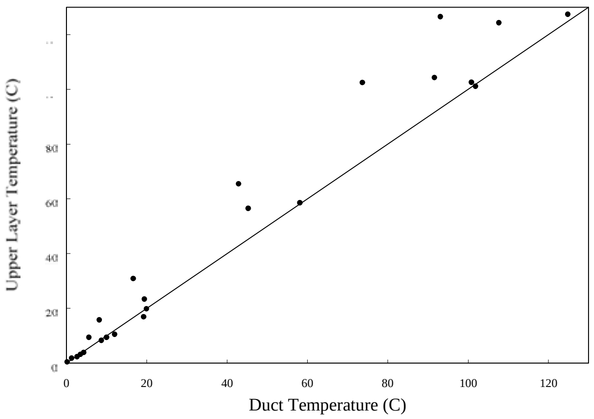

Fig. 3: Correlation of duct and upper layer temperature

Similarly, Fig. 4 shows a comparison between the calculated duct CO_2 concentrations and the CO_2 concentration in the upper layer. This plot confirms what is seen in Figure 3 that cold air is drawn into the exhaust system. The cases which resulted in cold air being drawn into the exhaust system were cases in which the flow rate of hot gases entering the hot layer was less that the fan flow rate. Although the efficiency of the exhaust system can be reduced in these cases, the system was still effective in extracting smoke from the space and maintaining an acceptable clear height. For the cases where there was a balance between the two airflow rates, the exhaust system was found to be not only effective but also efficient. That is, the air drawn into the exhaust system was predominantly from the upper hot layer.

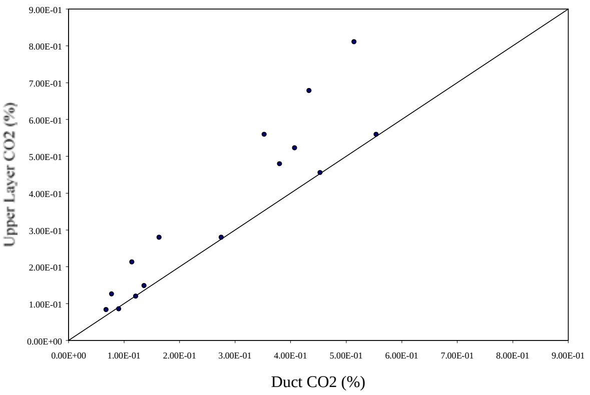

Fig. 4: Correlation of duct and upper layer CO_2 concentration

5. CONCLUSIONS

This paper presents the initial results of physical model studies performed in an atrium space with mechanical exhaust, as well as, a comparison between experimental data and CFD model predictions. It also compared the calculated upper layer temperatures and CO_2 concentrations with the corresponding values in the exhaust duct.

The initial results indicate that, for the atrium studied, the CFD model results compare well with the experimental data for both temperatures and CO_2 concentrations in the upper layer. They also showed that for a number of cases the exhaust system draws in air from the lower layer. These cases, however, were cases where smoke entrainment into the upper layer was less than the air drawn though the exhaust system. For all other cases the exhaust system was found to be efficient.

REFERENCES

1. NFPA 92B, Guide for smoke management systems in malls, atria, and large areas, National Fire Protection Association, Quincy, MA (1995).
2. J.K. Klote and J.A. Milke, Design of smoke management systems, American Society of Heating, Refrigerating and Air-conditioning Engineers, Atlanta, GA (1992).
3. G.O. Hansell and H.P. Morgan, Design approaches for smoke control in atrium buildings, BR-258,

Building Research Establishment, Garston, UK (1994).

4. P.L. Hinckley, Smoke and heat venting, SFPE Handbook of Fire Protection Engineering, National Fire Protection Association, Quincy, MA, pp. 3-160 to 3-17(1995).
5. J.K. Klote, Method of predicting smoke movement in atria with application to smoke management, NISTIR 5516, National Institute of Standards and Technology, Gaithersburg, MD (1994).
6. H.P. Morgan and J.P. Gardiner, Design principles for smoke ventilation in enclosed shopping centres, BR186, Building Research Establishment Garston, UK (1990).
7. D. Spratt and A.J.M. Heselden, Efficient extraction of smoke from a thin layer under a ceiling, Fire Research Note No. 1001, UK Joint Fire Research Organisation (1974).
8. ASC., TASCflow User Documentation, Version 2.3, Waterloo, Advanced Scientific Computing Ltd (1994).
9. N. Peters, “Laminar diffusion flamelet models in non-premixed turbulent combustion”, Progress in Energy and Combustion Science, Vol. 10, No. 3, pp. 319-339 (1984).
10. N. Peters, “Laminar flamelet concepts in turbulent combustion”, 21st Symposium (International) on Combustion/The Combustion Institute, pp. 1231-1250 (1986).