CFD investigation of balcony spill plumes in atria

McCartney, C. J.; Lougheed, G. D.; Weckman, E. J.

Publisher's version / Version de l'éditeur:

12th Annual Conference of the CFD Society of Canada, Combustion and Smoke Management [Proceedings], pp. 823-830, 2004-05-01

CFD investigation of balcony spill plumes in atria

McCartney, C.J.; Lougheed, G.D.; Weckman, E.J.

NRCC-46906

A version of this document is published in / Une version de ce document se trouve dans: 12th Annual Conference of the CFD Society of Canada, Combustion and Smoke Management, Ottawa, ON., May 9-11, 2004, pp. 823-830

CFD Investigation of Balcony Spill Plumes in Atria

C. J. McCartney[1], G. D. Lougheed and E. J. Weckman[2]

[1]Institute for Research in Construction, National Research Council Canada
Ottawa, ON, K1A 0R6, Canada

[2]Department of Mechanical Engineering, University of Waterloo
Waterloo, ON, N2L 3G1, Canada

Email: cameron.mccartney@nrc-cnrc.gc.ca

ABSTRACT

Smoke management in buildings during fire events often uses mechanical ventilation systems to maintain smoke layer elevation above a safe evacuation path. Design of these systems requires accurate correlations for the smoke production rate of the buoyant fire plume. One design issue is the smoke production rate of fire plumes which spill out from a fire compartment, under a balcony and up through an atrium or other large volume. Current engineering correlations for these balcony spill plumes are based on a combination of one-tenth scale test data and theoretical analysis. Questions have arisen over the suitability of these correlations for real-scale designs. A combined program of full-scale experimentation and CFD modeling is being conducted to analyze the accuracy of these correlations.

A full-scale experimental facility was constructed with a 5 m by 5 m by 15 m fire compartment connected to a four-story atrium. Propane fires in the compartment produce balcony spill plumes which form steady-state smoke layers in the atrium. Experimental variables include fire size, compartment opening width, balcony depth and compartment fascia depth. A variable exhaust system was used to achieve various smoke layer heights for each of 100 compartment configurations. Temperature, smoke obscuration and gas concentrations were measured in the compartment, atrium and exhaust system.

The experimental data was used to determine the atrium smoke layer elevation and balcony spill plume smoke production rate for each configuration and fire size. Comparison of this data with zone model results and design correlations for atrium smoke management systems will be performed to evaluate their accuracy.

A CFD model of the experimental facility was implemented using the Fire Dynamics Simulator software (Version 3). Large-eddy simulations of the flow were performed with a constant radiative fraction and an infinitely fast mixture fraction combustion model. A grid sensitivity analysis was performed to optimize the grid design. The CFD model will be validated using the experimental data. The validated model will then be used to extend the experimental results to atrium heights greater than four stories. This model will also be used to compare balcony plume entrainment rates with values available in the literature for buoyant line plumes.

1. INTRODUCTION

Over the past decade, NRC's Fire Risk Management Program (FRM) has conducted a number of joint research projects with the American Society of Heating, Refrigeration and Air Conditioning Engineers (ASHRAE) addressing smoke management systems in atria. This research has concentrated on developing effective solutions for balancing the economic and life safety costs of these systems through the development of fire safety engineering tools. This paper describes a current joint research project investigating smoke production rates of balcony spill plumes.

Until recently, atrium smoke management systems were designed assuming that the fire is located on the atrium floor. It was further assumed that sprinklers limit the size and thus the smoke produced by fires in compartments attached to the atrium. However, some North American model codes now call for consideration of smoke exiting from these attached compartments as balcony spill plumes.

2. BALCONY SPILL PLUMES

Atria with connected compartments are commonly found in office buildings and shopping malls. For a fire in a compartment that opens onto a balcony, smoke can flow under the ceiling and balcony and into the atrium as a balcony spill plume, or BSP (Figure 1). As this buoyant line plume rises into the atrium, air is entrained which increases the smoke volume. In order to size smoke exhaust fans to maintain a safe evacuation path, calculation of the mass flow rate of the balcony spill plume is required.

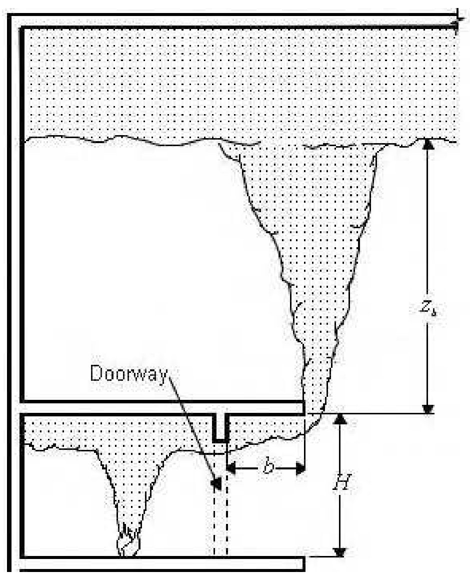

Figure 1. Balcony spill plume concept.

Existing correlations relate BSP mass flow rates to fire compartment dimensions and fire size. These correlations are primarily based on data from one-tenth scale physical models of shopping mall atria [1, 2]. Results from these tests were initially used to develop the BRE method [2], a complex series of equations used to calculate the BSP smoke production rate. Law developed Equation 1 below as a simple correlation for use by designers [3]. Concurrently, Thomas [4] provided an alternative correlation for the data.

The correlation used in North American system design guides ([5, 6]) was developed by Law [3]:

$$m = C_{sp}(Q_c W^2)^{1/3}(z_b + 0.25H) \quad (1)$$

where:

m = plume mass flow rate at elevation z_b (kg/s);
C_{sp} = scaling factor (0.36);
Q_c = fire convective heat release (kW);
W = plume width at balcony elevation (m);
z_b = smoke layer elevation above balcony (m);
H = balcony elevation above top of fuel (m).

There is considerable controversy over the accuracy of the various correlations. Significant variance exists in the BSP smoke production rates calculated using the various methods. These variances are particularly large for atria with high elevations. Smoke management systems designed using inaccurate smoke production rates may result in either reduced life safety or unnecessary costs. Recent research in this area [7, 8] has not reduced the controversy.

The lack of full-scale data for verification of the BSP correlations is a concern. These correlations, developed from scale models of low elevation atria, may not be applicable for atria with higher elevations. Data from full-scale fire experiments is needed to evaluate these correlations and to develop new correlations, if required.

NRC has initiated a joint research project with ASHRAE to determine the accuracy of existing balcony spill plume correlations and provide data for the development of new correlations if required. Results from this research project will be used to provide input to North American design guides for atrium smoke management systems. A CFD modeling effort performed in parallel with the experimental program will allow extension of the experimental results to other geometries such as higher elevation atria and fires occurring under balconies.

3. EXPERIMENTAL PROGRAM

In recent years, NRC has conducted major research projects related to atrium smoke management [9, 11, 12]. These studies were conducted in a full-scale atrium model with overall dimensions of 12.2 m by 18.3 m by 12.5 m. This facility includes a smoke exhaust system located at the atrium ceiling with a variable capacity up to 25 m^3/s.

A literature review was conducted to assure that all current information on balcony spill plumes was available. This information was used to design a suitable experimental facility. Existing correlations were reviewed to ensure that the primary parameters that effect balcony spill plume air entrainment were included in the experimental program.

The basic concept for the experimental program was to produce steady fires in a fire compartment and measure the BSP mass flow rate after a steady smoke layer forms in the atrium. Certain elements of the compartment geometry i.e. opening width, fascia depth and balcony depth, were required to vary over a range typically found in buildings. The nature of buoyant plume flow required that these experiments be conducted indoors to minimize wind effects.

3.1 Experimental Facility

The existing facility was modified for this project by installing a 5 m by 5 m by 12 m fire compartment at one end of the atrium. Figure 2 below shows the experimental facility.

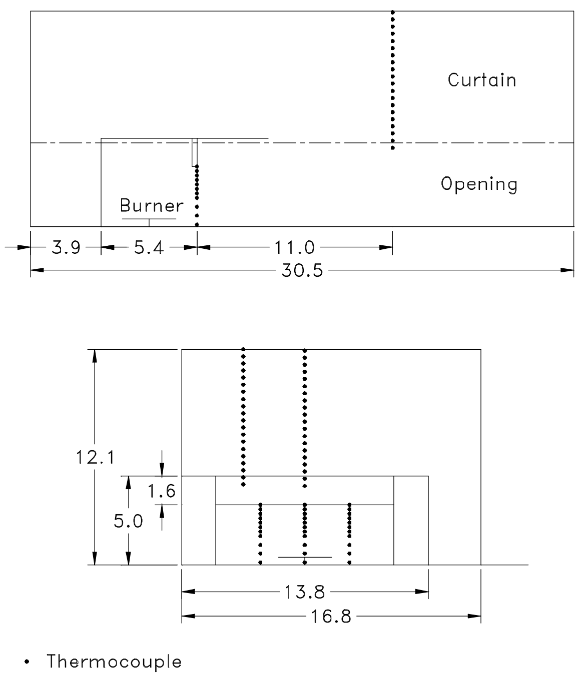

Figure 2. Experimental facility.

The front wall of the test compartment was modular in construction with opening widths ranging from 5m to 12 m. A removable 1.6 m fascia was installed at the top of the opening. A removable 4 m deep balcony was also installed. A single balcony height typical of those found in buildings was used. The selected balcony height provides a conservative result for the effect of the balcony height. The dimensions of the test compartment, balcony and fascia are consistent with the original one-tenth scale experiments. The volume of the fire compartment was kept constant since it was not a parameter considered in the existing correlations.

The fire compartment was lined with non-combustible mineral fibre insulation to protect against repeated fire exposure. This material has low thermal conductivity, thus minimizing conductive heat losses through the compartment boundaries and maximizing the temperature and resulting flow velocity of the hot layer exiting the compartment. The difference between the balcony and atrium ceiling elevations was approximately 7 m.

The fire source was located at the center of the fire compartment. A propane burner system was used to achieve steady, known heat release rates over extended periods of time. The propane flow rate was measured using rotameters to estimate the total heat release rate. Limited experiments with clothing, paper or other fuels typically found in commercial buildings will be conducted to compare against the propane test results.

A design fire size of 500 to 1000 kW has been suggested for the majority of fire situations in sprinklered office buildings [9, 10]. Similarly, a design fire size of 2500 kW covers most fire situations in retail malls [13]. Atrium smoke management system design guides ([5, 6]) suggest a design fire of 5000 kW for sprinklered retail applications. Based on this input, the experimental program included fire sizes in the range from 500 to 5000 kW.

The mechanical exhaust system consisted of 16 vents at the atrium ceiling elevation with a combined area of 2.63 m^2. This exhaust system was instrumented to measure the exhaust air mass flow rate. Oxygen, carbon dioxide and carbon monoxide concentrations in the exhaust duct were also measured to verify the fire's heat release rate using the oxygen depletion method [14]. Smoke obscuration was measured in the exhaust duct using a calibrated light attenuation smokemeter.

Since propane fires produce no smoke, accurate temperature measurements were required to determine the depth of the atrium smoke layer. Thermocouple trees were installed in the test compartment, at the compartment opening and in the atrium. The atrium thermocouple trees were used to determine the location of the smoke layer interface. Limited testing with optical smoke meters was performed to compare the atrium temperature layer elevation with the visible smoke elevation.

Approximately 100 tests were planned with various combinations of compartment geometry and fire size. Each test was conducted at five different exhaust flow rates in the range of 5 to 25 m^3/s varied every 500 s. Preliminary tests show that approximately seven to eight minutes is required for the atrium smoke layer to reach steady-state conditions.

3.2 Experimental Objectives

The height of the atrium smoke layer above the balcony is the major dependant parameter measured during the experiments. This position represents the elevation where a mass balance is achieved between the BSP smoke production rate and the exhaust rate from the top of the smoke layer. The smoke layer elevation is calculated from the atrium thermocouple tree measurements.

The exhaust system flow rate was adjusted to achieve various smoke layer depths for each geometry and fire size combination. This flow rate will be used to calculate the steady-state BSP smoke production rate for each unique combination of geometry and fire size.

The smoke production rates measured in the full-scale tests will be compared with estimates from existing BSP smoke rate correlations. These comparisons will be used to determine if the existing approaches accurately predict BSP smoke production rates or if alternative approaches are required. If so, the test data will be analyzed to determine suitable replacement correlations.

4. CFD Modeling Efforts

This research project included a CFD modeling effort to simulate BSP smoke entrainment. The objective of the modeling was to supplement the experimental data for evaluating BSP smoke production correlations. A validated CFD model will allow atria with higher elevations and compartments with different geometries to be simulated. As a more general objective, the BSP modeling effort will provide information on the use of CFD modeling for the design of building smoke management systems.

Accurate modeling of free combustion processes is a complex and difficult task. The fluid flows involved are highly turbulent and derive most of their power from density deficits. Phase changes of solids to combustible gases must be accounted for and heat transfer processes, especially radiation, have a greater relative effect on the flow field solution than for most low-temperature flows. Most of the key processes are non-steady state. The need to resolve small temporal variations makes Large-Eddy Simulation (LES) a more suitable model than Reynolds Averaged Navier-Stokes (RANS). The selection of CFD software which accurately models these processes is key in achieving accurate results.

4.1 Fire Dynamics Simulator (FDS)

The software chosen to model the BSP experimental facility is Fire Dynamics Simulator (Version 3), available from the National Institute of Standards and Technology [15]. FDS is specifically designed to model free combustion fluid flow and includes mechanisms for modeling combustion stoichiometry, smoke production and transport, and fuel pyrolysis. FDS is gaining popularity in both the research and engineering design communities due to its simplicity and open source nature.

FDS utilizes an explicit predictor-corrector scheme that has second order temporal and spatial accuracy. Spatial derivatives are approximated by second order central differences. Turbulence is modeled using a Smagorinsky form LES model. Direct Numerical Simulation (DNS) calculations are also possible if a fine numerical grid is specified. Low Mach number combustion equations [16] tailored for low-speed, buoyancy-driven flows are used to reduce the computational requirements.

FDS approximates the governing equations on a rectilinear grid. Non-rectilinear objects can be modeled as equivalent combinations of rectilinear objects since FDS allows vorticity generation at the corners to be ignored. Multi-blocking allows the use of more than one rectangular mesh in a simulation for grid refinement. Grid transformations are limited to two directions in any single mesh. Two-dimensional meshes and radial co-ordinate systems may also be specified.

Standard boundary conditions are available including open vents, symmetry planes and thin and thick wall conduction. Solid surfaces are assigned thermal boundary conditions and information about the burning behavior of the material i.e. ignition temperature, heat release rate, density. Tabulated material property data are typically used to define these parameters. For LES simulations, heat and mass transfer to and from solid surfaces are calculated using empirical correlations.

The combustion model in FDS defines mixture fraction as the fraction of gaseous fuel at a given point in the flow field. This conserved scalar quantity determines where combustion may occur in the flow domain. Mass fractions of the major reactants and products can be calculated from the mixture fraction via state relations which are in turn derived from a combination of simplified analysis and experimental data. Mixing-controlled combustion and infinitely fast reaction of fuel and oxygen are both assumed. The mixture fraction model treats fires as sources of combustion reactant(s) from which heat release is calculated. This is in contrast with other CFD software that models fires as heat sources only. The mixture fraction model also accounts for conditions that prevent combustion including oxygen depletion or temperature deficits.

Radiative heat transfer is modeled by solving the radiation transport equation for a non-scattering gray gas. A wide band radiation model is also available. The radiation equation is solved using a finite volume method similar to that used for convective transport. Approximately 100 discrete angles are used, requiring about 15 % of the total computational time for a simulation. FDS uses a constant radiative fraction for fuels. This is one of the major approximations in the radiation model since actual radiative fractions vary with flame temperature and combustion product concentrations.

The Smokeview program distributed with FDS displays model results graphically and allows some basic parameter editing of FDS input files. Standard visualizations are available including the commonly used Plot3D data format [17].

4.2 Fire-Specific Modeling Issues

Because of the variety of processes involved, fire dynamics modeling possesses an inherent trade-off between small- and large-scale resolution. Current workstations can perform calculations in a reasonable amount of time with the number of cells in the 10^6 range. Converting to a linear scale, this yields a dynamic range (i.e. ratio of largest to smallest eddy length scales) on the order of approximately 10^2. For combustion processes occurring at length scales less than 1 mm and building geometries on the order of 10^2 m, the dynamic range required for accurate modeling of all relevant fire processes is approximately 10^5. This limitation should be kept in mind when designing or interpreting fire dynamics simulations

The accuracy of LES fire dynamics simulations depends primarily on the grid resolution, especially around the fire. If the fire source is inadequately resolved, the mixture fraction combustion model suffers inaccuracies in predicted heat release rates due to numerical diffusion. A measure of grid resolution of the fire source can be derived from the characteristic fire diameter, D^*:

$$D^* = \left(\frac{Q}{\rho_\infty c_p T_\infty \sqrt{g}} \right)^{2/5} \quad (2)$$

where:

Q	= fire heat release rate (kW);
ρ_∞	= ambient air density (kg/m^3);
c_p	= heat capacity of air (kJ/K-kg);
T_∞	= ambient air temperature (K);
g	= acceleration due to gravity (m/s^2)

Assuming standard atmospheric conditions, Equation 2 simplifies to the following expression:

$$D^* = 0.0606\, Q^{2/5} \quad (3)$$

The non-dimensional quantity $D^*/\delta x$ where δx is the nominal grid cell size in metres can be used as a independent measure of fire resolution. Related research at NRC into this issue has shown that adequate modeling of building fires can be achieved with a grid resolution which yields $D^*/\delta x$ greater than 14 [18].

4.3 Fire Compartment Model

A limited number of fire compartment simulations were performed to aid in preliminary grid design. The FDS compartment model is shown in Figure 3. Accurate prediction of compartment temperatures and airflows was a necessary precursor to modeling of the BSP and atrium hot layer. All surfaces were specified as inert and the compartment opening was specified as an open boundary condition. The fire was modeled as a steady release of propane throughout the simulation. A variety of grid designs were evaluated. Simulation times of 120 s were chosen to achieve steady-state conditions.

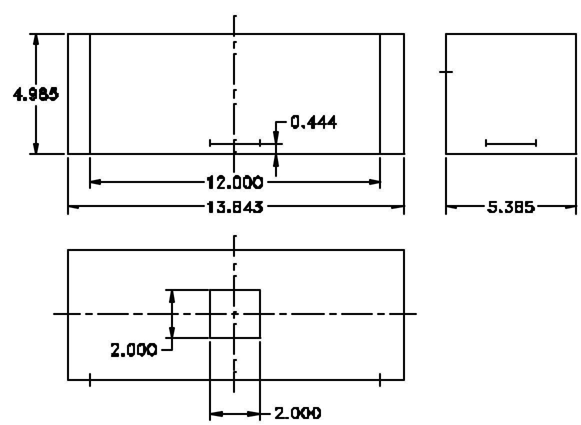

Figure 3. Geometry of fire compartment model.

4.4 Experimental Facility Model

The geometry of the full experimental facility model is shown in Figure 4. All solid surfaces were modeled as inert since no combustible surfaces were present in the experimental facility and conductive losses were assumed to be minimal. Two open vents were specified in the north wall of the facility to model the curtain opening. The exhaust system was modeled as a 2.0 m by 14.8 m opening centered on the fire compartment opening with a specified volumetric flow rate of 20 m^3/s. Three isometric computational meshes were defined: a mesh centered on the fire, a mesh covering the fire compartment, and a mesh covering the atrium. Simulation times of 120 s were chosen to achieve steady-state conditions.

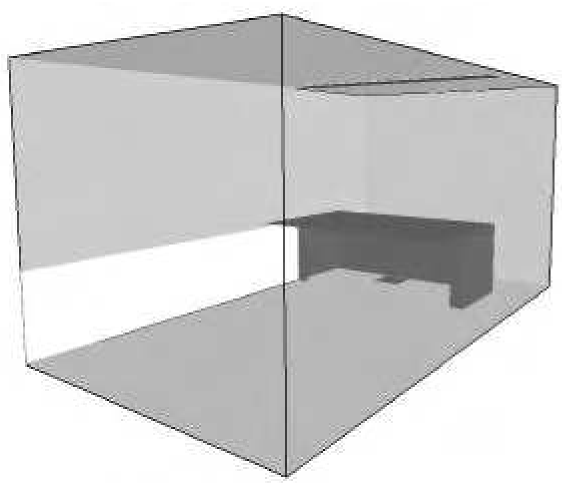

Figure 4. Geometry of experimental facility model.

4.5 Zone Model Comparison

Part of the future work planned for this research effort is a comparison of both the experimental data and CFD model results with zone fire models. Traditional zone models treat each fire compartment, including the atrium, as a two-zone volume. Simplified energy and mass balance correlations between the fire, hot layer and cold layer are solved. Temperatures and gas concentrations for both layers and the layer interface elevation are calculated as time-varying quantities. Limited agreement between the experimental data and zone models is expected since the ability of zone models to accurately model BSPs is unknown.

5. PRELIMINARY RESULTS

The simulations presented in this paper were performed on a Pentium IV workstation with a 2.5 GHz processor and 1.0 GB of RAM running Windows 2000.

5.1 Fire Compartment Model Results

Figure 5 shows compartment temperature profiles for models C1 through C3, C6 and test BSP-003. The various compartment models use different grid designs on the order of 0.1 m. For the 2.0 MW fire size modeled here, the corresponding value of D^*/?x is 12.7.

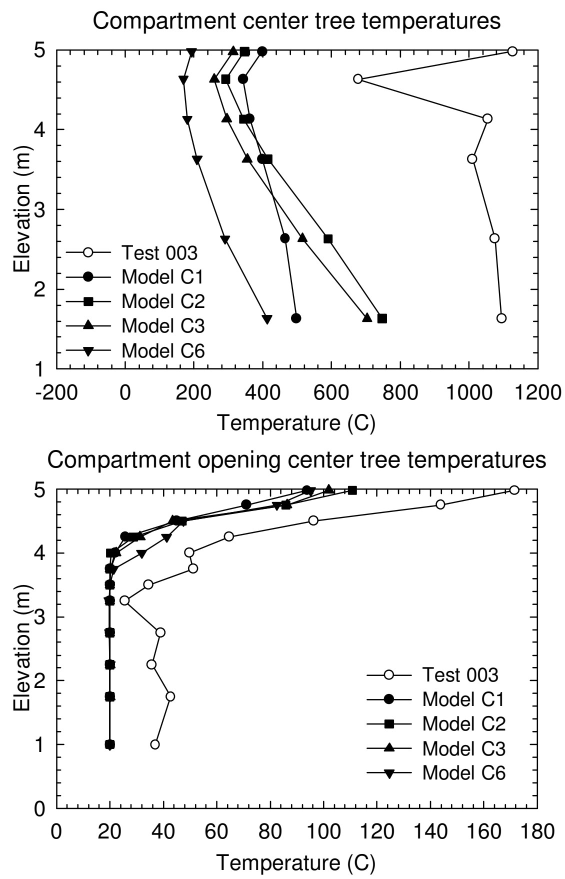

Figure 5. Fire compartment temperature profiles.

The FDS model underpredicts the flame and plume temperatures above the burner, resulting in low hot layer temperatures compared to the experimental data. The main parameter affecting the accuracy of the flame temperature prediction is grid size. A grid sensitivity study is being conducted to determine an optimal compartment grid design.

5.2 Experimental facility model results

Figure 6 shows the BSP temperature distribution for model 39.

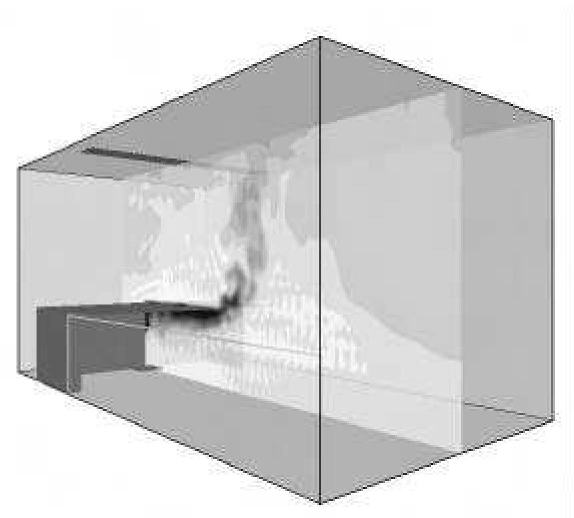

Figure 6. Atrium temperature visualization.

Results from model 39 show that the BSP projects well into the atrium, which agrees qualitatively with visualization obtained from smoke bomb tests in the facility. Atrium smoke layer temperatures are underpredicted compared to experimental data. This is attributed to less than optimal grid design in the fire compartment.

6. FUTURE WORK AND CONCLUSIONS

The preliminary model results presented here demonstrate fair agreement with experimental data. Further grid refinement will be performed to reduce the differences between the CFD model results and the experimental data. Future analysis will include comparison of the CFD model results with both zone model results and engineering correlations for smoke layer height derived from scale experimental work. Experimental and modeling work continues on this project.

ACKNOWLEDGEMENTS

The authors wish to acknowledge the staff at NRC's National Fire Laboratory for their efforts in the design and construction of the experimental facility.

REFERENCES

[1]. H. P. Morgan, N. R. Marshall, Smoke control measures in a covered two-story shopping mall having balconies as pedestrian walkways, BRE CP11/79, Building Research Establishment, Borehamwood, UK, 1979.

[2]. H. P. Morgan, N. R. Marshall, Smoke hazards in covered, multi-level shopping malls; an experimentally based theory for smoke production, BRE CP48/75, Building Research Establishment, Borehamwood, UK, 1975.

[3]. M. Law, A note on smoke plumes from fires in multi-level shopping Malls, Fire Safety Journal, Volume 10, p. 197-202, 1986.

[4]. P. H. Thomas, On the upward movement of smoke and related shopping mall problems, Fire Safety Journal, Volume 12, p. 191-203, 1987.

[5]. CIBSE, Relationships for smoke control calculations, Technical Memoranda TM19:1995, Chartered Institute of Building Services Engineers, London, 1995.

[6]. NFPA, Guide for smoke management systems in malls, atria, and large areas, NFPA 92B, National Fire Protection Association, Quincy, MA, 2000.

[7]. M. Poreh, H. P. Morgan, N. R. Marshall, R. Harrison, Entrainment by two-dimensional spill plumes, Fire Safety Journal, Volume 30, No. 1, p. 1-19, 1998.

[8]. P. H. Thomas, H. P. Morgan, N. Marshall, The spill plume in smoke control design, Fire Safety Journal, Volume 30, p. 21-46, 1998.

[9]. G. D. Lougheed, Expected size of shielded fires in sprinklered office buildings, ASHRAE Transactions, 103, (1), p. 395-410, 1997.

[10].D. Madrzykowski, R. Vettori, A sprinkler fire suppression Algorithm. J. of Fire Prot. Engr. 4: 151-164, 1992.

[11].G. D. Lougheed, G. V. Hadjisophocleous, C. McCartney, B. C. Taber, Large-scale physical model studies for an atrium smoke exhaust system, ASHRAE Transactions, 105, (1), p. 676-698, 1999.

[12].G. D. Lougheed, C. McCartney, B. C. Taber, Smoke movement for sprinklered fires, ASHRAE Transactions, 106, (Pt. 1), p. 605-619, 2000.

[13].G. D. Lougheed, C. McCartney, B. C. Taber, Sprinklered mercantile fires, ASHRAE Transactions, 107, (1), p. 730-743, 2001.

[14].ASTM, E 2067-03 Standard practice for full-scale oxygen consumption calorimetry Fire Tests, American Society of Testing and Materials, West Conshohocken, PA, 2003.

[15].K. B. McGratten, et. al., Fire Dynamics Simulator (Version 3) – User's Guide, National Institute of Standards and Technology, 2002.

[16].R. G. Rehm, H. R. Baum, The equations of motion for thermally driven, buoyant Flows, Journal of Research of the NBS, 83:297–308, 1978.

[17].P. P. Walatka, P. G. Buning., PLOT3D User's Manual (Version 3.5) - NASA Technical Memorandum 101067, NASA, 1989.

[18].A. Bounagui, N. Benichou, C. McCartney, A. Kashef, Optimizing the grid size used in CFD simulations to evaluate fire safety in houses, NRC 2nd Symposium on Computational Fluid Dynamics, Ottawa, Ontario, 2003.