Proceedings of the 2006 Winter Simulation Conference
L. F. Perrone, F. P. Wieland, J. Liu, B. G. Lawson, D. M. Nicol, and R. M. Fujimoto, eds.

USING A CFD SIMULATION IN DESIGNING A SMOKE MANAGEMENT SYSTEM IN A BUILDING

George Hadjisophocleous
Yoon J. Ko

Department of Civil and Environmental Engineering
Carleton University
1125 Colonel By Drive
Ottawa, Ontario, CANADA

ABSTRACT

This paper presents a study on the effectiveness of a smoke exhaust system in a complex building using the Computational Fluid Dynamics (CFD) models. The CFD model FDS (Fire Dynamics Simulator) was used for this study. To simulate fires in the building a design fire was selected to represent the fire loads expected in the lobby area. Smoke movement from the origin of fire in the lobby and smoke contamination in the interconnected corridors are simulated in order to design the exhaust system to be capable of maintaining tenable conditions in the corridor used for evacuation. The results of these simulations are presented and discussed.

1 INTRODUCTION

In building fires, most fatalities and injuries are often caused by inhalation of smoke. Smoke contains toxic and irritant gases and is the main cause of fatalities in fires (Klote 2002). In addition, smoke reduces visibility, which makes it difficult for occupants to escape the building (Lougheed and Hadjisophocleous 2001).

Smoke rises up from the origin of fire due to its buoyancy. As it rises up, it entrains more air and fills the upper region of the space forming an upper hot layer. Once the smoke layer is formed, its depth increases with time as more gases from the fire plume enter the hot layer. A critical design parameter of the smoke management system is maintenance of the required minimum height of the smoke layer, which allows safe conditions for occupants. The key to the effective performance of this strategy is estimating the amount of smoke that should be extracted from the hot layer so that its thickness does not increase with time.

The building considered for this study includes a lobby area which is open to ground and second floor corridors. One of the fire scenarios of concern is that of a fire originating in the lobby and smoke moving into the interconnected common areas such as the corridors. Since the common areas and corridors are the main routes used for evacuation, smoke in this area is a serious threat to the occupants of the entire building.

Designing fire protection systems for buildings with complex geometries cannot be done using simple correlations and simple models. For such buildings, the use of CFD models is a necessary tool that can predict the movement of smoke and the impact of smoke exhaust systems.

This study uses CFD computer modelling to design a smoke exhaust system for a new building. The goal of this study is to design a smoke exhaust system that will assist in maintaining tenable condition in the corridors of the building in the event that a fire starts in the lobby area.

1.1 Background of FDS

The Fire Dynamics Simulator (FDS) (McGrattan and Forney 2004), is used in this study. FDS is a computational fluid dynamics (CFD) model of fire driven fluid flow developed by NIST (National Institute of Standards and Technology), 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 (McGrattan and Forney 2004). FDS is used to simulate the fire and the conditions in the building due to the fire.

The formulation of the equations and the numerical algorithm are in the Fire Dynamics Simulator − Technical Reference Guide (McGrattan 2004). Its main features:

1. Hydrodynamic Model

- The fundamental conservation equations of mass, momentum, and energy.
- Large Eddy Simulation (LES) approach for turbulence (Baum et al 1998).
- Direct Numerical Simulation (DNS) if the numerical grid is fine enough.
- Limited to low Mach number flows.

1-4244-0501-7/06/$20.00 ©2006 IEEE

2. Combustion Model

- Mixture fraction based, equilibrium chemistry model for combustion.

3. Thermal Radiation Transport Model

- The Radiative Transport Equation (RTE).
- Numerical methods.
- Finite difference scheme (1st-2nd order); predictor corrector time integrator (2nd order); rectangular Cartesian grid; multi-block grid.

2 BUILDING DESIGN

2.1 Building Description

The building considered for this study is a community centre, which houses a pre-school area, full-size gymnasium, dance room, fitness room and multi-function rooms, which are laid along a cross-shaped corridor, as shown in Figure 1.

A relatively long (50 m) and high (9.8m) corridor extends from East to West connects the units of the building on the ground floor and is interconnected to the main entrance lobby. The East-West corridor is intersected with a North-South corridor, at each end of which is a set of open stairs serving as access to the second floor. The second level North-South corridor is 60 m long and 5 m high. At the junction, where the second level corridor bridges over the East-West corridor, the corridor opens onto the lobby and foyer, the ceiling height of the lobby being 5.8 m.

2.2 Fire Scenario and the Problem of Smoke Management

Various rooms located along the corridors have smoke doors that would not allow smoke to enter or escape from them. If a fire originates in the lobby, which is not smoke-separated from corridors, smoke will impinge the ceiling of the lobby and the buoyant hot gas will spill up into the interconnected East-West corridor as well as directly into the second level North-South corridor. In this scenario, the occupants of the entire building could be in danger since the corridor is the main route of evacuation.

To maintain safe egress routes, venting of the buoyant layer of hot gas is required in order to keep the height of the smoke layer above the highest level at which occupants are expected to be present for the time required for all occupants to evacuate the building.

The objectives of the computer modelling are the following:

- To determine the size and location of smoke exhaust vents required to maintain tenable conditions in the corridors and bridge.
- To show conditions in the interconnected corridors as a result of a fire in the lobby.

3 MODEL DESCRIPTION

3.1 Description of the Geometry

Figure 1 shows the geometry of the building, and Figure 2 shows the geometry used in the model. As it can be seen from Figure 2, only the common areas, consisting of two corridors, the lobby and foyer areas, are considered in the model. The other building areas are not considered as they are smoke-separated and not expected to have any impact on the results.

The geometry of the two corridors and other spaces are simplified to conform to the rectangular grid. This improves model efficiency without any significant impact on the results.

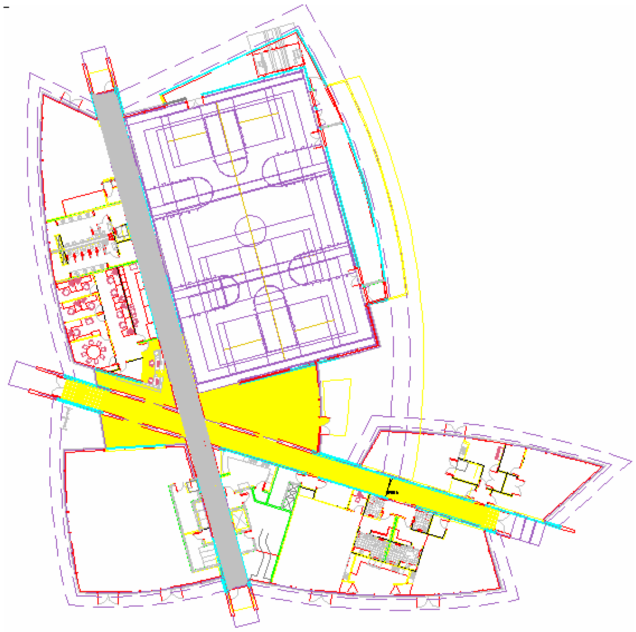

Figure 1. Building geometry

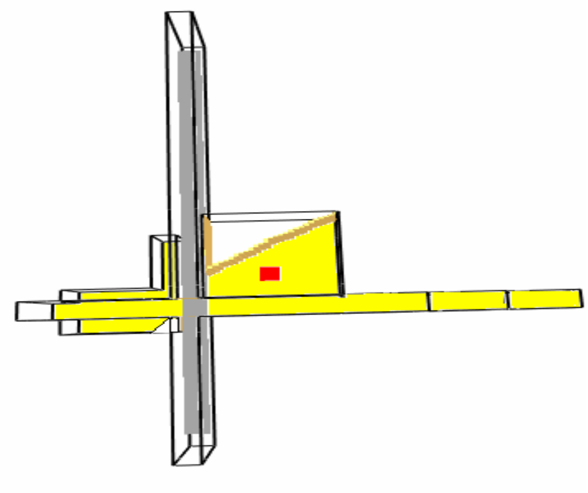

Figure 2. Floor plan used in the model

Figure 3 shows a three dimensional view of the building geometry as used in the model.

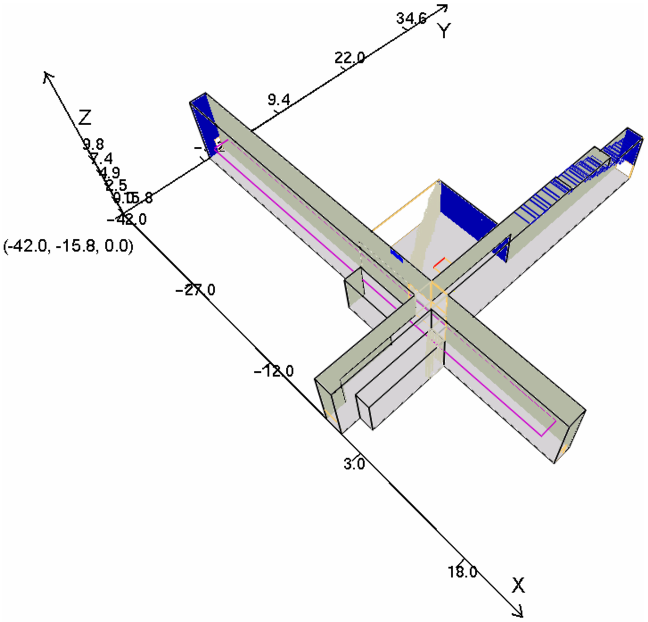

Figure 3. Three-dimensional view of the building geometry as used in the model

3.2 Grid Size and Boundary Condition

The grid size used for the FDS simulations is shown in Figure 4. A grid spacing of 0.25 m was used in the lobby where the fire originates, and a grid spacing of 0.5 m was used elsewhere. This grid was found adequate in resolving the plume dynamics in the fire room and smoke movement through the building.

The boundary conditions used in the model are as shown in Figure 5. The walls of the building were gypsum board. The skylight and the roof of the corridors, were set to be glass. The figures also show the location of the doors.

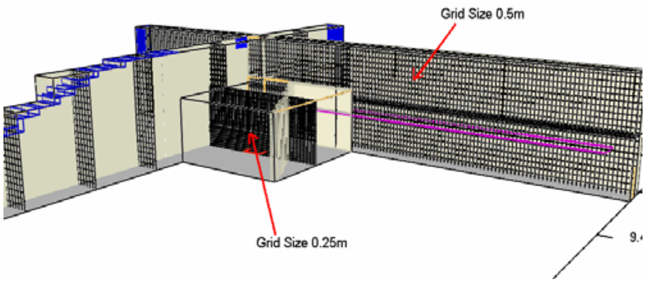

Figure 4. Grid size used in the model.

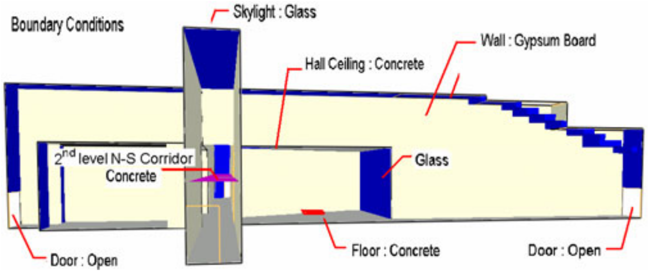

Figure 5. Boundary conditions used in the model.

3.3 Fire size

The fire location is considered to be the lobby area as it is the only area that is expected to have fire loads. For the simulations, the fire is assumed to be located at the geometric centre of the lobby area.

To determine the expected fire size to be used in the model, the fuel load in the lobby was considered. The fuel load was mainly due to sofas and during the holidays a Christmas tree. Experimental data of sofa and Christmas tree fires were used to estimate the fire size and duration. Both items, sofas and Christmas trees, burn relatively fast and release a maximum heat of 3.5 MW and 5 MW, respectively. Based on these data, it was decided to use a fire that will grow immediate to a size of 3.5 MW and stay at that level for the entire duration of the fire. In order to simulate a severe fire, the sprinkler system in the building was assumed not to activate.

4 RESULTS

A number of runs have been conducted using different fan sizes and locations. For each run, temperatures, velocities, CO concentrations and visibility were monitored to determine whether the exhaust system can maintain tenable conditions in the corridors. Special attention was placed on the second level of the North-South corridor where occupants moving towards the exit will have to pass through an elevated bridge.

4.1 Smoke Movement

The output from the FDS simulations is visualized using the Smokeview 3.1 visualization tool, developed by NIST. Figure 6 shows smoke from a fire in the lobby spilling into the interconnected East-West corridor as well as directly into the second level North-South corridor.

The preliminary runs showed that the buoyant smoke layer quickly drops down to unacceptable levels in the second floor North-South corridor. From the preliminary runs, it was found that tenable conditions can be achieved only when the door in the lobby is kept closed and the doors at the end of the each corridor are maintained open.

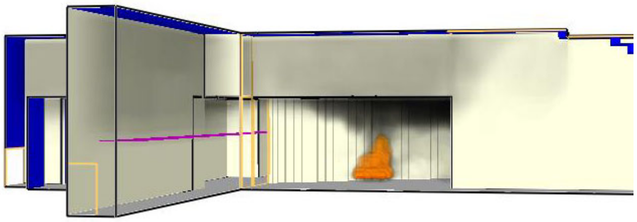

Figure 6 Simulation results of smoke movement

4.2 Fan Size and location

From a number of preliminary runs, locations and capacities of exhaust fans are determined. The proposed smoke exhaust system has three exhaust vents: one on the roof of the room of fire (the lobby), one on the north side of the North-South corridor and one on the east side of the East-West corridor. Figure 7 shows the location of the exhaust vents and door openings. The location of the exhaust vents shown in the Figure was found to be the most effective in minimizing smoke flow into the corridors.

The exhaust on the roof of the lobby has a capacity of 15 m^3/s, while the other two vents have a capacity of 10 m^3/s. Smoke exhaust vents in two corridors had to be wall-mounted because the corridor has a skylight rooftop.

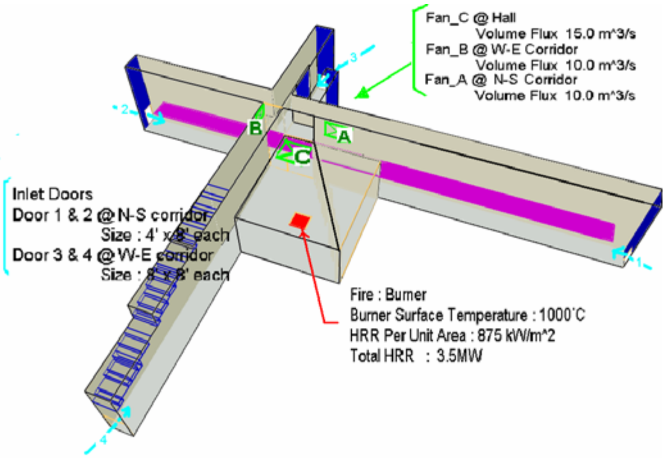

Figure 7. Location of exhaust fans and door openings

4.3 Temperature, CO concentration and Visibility

Results of temperature, CO concentration and visibility indicate that the proposed smoke exhaust system is capable of maintaining tenable conditions in the corridors so that the occupants will be able to evacuate safely.

Figure 8 shows temperature distributions on two vertical planes along the two corridors. This figure clearly demonstrates that the temperatures in the corridors are at low level except at the location where hot gases leave the lobby area. The system keeps the smoke interface level at 5.5 m in the East-West corridor, which is high enough to provide safe egress route for the occupants on the ground level.

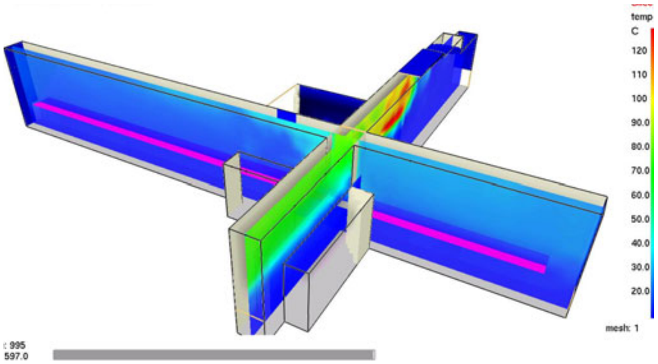

Figure 8. Temperature distribution along vertical planes in the corridors.

Figure 9 shows locations where the profiles of parameters are plotted in the North-South corridor, which is the area of concern. The results shown in Figures 10, 11 and 12 are at steady state, which was achieved at 600 s.

Figure 10 depicts temperature profiles at the monitored locations. The Figure shows that temperature at the centre of the upper region of the North-South corridor is high as the corridor is open to the lobby. Even at that location, however, the temperature at a height of 5.75 m or 2 m

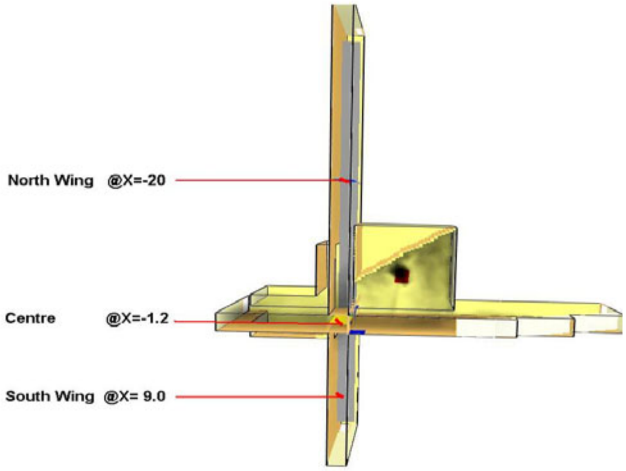

Figure 9. Data sampling points.

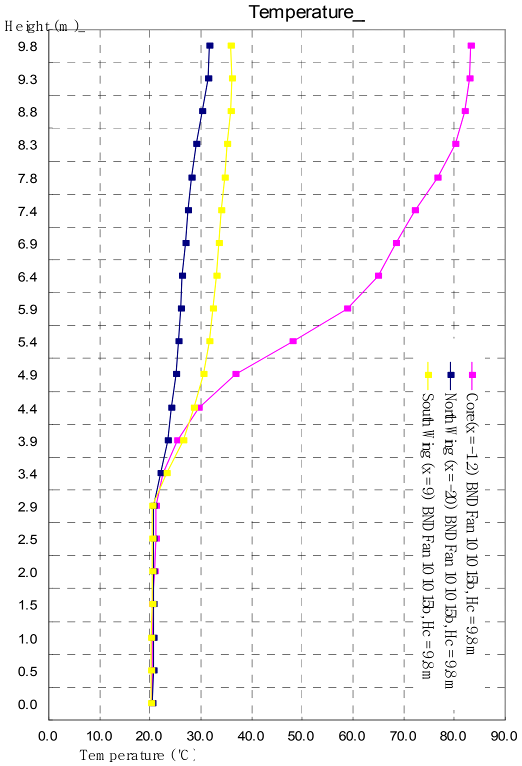

Figure 10. Temperature profiles at the points shown in Figure 9.

above the bridge is at around 57 °C. Temperatures at the North and South Wings are lower than 40 °C.

Figure 11 and Figure 12 depict visibility levels and Carbon monoxide (CO) concentration at the monitored locations in the North-South corridor, confirming that conditions in the corridors are tenable.

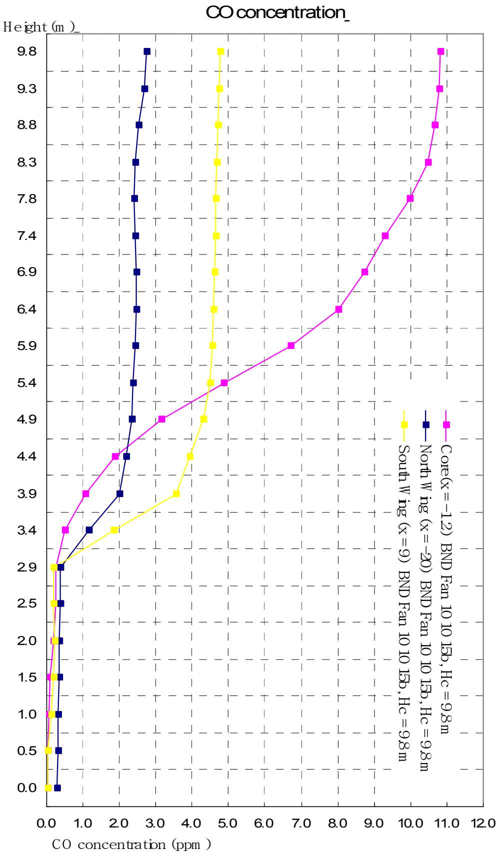

Figure 11. CO concentrations at points in Figure 9

CO concentrations remain at very low levels as shown in Figure 11. Considering the fact that CO concentration considered lethal for 50% of the population for a 30 min exposure is 3000 ppm, the concentration in the corridor is very low posing no threat to occupants.

Visibility in the corridor remains also at acceptable levels as shown in Figure 12. The lowest visibility value in the corridor is at 19 m, which is much higher than the 10 m usually considered unacceptable for buildings with large spaces. For smaller buildings a visibility of 5 m is the acceptable criterion.

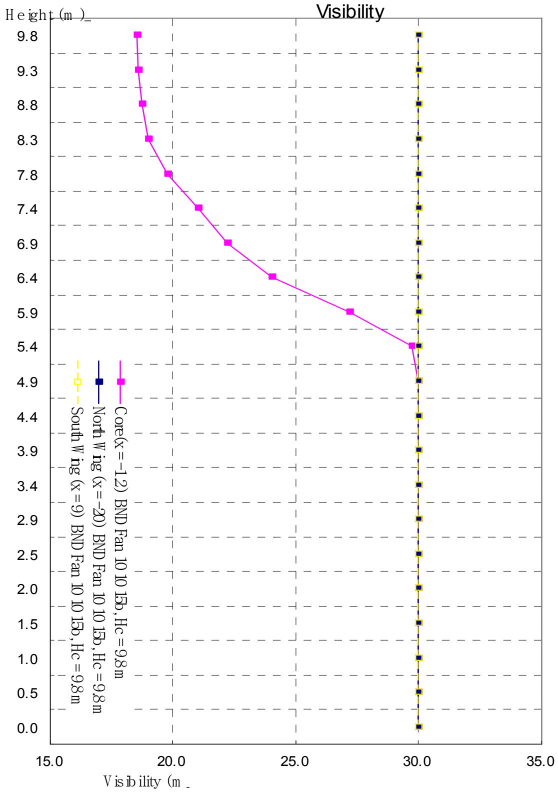

Figure 12. Visibility level at points in Figure 9.

5 CONCLUSION

This study demonstrated the use of computer modelling and simulation, as well as visualization tools for the design of smoke management systems in buildings. The performance of the smoke exhaust system was evaluated using temperature, CO concentrations and visibility through the entire model geometry, as well as the impact of door openings. The use of visualization tools was critical as it facilitated the analysis of the results and helped to locate areas of concern demanding further actions.

REFERENCES

Klote J H. 2002. Principles of smoke management, *American Society of Heating, Refrigerating and Air-Conditioning Engineers Inc.*

Lougheed G D, Hadjisophocleous G V. 2001. Smoke hazards from fires in high places *ASHRAE Journal*.

McGrattan K B, Forney G P. 2004. Fire Dynamics Simulator (Version 4) - User Guide. *NIST Special Publication 1019*, National Institute of Technology and Standards.

McGrattan K B. 2004. Fire Dynamics Simulator (Version 4) - Technical Reference Guide. *NIST Special Publication 1018*, National Institute of Technology and Standards.

AUTHOR BIOGRAPHIES

GEORGE HADJISOPHOCLEOUS holds the Industrial Research Chair in Fire Safety Engineering at Carleton University, where he moved in January 2001. He leads a team of 20 students working in a number of areas including Fire Risk Analysis, Smoke Movement, Design Fires and Fire Modelling. Prior to moving to Carleton University Prof. Hadjisophocleous was a Senior Research Officer at the Fire Risk Management Program of the National Research Council of Canada and Group Leader of the Fire Risk Assessment Group. He holds a Ph.D. degree in Mechanical Engineering from the University of New Brunswick and he is the author of over 150 publications in the areas of fire research, fire risk assessment, performance-based codes and CFD modelling. Dr. Hadjisophocleous is a Fellow of SFPE, member of NFPA, IAFSS, CIB W14 and a Registered Professional Engineer in the Province of Ontario.

YOON J. KO is a Ph.D student in Fire Protection Engineering Program at Carleton University in Canada. She received a M.S. degrees in Civil Engineering from Carleton University. Her current research interests include smoke management in atria, CFD study of balcony spill plume, and fire modeling. Her e-mail address is <yko@connect.carleton.ca>