2nd International Conference "From Scientific Computing to Computational Engineering"
2nd IC-SCCE
Athens, 5-8 July, 2006

SIMULATION OF THE SMOKE LAYER INTERFACE HEIGHT FOR A FIRE INDUCED IN AN ATRIUM

Cándido Gutiérrez-Montes, Enrique Sanmiguel-Rojas and Antonio Viedma

Department of Thermal Engineering and Fluid Mechanics
Polytechnical University of Cartagena
C/ Dr. Fleming, s/n (Campus Muralla) 30.202 Cartagena , Murcia, Spain
e-mail: candido.gutierrez@upct.es, web page: http://www.etsii.upct.es/index_esp.htm

Keywords: CFD, Fire, Building Atria, Experiments.
Abstract. *The calculation of the smoke layer interface height in case a fire breaks out in an atrium is determinant to make an appropriate fire safety system design. Most of the expressions used to calculate it assume no mass flow through the smoke layer interface.*

With this object a numerical model that simulate fire-induced thermal and fluid fields in an atrium using the finite volumes procedure code FLUENT has been developed. The model has been validated from new full-scale fire experimental tests performed at the 'Fire Atrium' of the Technological Metal Centre, Murcia, Spain. At the present work the experimental data from one fire experiment are shown. In this experiment an heptane pool fire placed at the centre of the floor was used, releasing an average heat of 1.8 MW.

A comparison between one empirical expression commonly used and an experimental temperature criterion, both for computing the smoke layer interface height, with the one predicted by the model is performed. The model agrees better with the second one and shows a mass of air across the smoke layer interface should be taken into account.

1 INTRODUCTION

Smoke management is one of the most important aspects of fire protection as smoke is, together with heat, often the main dangerous factor in an indoor fire.

When a fire takes place on the floor of an atrium the hot smoke will rise up to the ceiling. This smoke will be accumulated and will form a smoke layer depending on the smoke exhaust rate of the exhaust system. This system must assure good enough conditions for the hot gases smoke and temperatures not to pose a health hazard that put at risk the occupants' lives[1], keeping the smoke layer over a height, Z_{smoke}, larger than the highest level reached by the occupants of the atrium to give enough time to evacuate them. For a tall atrium, the evacuation time to fill up 80% of the space with smoke might be considered[2]. Depending on the building dimensions heights of 3 or 4 m are also accepted for the design of smoke management systems.

Nowadays, most of the atria smoke exhaust systems designs are based on code-type equations and correlation-based methods. However, these are very general equations not always accurate enough for this type of buildings leading to not optimized designs in capacity, localization and cost.

It is usually supposed that in enclosure fire conditions there are two clear different layers at the room, the smoke and the clear and cool air layer, and that no mass flow interchange between both layers happens. It is appropriated for small rooms. However, it has been recently demonstrated that, for fires in atria with a mechanical smoke exhaust system installed at the ceiling with floor air inlet to give an air inlet mass flow, this assumption is not very precise[3, 4]. The use of the equations and correlation-based methods above mentioned might be non recommendable. There is a mass flow across the layers' interface that in quasi-steady state depends on the heat release rate and on the number of air changes per hour. This mass flow has to be taken into account to compute the smoke layer height and so the exhaust rate of the smoke extraction system to be mounted.

Due to the increase on the computational power, and the numerous numerical and validation studies performed the CFD is becoming an important and reliable tool for fire safety systems building designs[5-7]. This is inducing a change from prescriptive-based to performance-based methodologies and codes[8, 9].

For the present work a CFD fire model has been developed using the finite volumes procedure code FLUENT. This model has been validated using experimental data obtained from full-scale burning tests carried out at the “Fire Atrium” new burning facility.

Finally, the fire field model predicted data have been compared with other expressions used to calculate the smoke layer height for a quasi-steady state atrium enclosure fire, demonstrating again the utility and accuracy of CFD for fire understanding and prevention and for smoke exhaust system design in atria.

2 PLANT AND MEASURING EQUIPMENT

The experiments have been carried out at the “Fire Atrium” of the Technological Metal Centre, in Murcia, Spain. It is an aluminium prismatic squared base which dimensions are 19.5 m x 19.5 m x 20 m, with pyramidal shaped roof, see figure 1. It has four exhaust fans set up at the roof and several vents arranged in a no symmetrical way in its walls. For the experiment of the present work a standard pool-fire (21-b) of 0.92 m diameter has been placed at the centre of its base.

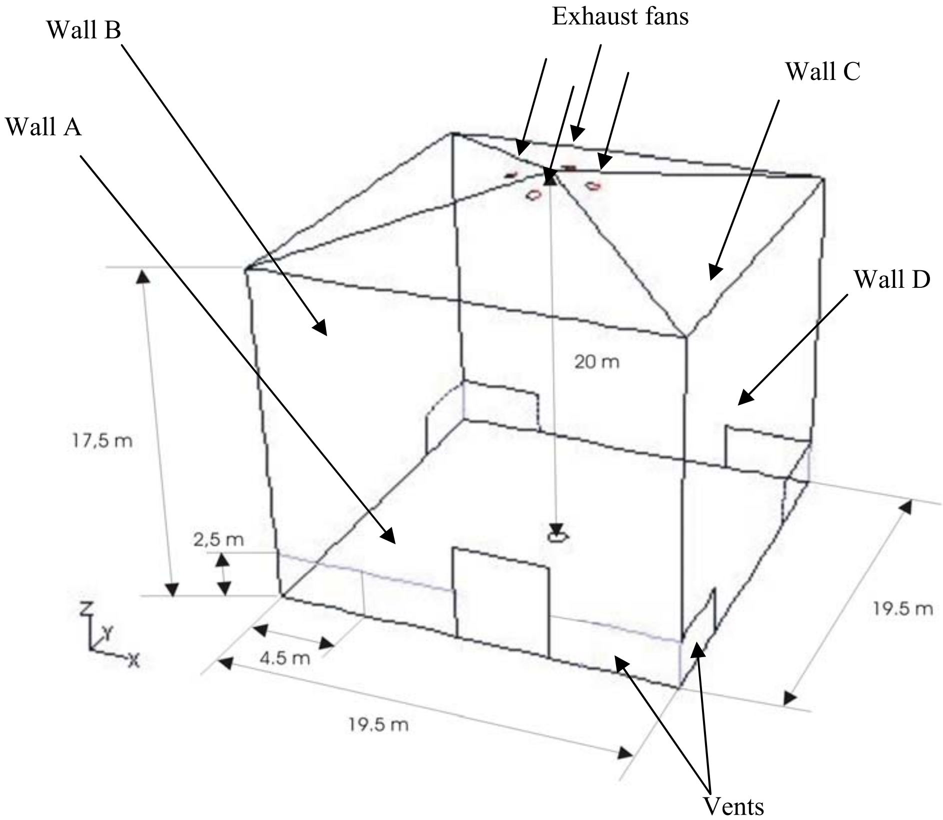

Figure 1. Plant scheme.

The variables measured were walls, ceiling and air temperatures and also air through the vents velocity and pressure at the exhaust fans. There are 61 sensors in all, see table 1.

Sensor	Type of sensor	Number of sensors
Temperature	Thermocouple K	26
Temperature	Pt 100	26
Pressure	Capacitive transducer	4
Velocity	Hot wire anemometer	5

Table 1. Type and number of sensors at the plant.

There are three vertical sections where the sensors are set up: wall A section, wall C section, and a central section parallel to the two sections before. There are also sensors set up at the roof and the exhaust fans.

Wall A and C have both four sensors placed at five and fifteen meters high to measure the temperature of the wall. These walls contain also eight and nine sensors respectively at the heights of five, ten and fifteen meters that register the temperature of the air close to the wall, 30 cm inside. There are also installed temperature and velocity sensors at the centre of each vent. Wall A has three vents and Wall C has two vents.

The central section contains twelve temperature sensors arranged at three different heights, five, nine and thirteen meters. Three of them are placed over the pool fire and the rest are mounted six meters from the centre on the side next to Wall B and three meters from the centre on both sides.

Finally, four temperature sensors measure the temperature of the roof. At the exhaust fans closest to the Wall A and the Wall C an absolute pressure sensor, a differential pressure sensor and a temperature sensor were also installed.

The weather conditions have been recorded using a meteorological station.

3 NUMERICAL MODEL

A numerical model has been built in order to simulate the thermal and fluid fields induced by an enclosure fire into an atrium. The three-dimensional geometry has been created using GAMBIT. Later this model has been implemented using the finite volumes procedure code FLUENT.

Due to the high symmetry characteristics of the topology of the experiment employed to validate this model, see next section, it has been studied only a quarter of the 'Fire Atrium' in order to require less computational cost. A quasi-structured non-uniform grid of 89000 cells, approximately, has been used, where cells have been refined near the pool-fire, see figure 2. Steady-state simulations have been performed with a segregated solver. The SIMPLE algorithm[10, 11] has been used to solve the coupling between continuity and momentum equations through pressure and the PRESTO! scheme has been used to discretize the momentum equation. QUICK scheme[12], which is supposed to be accurate on structured grids aligned with the flow direction, has been used for the rest of the discretizations.

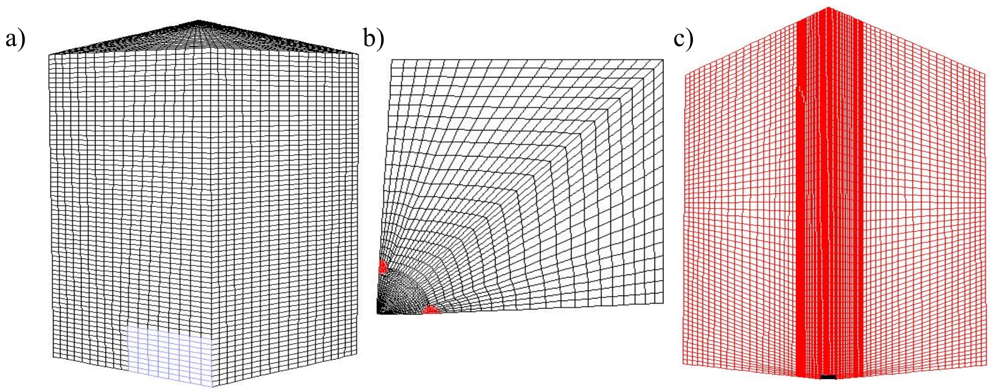

Figure 2. Domain mesh: Walls' mesh in a), roof's mesh in b) and symmetry planes' mesh in c).

Standard k-ε turbulence model[13] has been used. In combustion processes, such as the one of this work, thermal radiation plays an important role. To model radiation it has been used the P1 radiation model[14, 15], which is the simplest case of the more general P-N model.

For the walls, the roof and the ground it has been set a logarithmic wall-law for both temperature and pressure fields. Walls and roof are made of aluminium and they are 6 mm thick. It has been used a mixed heat transfer model to simulate heat exchange with the outer atmosphere, which takes into account convective and radiative outer heat transfer. A radiation heat transfer model has been used for the ground.

The vents have been modelled as outer air inlets which generate pressure losses equal to the 30 % of the dynamic pressure.

A constant pressure drop has been set to simulate the steady state at the exhaust fans which generates a mass flow outlet equal to the one measured at the experiment.

In order to simulate the heptane's combustion, the species transport combustion model has been used. The combustion is supposed to be mixing-limited. This means that turbulence slowly mixes fuel and oxidizer into the reaction zones where they burn quickly. And so, the Eddy-Dissipation-Model, based on the work of Magnussen and Hjertager[16], has been used. The pool-fire has been simulated as a cylindrical volume with its same dimensions (0.2 m high and a diameter of 0.92 m). A constant uniform injection rate of vaporized heptane is set at the top of the pool as

$$v \equiv \frac{4\dot{m}}{\rho_v \, \pi D^2}\,, \qquad (1)$$

where ρ_v is the vaporized heptane's density (4.25 kg/m^3), D is the pool's diameter and the mean burning rate, $\dot{m}$, is calculated as follows[17, 18]:

$$\dot{m} \equiv \frac{\Delta W}{\Delta t \; A_f}\,. \qquad (2)$$

4 MODEL VALIDATION

4.1 Test characteristics

The experimental case presented at this work was held in the month of July, 2005. The weather conditions were soft wind, atmospheric pressure of 1018 mbar and ambient temperature equal to 28.9 ºC.

As it has been said before, it was used a standard pool-fire (21-b) of diameter equal to 0.92 m placed at the centre of the atrium's base which contained 52 litres of heptane fuel, see figure 3 a). Only the vents of Wall A and C were opened in a symmetrical layout, see figure 3 b). The combustion lasted 883 s and the average heat release rate computed was 1.8 MW.

a)

b)

Figure 3. Pool-fire with D = 0.92 n and H = 0.2 m, in a), and test topology, in b).

4.2 Results and discussion

A comparison between the averaged experimental data obtained with the predicted results from numerical simulation is done for the validation of the model.

In table 2, data from the exhaust fans are compared. The flow through the exhaust fans predicted numerically is a little bit smaller than the one measured experimentally. This can cause the difference in the outlet flow's temperature, that is also due to the fact that it is being compared a steady-state simulation with quasi-steady experimental results. These discrepancies will be bigger at the upper part of the 'Fire Atrium', as the hot gases from the plume hit directly on the roof.

Exhaust fan	Predicted Data	Results registered

	Flow	Temperature	Flow	Temperature
Closest to Wall A	3.4	390.8	3.7	367.2
Closest to Wall D	3.4	391.1	Not measured experimentally	

Table 2. Air flow through the exhaust fans, in m^3/s, and hot air temperature, in K.

In the same way, the predicted temperature of the roof is higher than the experimental one, see figure 4. The differences are smaller than 13 K, so a good agreement between numerical and experimental data is achieved.

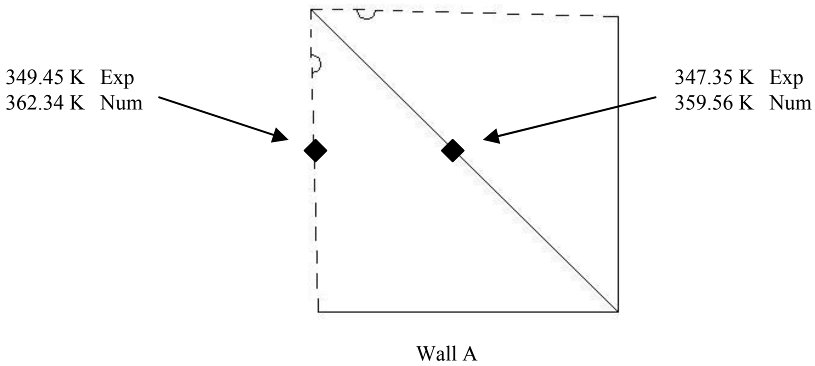

Figure 4. Comparison between experimental and numerical results for the roof temperature.

Wall temperatures obtained numerically show a maximum difference respect to measured experimental data smaller than 7 K, see figure 5 b), whereas the air close to Wall A temperatures show a maximum difference smaller than 10 K, see figure 5 a). Numerical temperatures are higher than experimental temperatures at the higher zones of Wall A due to the same reason commented above. However, at the lower zones the experimental results are higher than the predicted ones because of the effect of radiation over the sensors. Anyway, the model's predictions agree well with the experimental data.

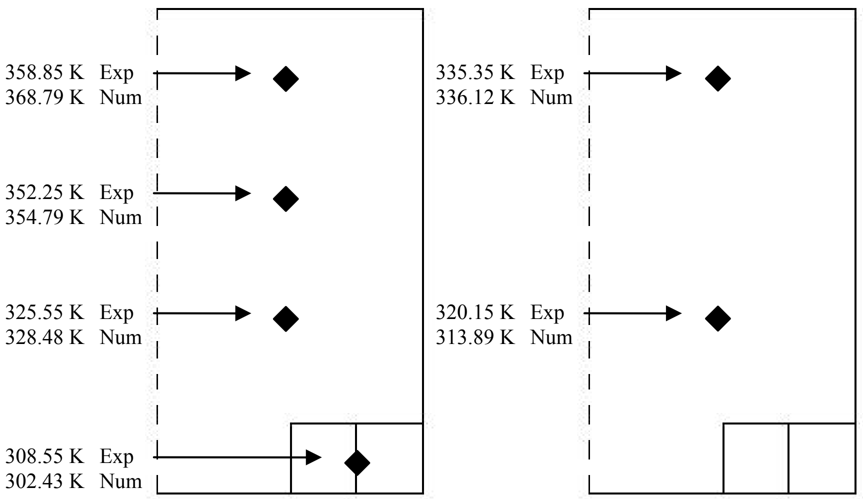

Figure 5. Comparison between experimental and numerical results for the air close to the Wall A temperature, in a), and for the Wall A temperature, in b).

Comparison between experimental and numerical temperatures at the central section shows relative big differences over the plume, see figure 6. These might be probably owed to the combination of three factors: first, the heat insulation of the temperature sensors that makes these measure air temperatures lower than the real ones, second, the combustion model is also not the most accurate available in Fluent, however it is quite proper to simulate pool-fires, and, finally, the P1 radiation model is known to over-predict temperatures near a heat source, although it is proper for combustion modelling. The other three temperature points show good agreement between experimental and numerical data.

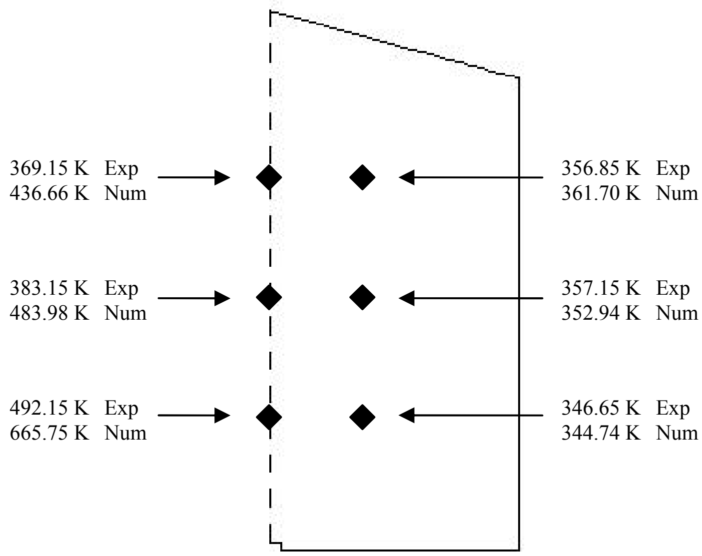

Figure 6. Comparison between experimental and numerical results for the air flow temperature at the central section, seen from Wall A.

5 SMOKE HEIGHT COMPARISON

As it has been mentioned before, in an atrium it is very important to have an appropriate smoke exhaust system that, in case of fire, is capable to maintain the smoke layer in a height that does not put at risk the lives of the occupants during a period of time enough to evacuate them.

For an atrium fire with quasi-steady venting conditions it can be found many expressions at the technical bibliography to compute the height of the smoke layer[1], Z_{smoke}. One of the most commonly used is the expression

$$Z_{smoke} \equiv \left(\frac{\dot{m}_{exhaust} - 0.0018\,\dot{Q}_{convective}}{0.071\,\dot{Q}_{convective}^{1/3}} \right)^{3/5}, \quad (3)$$

where $\dot{Q}_{convective}$ is the convective portion of heat release rate of the fire (kW), and $\dot{m}_{exhaust}$ is the exhaust mass flow rate (kg/s). In this expression no mass flow through the smoke layer interface is supposed.

Recently, Chow et al[4] have studied the mass flow rate across the layer interface in an atrium with mechanical exhaust system concluding that it should be taken into account, and that so the expression (3) should be modified. At the experiments, Chow et al[19] take the smoke layer interface height as the one where the expression

$$T \equiv 0.1\left(T_{\max} - T_0\right) + T_0, \quad (4)$$

is fulfilled, being T_0 the ambient temperature and $T_{\max}$ the maximum smoke temperature measured at the vertical column of sensors.

For the present work, these two expressions are compared with the predicted data obtained from our model for two different vertical lines located each one at one different central section of the ‘Fire Atrium’ respectively, see figure 7.

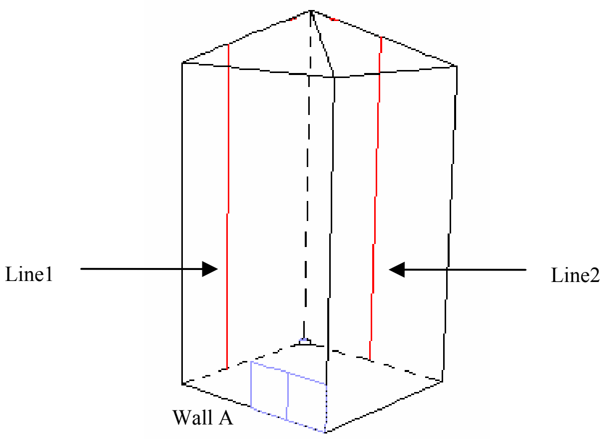

Figure 7. Location of the two lines defined at the central sections. View from the outer corner.

The model shows that there are large temperature and oxygen mole fraction gradients that locate the smoke layer interface at around 3 m high, see figure 8. It can not be fixed an only height for the smoke layer since depending on the atrium geometry and the air intake openings' layout there will be more smoke concentration in some parts of the atrium than in others for the same height, see figure 8 b). The expression (3) estimates a $Z_{smoke}= Z_1$ higher than the results predicted by the model, whereas these agree better with the temperature criterion used by Chow, which is more accurate.

The mass flow through the layers' interface should be taken into account as it does the numerical model. It can be critical when building design is being performed as it can involve unnecessary investments, wrong smoke exhaust system location and capacity, or even human losses.

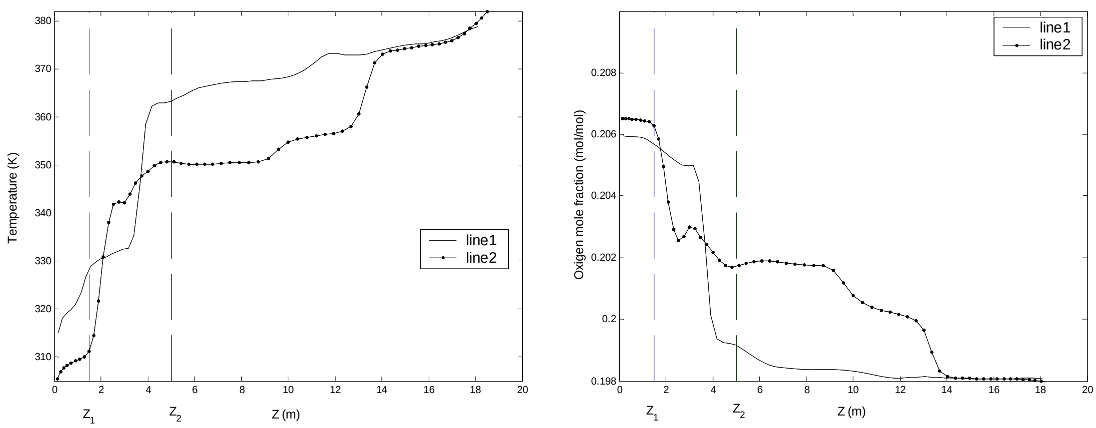

Figure 8. Temperature distribution for line1 and line2, in a). Oxygen mass fraction distribution for line1 and line2, in b). Z_1 is the smoke layer height from (3) and Z_2 from (4).

6 CONCLUSIONS

A numerical model has been developed for simulating atrium enclosure fires using the commercial code Fluent.

A validation study has been done. The model shows good agreement with the experimental data from a fire test carried out at the 'Fire Atrium'.

A comparison between different expressions for computing the smoke layer interface height for an atrium fire with quasi-steady venting conditions with the results predicted by the model is performed. The model agrees well with the expression proposed by Chow that takes into account the mass flow through the smoke layer interface. The model also agrees with the temperature criterion used by Chow to locate the smoke layer height.

This work represents a new proof of CFD computing power to predict fire conditions reliably and so its value for building fire safety design.

7 REFERENCES

[1] NFPA 92B. Guide of smoke management systems in atria, covered malls, and large areas. National Protection Association, Quincy, MA, USA, 1995.

[2] Chow WK, Chow CL. Evacuation with smoke control for atria in green and sustainable buildings. Building and Environment, 2005; 40: 195-200.

[3] Chow WK, Yi L, Shi CL, Li YZ, Huo R. Experimental studies on mechanical smoke exhaust system in an atrium. Journal of Fire Sciences, 2004.

[4] Chow WK, Yi L, Shi CL, Li YZ, Huo R. Mass flow rates across layer interface in a two-layer zone model in an atrium with mechanical exhaust system. Building and Environment, 2006, 41, pp: 1198-1202.

[5] Cui E, Chow WK. Simulation on indoor aerodynamic induced by an atrium fire. Building and Environment 2005; (40): 1194-1206.

[6] Qin TX, Guo YC, Chan CK, Lin WY. Numerical investigation of smoke exhaust mechanism in a gymnasium under fire scenarios. Building and Environment. In press.

[7] Jianguo C, Haixin C, Song F. Numerical Investigation of Fire Smoke Transport in the Tsinghua University Sports Center. Tsinghua Science and Technology. 2005; 10 (5): 618-622.

[8] Chow WK. A preliminary discussion on engineering performance-based codes in the Hong Kong special administrative region. International Journal on Engineering Performance-Based Fire Codes. 1999; 1 (1):1-10.

[9] SFPE engineering guide to performance-based fire protection analysis and design of buildings. Bethesda, Maryland, USA: Society of Fire Protection Engineers; 2001.

[10] Patankar SV, Spalding DB. A Calculation Procedure for Heat, Mass and Momentum Transfer in Three-dimensional Parabolic Flows. International Journal for Heat and Mass Transfer, 1972, 15, 1787-1806.

[11] Patankar SV. Numerical Heat Transfer and Fluid Flow, Hemisphere Publishing Corporation, Taylor & Francis Group, New York, 1980.

[12] Leonard BP, Mokhtari S. ULTRA-SHARP Nonoscillatory Convection Schemes for High-Speed Steady Multidimensional Flow. NASA TM 1-2568 (ICOMP-90-12), NASA Lewis Research Center, 1990.

[13] Launder BE, Spalding DB. Lectures in Mathematical Models of Turbulence. Academic Press, London, England, 1972.

[14] P. Cheng. Two-Dimensional Radiating Gas Flow by a Moment Method. AIAA Journal, 2:1662-1664, 1964.

[15] R. Siegel y J. R. Howel, Termal Radiation Heat Transfer, 1992, Hemisphere Publishing Corporation, Washington D.C.

[16] Magnussen BF, Hjertager BH. On mathematical models of turbulent combustion with special emphasis on soot formation and combustion. *16th Symp. (Int'l.) on Combustion*. The Combustion Institute, 1976.

[17] Thomas P.H. Ninth Symposium (International) on Combustion, Academic Press, New York, 1963, pp: 844-859.

[18] Kim MB, Jang YJ, Kim JK. Burning rate of a pool fire with downward-directed sprays. Fire Safety Journal 1996; 27: 37–48.

[19] Huo R, Chow WK, Jin XH, Li YZ, Fong NK. Experimental studies on natural smoke filling in atrium due to a shop fire. Building and Environment, 2005, (41), pp: 1185-1193.