No. 53, 38-46.

Saito, K. and Cremers, C. J., **1995**, "Fire Whirl Enhanced Combustion", Proceedings of the Joint ASME/JSME Fluids Engineering Conference, ASME FED-Vol. 220.

Satoh, K. and Yang, K. T., **1996**, "Experimental Observations of Swirling Fires", 1996 International Mechanical Engineering Congress and Exhibition, Atlanta, GA, ASME HTD-Vol. 335, 393-400.

Satoh, K. and Yang, K. T., **1997**, "Simulations of Swirling Fires Controlled by Channeled Self-Generated Entrainment Flows", Proceedings of the 5th International Symposium on Fire Safety Science, Melbourne, Australia, 201-212.

Satoh, K. and Yang, K. T., **1998**, "Study of Swirling Fires in a Vertical Channel with a Single Gap", Proceedings of the 35th National Heat Transfer Symposium of Japan, Nagoya, Paper No. B321

Satoh, K. and Yang, K. T., **1998**, "Experiments and Numerical Simulations of Swirling Fires due to 2x2 Flames in a Channel with a Single Gap", Proceedings of the 1998 ASME International Mechanical Engineering Congress and Exposition, Anaheim, CA, ASME HTD-Vol.2, 49-56

Satoh, K. and Yang, K. T., **1999**, "Measurements of Fire Whirls from a Single Flame in a Vertical Square Channel with Symmetrical Corner Gaps", paper presented at the 1999 ASME International Mechanical Engineering Congress and Exposition, Nashville, TN, ASME HTD-Vol.364-4,167-173

Satoh, K., Kuwahara, K. and Yang, K. T., **1999**, "Numerical Simulations of Swirling Fires in a Vertical Channel ", Proceedings of the 1999 Japanese Safety Engineering, 58-61.

Field Model Simulations on Air Movement of the Room-Corner Fire Test

W.K. MOK and W.K. CHOW
Department of Building Services Engineering
The Hong Kong Polytechnic University
Hong Kong, China

ABSTRACT

The air flow and temperature distribution induced by a fire in the ISO Room-Corner fire test was studied by field modelling technique. Two fire field models, UNSAFE-N and the one developed by Chow were applied. The experiment on melamine faced particle board carried out by SP was taken as the example to compare with, but not validate the predicted results. Combustion processes were not simulated and so the fire was taken as a heat source with varying heat release rate and burning area fitted from experimental data. It is found that both models can give good prediction on the indoor air flow and temperature induced by the fire.

KEY WORDS: Fire simulations, computational fluid dynamics, room-corner fire tests, aerodynamics.

NOMENCLATURE

a_i, a_p, b_i	coefficient of the linear equations
E_i	mass source errors
ρ	air density
k	turbulent kinetic energy
R_p	residual source term
R_ϕ	absolute residual error
R_{ref}	reference value of R_ϕ
T	absolute temperature
t	time
u, v, w	velocity components in directions x, y and z respectively
Γ_ϕ	effective exchange coefficient of the property ϕ
ε	dissipation rate of turbulent kinetic energy
ϕ	flow variables of air

$\Delta\tau$	control volume
μ_t	turbulent viscosity of air
μ_l	laminar viscosity of air

INTRODUCTION

Field models or application of Computational Fluid Dynamics (CFD) for fire simulations appeared in the literature for over two decades [e.g. 1-3]. Very detailed information on air flow and temperature induced by a fire are predicted. Results can be taken as pseudo experimental data which are useful for deriving semi-empirical equations. However, people are still interested in knowing how good a field model is, and the points to consider while using it.

On the other hand, contribution to fire growth by a surface product can be tested by ISO 9705 [4], which is becoming more and more important [e.g. 5]. In the test room with a single open doorway, a fire under well-ventilated conditions starting from a corner was studied. Relevant data for the early stage of a fire can be provided. Materials to be tested are installed along the walls and sometimes the ceiling as well. A square burner is placed at the corner of the room but in contact with the testing specious surface. The burner would emit presetted values of heat release rates by controlling the gas flow rate. This arrangement will give a condition worse than burning a flat surface. It is interesting to understand the indoor air flow patterns and temperature distribution induced by the fire. Reporting such a study becomes the objective of this paper. Experimental results reported by Swedish Technical Institute (SP) [6] were used to justify, but not validate the numerical predictions.

Only the thermal aspects considered as the combustion effects have not yet simulated completely in this approach. The fire is taken to be a burning area with the distribution of heat traced back from experiment. Two CFD codes for studying building fire field are used. The first one is the UNSAFE-N model of Yang [e.g. 3,7] where an algebraic model is used. The second one is a self-developed package by Chow [e.g. 8] which is based on the k-ε model. The theories behind the two models were described in the literature [3,7,9-11] and would not be repeated here.

FIRE FIELD MODEL

A fire field model is good for simulating the air flow induced by a heat source. The air flow equations can be set up from the law of conservation of mass, momentum and enthalpy. The air flow is turbulent in nature and in most CFD packages. The time (Reynolds) averages of flow variables were predicted. Instantaneous values of those air flow variables ϕ_t (e.g., air velocity components, enthalpy, mass concentration) are expressed as a sum of the time average value ϕ and the fluctuation ϕ'. The set of equations describing conservation laws on ϕ_t is transformed into a form in ϕ with the fluctuation terms ϕ' separated out through turbulence models. The equations are of the form:

$$\frac{\partial(\rho\phi)}{\partial t} + \mathrm{div}[\rho\overline{V}\phi - \Gamma_\phi \mathrm{grad}_\phi] = S_\phi \tag{1}$$

where ρ is the air density, $\overline{V}$ is the air velocity vector, Γ_ϕ is the effective exchange coefficient, and S_ϕ is the source term, both for the flow variable ϕ.

Equation (1) will be discretized in the following form at a point p:

$$a_p\phi_p = \sum_i a_i\phi_i + b_p \tag{2}$$

The coefficients a_p, a_i and b_p are described elsewhere.

Key notes for UNSAFE-N [3,7] are:

- Algebraic model for describing turbulence;
- Upwind differencing scheme in descretizing the conservation equations;
- SIMPLE algorithm for solving the pressure-velocity linked equations.

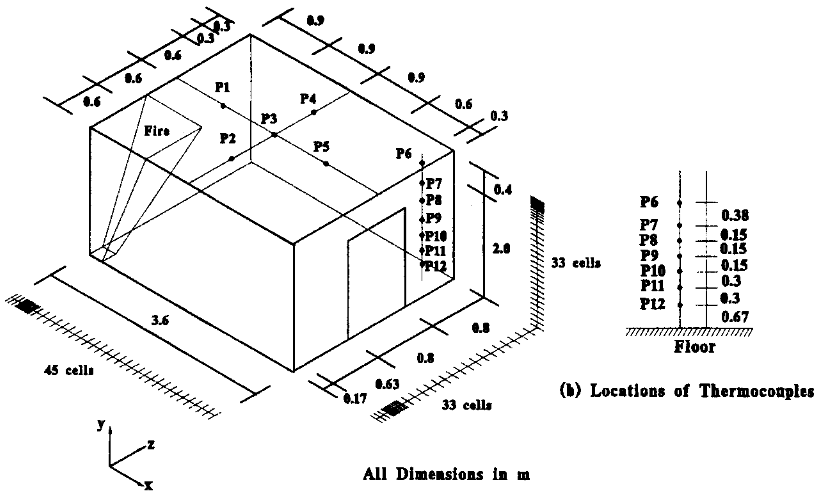
The following are the key points in the field model by Chow [8]:

- k-ε model;
- Power law difference scheme;
- SIMPLER algorithm.

NUMERICAL EXPERIMENTS

For the ISO 9705 [4], a test room is constructed of non-combustible materials with length 3.6 m ± 0.05 m; width 2.4 m ± 0.05 m; and height 2.4 m ± 0.05 m. A doorway is constructed at the centre of the shorter wall of width 0.8 m ± 0.01 m; and height 2.0 m ± 0.01 m. Apart from that, no other openings are allowed. A 0.17 m square propane gas burner is taken as the ignition source. It is placed on the floor at a corner opposite to the wall with the doorway. Uniform gas flow is achieved over the entire opening area of the burner by passing propane gas through porous and inert materials such as sand. The burner is put in contact with the specimen of the surface materials to be tested.

The geometrical configuration of the chamber in the SP experiment [6] was of length 3.6 m, width 2.4 m and height 2.4 m as shown in Fig. 1a. Temperatures were measured in 12 points in locations as shown in Fig. 1 as well. Thermocouples P1 to P6 were used to measure the temperature distributions at the ceiling. The vertical temperature profile inside the room was measured by thermocouples P6 to P12 as shown in Fig. 1b. Those key positions were marked while working out the computing domains. The room was divided into 49005 parts with 45, 33 and 33 parts along the x-, y- and z-directions.



(a) Geometrical configuration

(b) Locations of Thermocouples

FIGURE 1. Geometry of the room

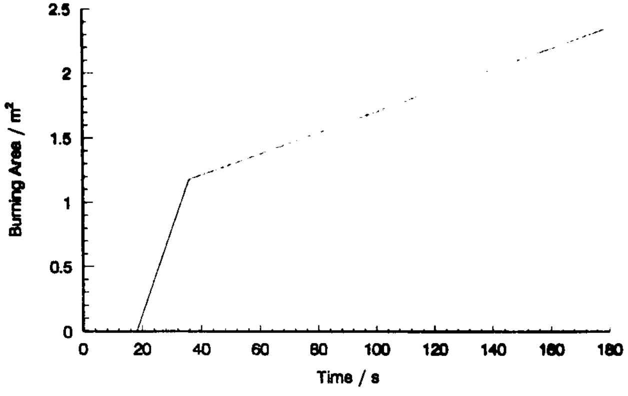
The fire was coupled with a burner of size 0.17 m by 0.17 m, height 1.0 m located at the corner of the chamber; and the burning area increased as in Fig. 2. The heat release rate of the burner increased linearly from 0 to 100 kW in the first ten minutes.

The set of test results [6] on melamine faced particle board was selected to assess the two field models. There, melamine faced particle boards were placed in the ceiling and the other three walls except the one with the door. The lining product was ignited by the propane gas burner. It was observed from the recorded video taken during the tests that fire spread along the walls and ceiling. The estimated transient burning area and the rate of heat release including the thermal power due to the burner are shown in Figs. 2a and 2b respectively.

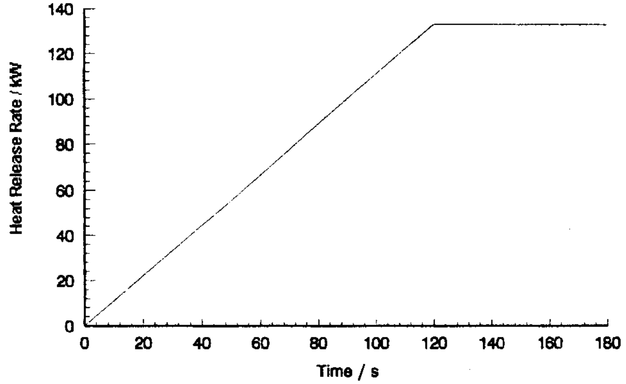
Simulations were performed with UNSAFE-N [3,7] and the self-developed CFD code by Chow [8] in an Intel-Pentium 133. The time step was 0.005 s for UNSAFE-N model, and the total CPU time was about 144 hours. The time step was 0.25 s for the model by Chow [8], and the total CPU time was about 384 hours. Two types of boundary conditions: solid wall boundary and free boundary, were used in the numerical experiments with a summary reported earlier.

CONVERGENCE CRITERIA

The following convergence criteria similar to those for the Gauss-Seidel iteration method are used in solving the set of equations given by (2):



(a) Burning Area



(b) Heat Release Rate

FIGURE 2. Burning parameter

$$\frac{\sum |a_{nb}|}{|a_p|} \leq 1 \quad \text{for all equations} \tag{3a}$$

$$\frac{\sum |a_{nb}|}{|a_p|} < 1 \quad \text{for at least one equation} \tag{3b}$$

A residual source term R_p at the point p is defined as:

$$R_p = a_p\phi_p - \left(\sum a_{nb}\phi_{nb} + b\right) \quad (4)$$

The sum of absolute residual errors appeared in solving for variable ϕ is checked. The value has to be small, e.g. less than 10^{-6}.

Convergence is assumed if the sum of absolute residual errors R_ϕ for any variable ϕ is less than a reference value R_{ref}.

$$R_\phi = \sum \left| R_p \right| < R_{ref} \quad (5)$$

This condition must be satisfied before marching to the next time step. Further, the requirements of having positive coefficients and negative values for the slope of the source term S_p are followed throughout the computation.

For UNSAFE-N [3,7], R_ϕ is the sum of the mass and R_{ref} is 5% of the total mass. For the field model by Chow [8], R_p is the differences of predicted temperature [e.g. 9-11]. Value of R_p must be less than 1 in all grid points before marching to the next time step.

Further, the mass source error E_i (in kgs^{-1}) for the ith control volume $\Delta\tau$ was computed by the equation of continuity:

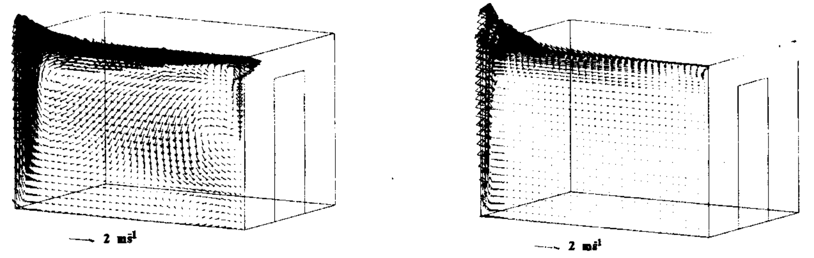
$$E_i = \frac{\Delta\rho}{\Delta t}\Delta\tau + \left[\frac{\Delta(\rho u)}{\Delta x} + \frac{\Delta(\rho v)}{\Delta y} + \frac{\Delta(\rho w)}{\Delta z}\right]\Delta\tau \quad (6)$$

The mass source error m_E (in kgs^{-1}) is given by:

$$m_E = \sum_i \left| E_i \right| \quad (7)$$

RESULTS

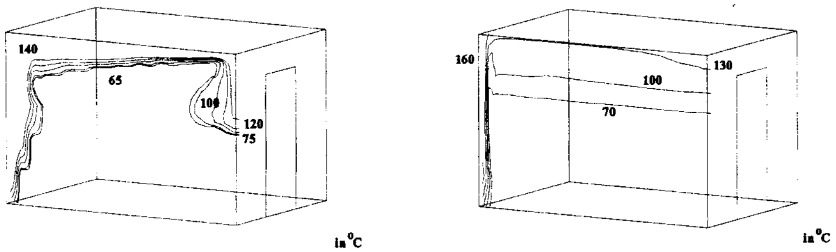
Air flow and temperature fields predicted by the two fire field models across the vertical z-plane (the third plane) of the fire at 180 s after burning are shown in Fig. 3. An upward air current induced by the fire was predicted by both models. Its motion diverted to form a ceiling jet upon reaching the roof, then moved horizontally outward. Results on the velocity vector predicted by the two models are similar, though there are some differences in positions near the door. This is consistent with the predicted temperature contours shown in Fig. 3b. A stable thermal stratified layer of hot air was predicted by UNSAFE-N [3,7]. This hot air layer was also predicted by Chow's model [8] but shifted to the upper part of the room. However, the temperatures predicted by both models at the upper regions are similar.



(i) Chow

(ii) UNSAFE-N

(a) Velocity



(i) Chow

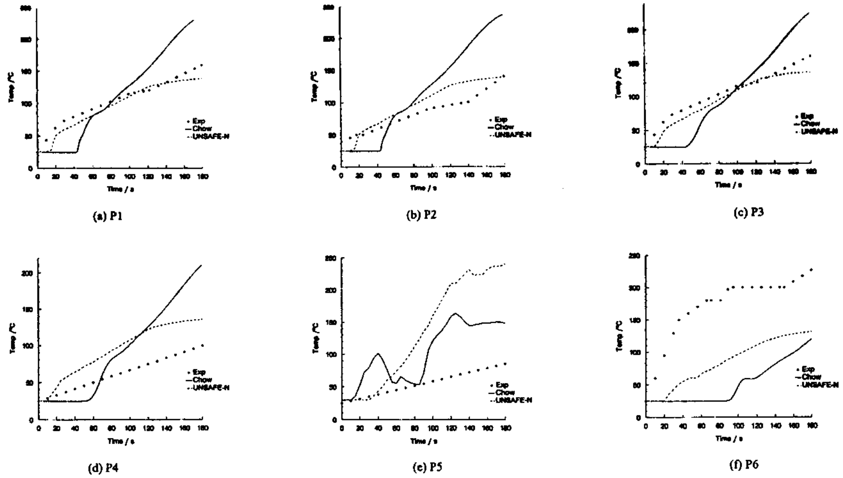
(ii) UNSAFE-N

(b) Temperature

FIGURE 3. Predicted flow field at a vertical plane across the fire

The transient temperature predicted at thermocouples P1 to P12 are shown in Figs. 4 and 5. On the ceiling positions P1 to P6, results predicted at P1 and P3 by UNSAFE-N agreed well with the experiment, however, the agreement was not so good for Chow's model. At P2, results predicted by UNSAFE were also better than Chow's model, though both sets of predicted curves deviated away from the experimental curve. The two models deviated quite far away from the experiments at P4 and P5. There were large deviations on the predicted curves by the two models from the experimental results at P6.

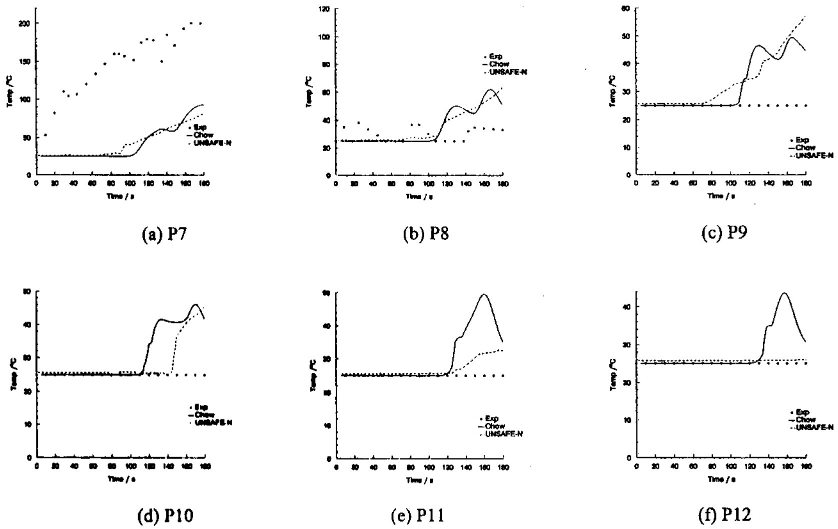
For the vertical distributions near the doorway, results predicted by both models were similar from P7 to P10, but deviated quite far away from the experiments. There are bigger deviations for results predicted by Chow's model from experiment than those by UNSAFE-N.



(a) P1 (b) P2 (c) P3

(d) P4 (e) P5 (f) P6

FIGURE 4. Comparison of temperatures: P1 to P6



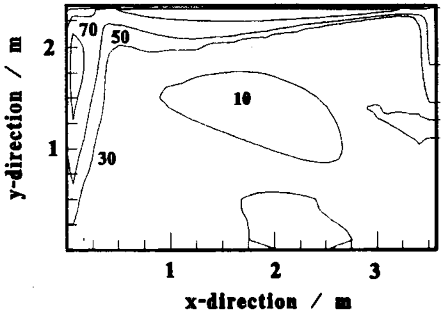
(a) P7 (b) P8 (c) P9

(d) P10 (e) P11 (f) P12

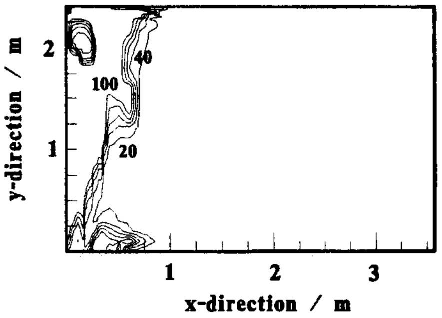
FIGURE 5. Comparison of temperatures: P7 to P12

Contours for the turbulent viscosity μ_t expressed in terms of a multiple of the laminar viscosity μ_ℓ across that vertical fire plane are presented in Fig. 6. Both models indicated that the turbulent viscosity was at most 100 μ_ℓ at the positions above the gas burner.

The contours for turbulent kinetic energy k and dissipation rate of turbulent energy ε across the vertical fire plane predicted by Chow's model are shown in Fig. 7. The value of k was up to 0.2 Jkg and value of ε up to 0.3 $\mathrm{Jkg^{-1}s^{-1}}$.

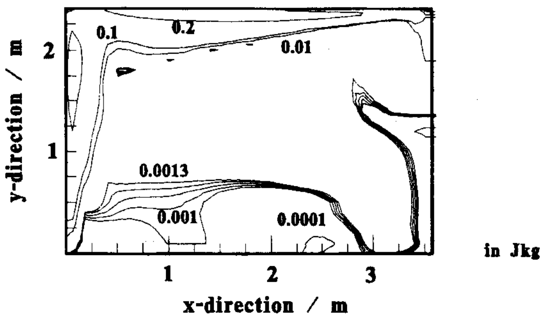


(a) Chow

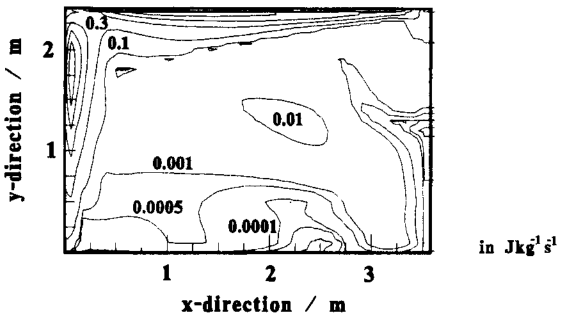


(b) UNSAFE-N

FIGURE 6. Predicted turbulent to laminar viscosity ratios at a vertical plane across the fire



(a) Turbulent kinetic energy



(b) Dissipation rate of turbulent kinetic energy

FIGURE 7. Predicted turbulent parameters at a vertical plane across the fire

Comparison of the mass source errors for the two models is shown in Fig. 8. Note that the value of m_E was up to 16 kgs^{-1}. This means that the average value of E_i at each cell was very small, i.e. 3×10^{-4} kgs^{-1}.

Lastly, the mass flow rates of air drawn in and out through the door for the two models are shown in Fig. 9.

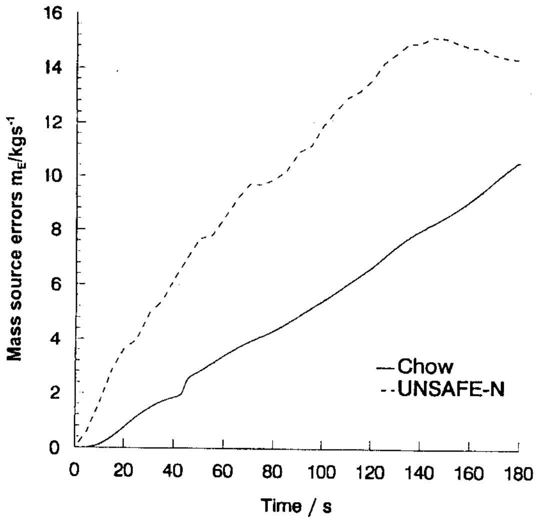


FIGURE 8. Rate of mass source errors

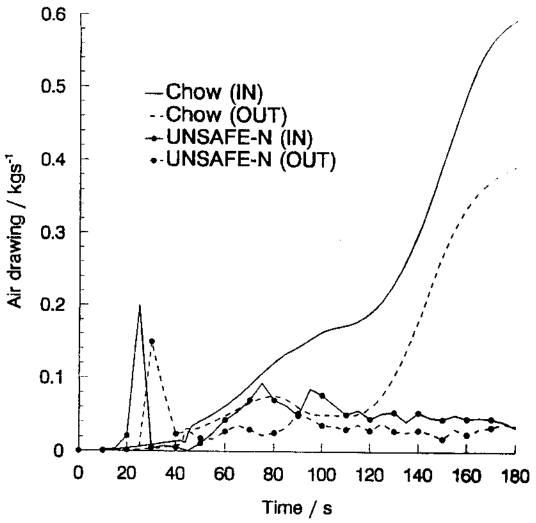


FIGURE 9. Rate of mass flowing through the door

CONCLUSION

Two fire field models, UNSAFE-N [3,7] and the one by Chow [8], were applied to simulate the air flow pattern and temperature distribution induced by a fire while carrying out the ISO room-corner fire test [4]. Experiments on the melamine faced particle board carried out by SP [6] was taken as the example for comparison.

- It is found that both fire field models are able to simulate the air flow and temperature distribution. However, the UNSAFE-N gave a better prediction on comparing with the experiment. A possible explanation is that adopting the algebraic turbulence model might give better prediction.

- The transient heat release rate and burning area curves fitted from experiments were taken as the input heat source. Such an approach is practical when combustion process is not simulated. But it is difficult to develop a fire field model with combustion process as the phenomena on turbulence, combustion and thermal radiation have to be included [1,2].

- For the turbulent viscosity μ_t, both models illustrated that the values of μ_t would be about 100 μ_ℓ in most parts of the room. This point has to be considered carefully in understanding the turbulent air flow.

- The convergence criteria in solving the equations are reviewed. Typical results presented in Figs. 8 and 9 should be noted carefully in using the field models.

ACKNOWLEDGEMENT

The authors wish to thank Professor K.T. Yang for allowing the use of UNSAFE-N and his advices on this subject area. The project is funded by PolyU under research account G-V470.

REFERENCES

1. Bilger, R.W., "Computational Field Models in Fire Research and Engineering", in Proceedings of the Fourth International Symposium on Fire Safety Sciences, ed. T. Kashiwagi, pp. 95-110, International Association for Fire Safety Science, 1994.

2. Lewis, M.I., Moss, M.B. and Rubin, P.A., "CFD Modelling of Combustion on Heat Transfer in Compartment Fires", in Proceedings of the Fifth International Symposium on Fire Safety Sciences, ed. Y. Hasemi, p. 463-474, International Association for Fire Safety Science, 1994.

3. Yang, K.T., "Role of the fire field models as a design tool for performance-based fire-code implementation", International Journal on Engineering Performance-Based Fire Codes, 1:1, 11-17, 1999.

4. International Standard ISO, "Fire Tests - Full-Scale Room Test for Surface Products", ISO 9705, 1993(E).

5. SFPE / NFPA, Handbook of Fire Protection Engineering, SFPE / NFPA, U.S.A., 1988.

6. Sundstrom, B., "Full Scale Fire Testing of Surface Materials, Measurements of Heat Release and Productions of Smoke and Gas Species", Swedish National Testing Institute, Fire Technology, Technical Report SP-RAPP 45, 1986.

7. Yang, K.T. and Nies G.F., "Numerical Field Model Simulation of Full Scale Fire Tests in a Closed Vessel", Thesis, Naval Postgraduate School, Monterey, California, U.S.A., 1986.

8. Chow, W.K. and Leung, W.M., "Solid-Wall Boundary Effect on a Building Fire Field Model", Combustion Science and Technology, 71:1, 77-93, 1990.

9. Chow, W.K. and Mok, W.K., "On the Simulation of Forced-Ventilation Fires using Three Field Models", in Proceedings of 29th Australian and New Zealand Architectural Science Association Conference, 11-13 July 1995, pp. 62-66, University of Canberra, Canberra, Australia, 1995.

10. Chow, W.K. and Mok, W.K., "On the Simulation of Forced-Ventilation Fires", Numerical Heat Transfer, Part A: Applications, 28:3, 321-338, 1995.

11. Chow, W.K. and Mok, W.K., "CFD Fire Simulations with Four Turbulence Models and Their Combinations", Journal of Fire Sciences - Accepted to publish, August, 1999.

isolal