Numerical Prediciton of Heat Flux from Flame in Room Fire

Yan, Zhenghua; Holmstedt, Göran

1997

LUND UNIVERSITY

PO Box 117
221 00 Lund
+46 46-222 00 00

Brandteknik
Lunds tekniska högskola
Lunds universitet

Department of Fire Safety Engineering
Lund Institute of Technology
Lund University

Report 3090

Numerical Prediction of Heat Flux from Flame in Room Fire

Research financed by the Swedish Fire Research Board (BRANDFORSK)

Yan Zhenghua
Göran Holmstedt

Lund 1997

Numerical Prediction of
Heat Flux from Flame in Room Fire

Yan Zhenghua
Göran Holmstedt

ISSN 1102-8246
ISRN LUTVDG/TVBB--3090--SE

Brandforsk projekt No. 058-951

Keywords: Computational Fluid Dynamics, combustion, heat transfer, room fire

Omslag: Maria Andersen

Layout: Yan Zhenghua

Figurer: Yan Zhenghua

Department of Fire Safety Engineering · Lund Institute of Technology · Lund University

Adress/Address	Telefon/Telephone	Telefax	E-post/E-mail
Box 118 /John Ericssons väg 1	046 - 222 73 60	046 - 222 46 12	
S-221 00 LUND	+46 46 222 73 60	+46 46 222 46 12	Goran.Holmstedt@brand.lth.se

CONTENTS

ABSTRACT

A number of CFD (Computational Fluid Dynamics) calculations were carried out to simulate the large scale room corner fire, which is an important scenario for the evaluation of the fire performance of the surface lining material. Considered are turbulent gas flows, turbulent combustion, radiation and heat conduction inside solid boundary. Heat transfer from flame and hot gas is calculated, with the important radiation component presented by discrete transfer (DT) method and the convection heat transfer considered by the wall function. An absorptivity and emissivity model was employed to predict the radiation property of combustion products including soot, CO_2 and H_2O, which are usually the primary radiating species in the combustion of hydrocarbon fuels. Configurations are a square burner flame in the corner of the standard full scale fire room, with three different standoff distances: 0 cm, 5 cm and 10 cm, and two different burner outputs: 40 kW and 150 kW. Totally, six cases were studied. The results, including the temperature and heat fluxes, are discussed and compared with experimental measurements.

ACKNOWLEDGEMENTS

This work was supported by the Swedish Fire Research Board (BRANDFORSK), which is gratefully acknowledged.

INTRODUCTION

Heat transfer to a solid surface is critical to assessing flame spread and fire growth. In fire, the virgin combustible material is heated up by the heat flux from the flame and hot gas, and then ignited. Therefore, the ability to predict heat flux is very important in fire protection engineering. Unfortunately, prediction of heat flux remains difficult, since the heat transfer to a solid surface can be affected by a large amount of factors including temperature distribution, flow characteristics, gas properties and solid properties, etc.

Quintiere et al. [1] tried to seek a correlation for flame heat flux in terms of the configuration and fuel properties. Several different configurations including a line fire against a wall, a square burner flame against a wall and in a corner, and window flame impinging on a wall were studied. By using dimensional analysis, they attempted to correlate the heat flux as a function of several dimensionless variables. However, due to the complexity of the problem, no general correlation was developed.

Field modeling, which is based on the CFD technology, has made a significant contribution to fire research over the last decade. A very rapid and great progress has been made since it emerged in the late 1970s. In the early 1980s, field models of enclosure fires were steady and 2-dimensional. Combustion and radiation were not included. A few years later, the calculations were extended to 3-dimensional and transient problems. Nowadays, with the complex radiation included, it can be used to calculate heat flux and predict flame spread [2, 3]. As the progress is made in CFD technology and computer power increases at decreasing cost, it is expected for field modeling to play an increasingly important role in fire research. Moreover, since the basis of field modeling is the fundamental equations describing the physical sub processes of fire, it can provide detailed information on these processes, and once well established and validated, it can be generally applicable to different fire scenarios.

This paper presents a preliminary CFD study of the heat fluxes in a room fire. Configurations include a square burner flame in the corner of the standard full scale fire room, at three different standoff distances: 0 cm, 5 cm and 10 cm, and with two different burner outputs: 40 kW and 150 kW. Totally 6 cases were studied. The numerical results are discussed and compared with the experimental measurements made by Williamson et al [4].

THEORETICAL MODEL

In fire, the involved sub processes include turbulent flow, turbulent combustion, radiation and heat transfer inside solid boundary, etc. Computer modeling of the fire is based on the numerical solution of a set of mathematical equations, which describe the involved fire sub processes.

The Fluid Dynamics

The turbulent flow in fire is controlled by the fluid dynamics, which is mainly represented by a set of partial differential governing equations [5, 6]. The general form of the governing equations can be written as

$$\frac{\partial(\rho\varphi)}{\partial t}+\frac{\partial(\rho u_i\varphi)}{\partial x_i}=\frac{\partial}{\partial x_i}(\Gamma_{ij}\frac{\partial\varphi}{\partial x_j})+S_\varphi \qquad (1)$$

Fire is usually a buoyancy-controlled turbulent process. The standard $k-\varepsilon$ model, adapted to incorporate the buoyancy effect, was used to study the turbulence characteristics of the gas flow.

Combustion Model

Flame in fire is a typical turbulent buoyant diffusion flame with very low source momentum. The buoyancy promotes the irregular motion which governs the rate of mixing of fuel volatile and air. The rate of the reaction of fuel and air is thus controlled by the relatively slow turbulent mixing process. In this study, combustion was simulated by one-step chemical reaction, where complete oxidation is assumed when sufficient oxygen is available, and the local reaction rate is determined by the eddy dissipation combustion model to be the slowest of the turbulence dissipation rates of either fuel (propane in this study) or oxygen [7],

$$R_{fu} = -\rho\frac{\varepsilon}{\kappa}\min(C_R m_{fu,} C_R \frac{m_{ox}}{S}) \quad (2)$$

Radiation

Radiation is an important heat transfer mechanism in fire, especially in a large fire. In the flame spread problem, the fuel ahead of the flame is mainly heated by the flame radiation. In this study, discrete transfer method was adopted to calculation the radiation, with Modak's simple model to present the radiation properties of the combustion products.

Discrete transfer model

The discrete transfer (DT) method was developed by Lockwood et al [8]. DT is one of the most popular methods used in the numerical calculation of radiation. This method has good accuracy, flexibility and it is also suitable for the calculation of the oblique radiation which is important to flame spread.

The radiation source term of the energy equation of the gas phase and the radiation flux to the solid surface are given by

$$S_r = \oiint I d\vec{\Omega} d\vec{A} = \sum_{Rays} (I_{n+1} - I_n)\vec{\Omega} \bullet \Delta\vec{A}\Delta\Omega \quad (3)$$

$$R_{flux} = \sum_{Rays} I_w(\vec{\Omega} \bullet \vec{n})\Delta\Omega - \varepsilon_w \sigma T_w^4 \quad (4)$$

where I_n is the intensity on entry and I_{n+1} is the intensity on exit of the control volume. I_w is the radiation intensity incident on the boundary surface. The radiation intensities are provided by the solution of the radiation equation [3] along a discrete set of rays from every element of the boundary surface.

Radiation property model

According to Modak's model [9], the absorptivity of a homogeneous and isothermal mixture of soot, CO_2 and H_2O is calculated by

$$\alpha = \alpha_g + \alpha_s - \alpha_g \alpha_s \tag{5}$$

where α_g is the absorptivity of CO_2 and H_2O approximated in a manner similar to that suggested by Hottel and co-workers [10, 11], α_s is the soot absorptivity which is presented by

$$\alpha_s = 1.0 - \frac{15}{\pi} \psi^{(3)} (1 + \frac{\lambda_0 \kappa_0 T_s l}{c_2}) \tag{6}$$

where $\psi^{(3)}$ is the pentagamma function; c_2 is Planck's second constant; T_s is the source temperature; l is the pathlength; $k_0 \cong 7 f_v / \lambda_0$; $\lambda_0 = 0.94 \mu m$ and f_v is the soot volume fraction.

Consideration of soot

Soot contributes significantly to the radiation in fire. In order to calculate radiation accurately, soot must be considered. Unfortunately, sooting is very complex and no good soot model is available for the building fires at the moment. In this paper, as an approximation, soot was considered by assuming a constant soot conversion factor, 4%, chosen with reference to experimental measurement [12]. The measurement was made after some of the generated soot was oxidised. The measured value, 2.4%, is therefore increased to 4% in this study. No optimum choice was made. The soot formation rate was simply assumed to be locally proportional to the fuel consumption rate. No oxidation was considered. An additional transport equation which can be written in the general form of Eq.(1) was solved to calculate the soot mass concentration.

The soot volume fraction, which is central to the radiation calculation, was simply determined from the soot mass concentration by assuming a constant soot density of $1800\, kg/m^3$.

Heat Transfer Inside Solid Boundary

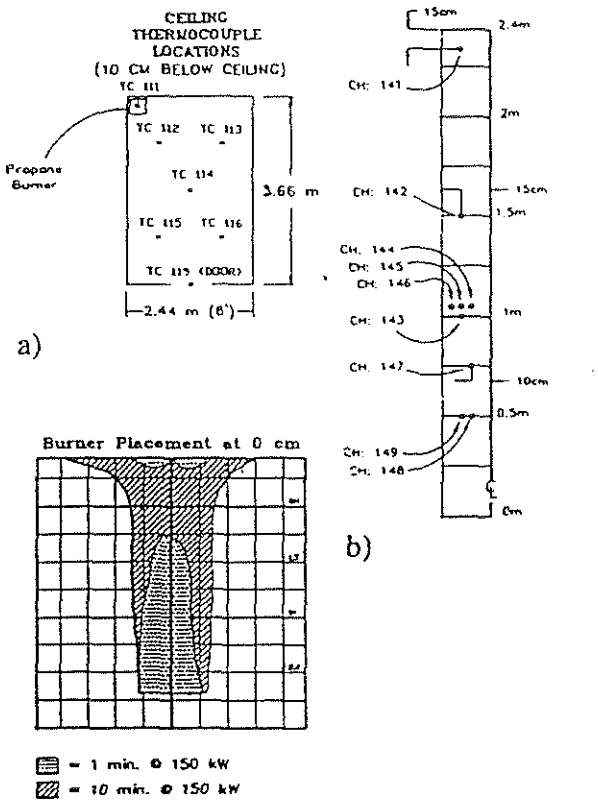
During fire, the solid wall is heated up through heat transfer including convection and radiation. Usually, only the heat conduction perpendicular to the face is important. Thus, the heat transfer in the wall is simplified to a one-dimensional transient process. In this study, the solid wall surface is divided into many elements, according to the CFD grid generation. For each boundary element, the following equation is numerically solved:

$$\frac{\partial(\rho H)}{\partial t} = \frac{\partial}{\partial x}(k\frac{\partial T}{\partial x}) \tag{7}$$

where H is the enthalpy given by $\int c_p dT$. c_p and k are specific heat and conductivity, respectively.

BRIEF DESCRIPTION OF THE SIMULATED EXPERIMENTS

The experiment carried out by Williamson et al [4] was a standard large scale room corner fire. The room dimensions are 2.4m(w)$\times$3.6m(l)$\times$2.4m(h). The front wall, which is, 2.4m(w)$\times$2.4m(h), contains a 0.76m(w)$\times$2.03m(h) opening. All the walls and ceiling were lined with gypsum board. A 0.3m$\times$0.3m square propane gas burner was located at the room corner, at various standoff distance: 0 cm, 5 cm and 10 cm. The output of the gas burner was programmed to follow a Rate of Heat Release (RHR) protocol similar to that specified in the UBC42-2 test procedure. In the experiments, temperature of the gas directly above the burner and 10 cm below the ceiling, and heat flux on the surface of the side wall were measured. The thermocouples were fabricated from chromel-alumel wire with about 1.7 mm diameter beads and the heat flux gauges were of "Schmidt-Boelter" and "Gardon" types. The measurement locations are shown in Figs. 1(a-b).



c)

Figure 1 Experimental measurement locations and burn pattern in the case at 0 cm standoff distance and 150kW RHR level a) Gas temperature measurement locations b) Heat flux measurement locations c) Burn pattern

RESULTS AND DISCUSSIONS

The experimentally measured RHR indicates that the contribution of gypsum board to the heat release rate is negligible [4]. Thus, the effect of gypsum board on RHR is ignored in the numerical simulation. In order to save computation time, the experimental output [4] of the gas burner was not followed in the simulation. Each of the experiments was divided into two study cases, at the burner output levels of 40 kW and 150 kW respectively. For each study case, the fire process of 1 minute was simulated. In the following, all the results presented are those at the time of 1 minute. The calculation was tested with various numbers of rays and grid sizes, showing that the present choice gives practically grid-independent and ray number-independent predictions.

Table. 1

Calculated and measured temperatures of the gas in the corner , above the burner.

Scenarioes	40kW 0cm standoff	40kW 5cm standoff	40kW 10cm standoff
Measured Temp. (K)	420	415	400
Predicted Temp. (K)	519	505	468
Scenarioes	150kW 0cm standoff	150kW 5cm standoff	150kW 10cm standoff
Measured Temp. (K)	870	730	630
Predicted Temp. (K)	1030	930	780

Table. 2

Average temperatures of the gas 10 cm below the ceiling

Scenarioes	40kW 0cm standoff	40kW 5cm standoff	40kW 10cm standoff
Measured Temp. (K)	360	355	350
Predicted Temp. (K)	406	405	404
Scenarioes	150kW 0cm standoff	150kW 5cm standoff	150kW 10cm standoff
Measured Temp. (K)	560	520	490
Predicted Temp. (K)	562	556	550

Table 1 is the comparison of the calculated and measured temperatures of the gas 10 cm below the ceiling in the corner of the test compartment, directly above the gas burner. The comparison shows that the trend is reasonably good. Both the measurement and the prediction show that the temperature decreases with the increase of the standoff distance. This is most likely attributed to differences in plume entrainment resulted from the presence of the free space between the plume and the solid corner boundary. However, from the comparison, one can see clearly that the calculated value is higher than the measured in all the studied cases. Perhaps, one important reason is that the air entrainment into the plume was possibly under-predicted. Another possible reason is the radiation effect on the thermocouple, which could make measured temperature lower than the real value.

The predicted and measured average temperature of the gas 10 cm below the ceiling are compared in Table 2. As illustrated in the table, the measured average temperature is generally well reproduced by the CFD calculation. Again, the temperature of the gas is over-predicted in most of the study cases and it decreases, but only slightly, with the standoff distance in both calculation and measurement.

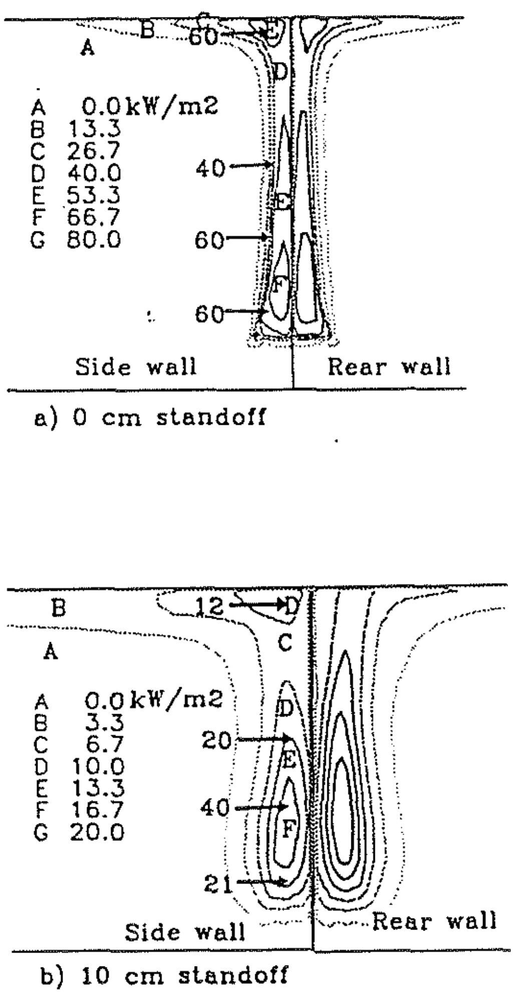
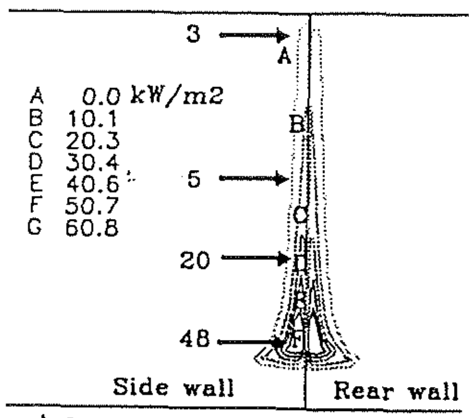
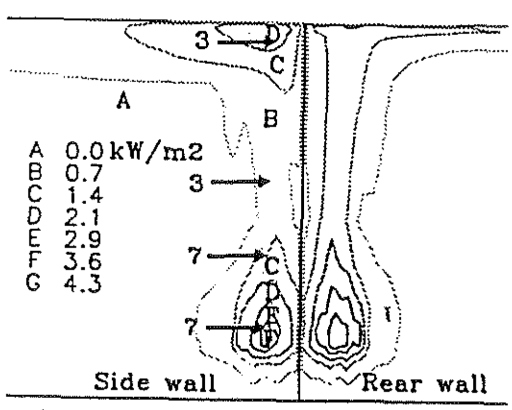


Figure 2 Comparison of calculated and measured heat flux, RHR=150 kW a) 0 cm standoff b) 10 cm standoff

At the RHR levels of 40 kW and 150 kW, the comparisons of the calculated and measured heat fluxes are shown in Figs. 2(a-b) and Figs. 3(a-b) respectively. Due to the limited space, the results for the case of 5 cm standoff distance are not presented The measured data at the location pointed by the arrow is indicated by the number and the calculation is represented by contour plot. The letters marked inside the contour plot show the heat flux range. For example, in Fig. 2(a), the heat flux on surface marked with 'E' is between 53.3 and 66.7 kW / m^2. Generally, the measured heat flux was reasonably well predicted. Both calculation and experiment present a significant variation of heat flux with elevation and a strong dependence of heat flux on the flame size and the standoff distance. However, the comparison shows that significant disparities exist at some specific locations. At 10 cm standoff distance, the predicted heat flux is significantly lower than the measured at the ch:143 (Fig. 1) location, at both RHR levels. The calculation shows the heat transfer is dominated by radiation at this location. Due to the complexity of the problem, no specific reason was found to be attributed to this result.



a) 0cm standoff



b) 10 cm standoff

Figure 3 Comparison of calculated and measured heat flux, RHR=40 kW a) 0 cm standoff b) 10 cm standoff

One interesting observation of the experiment is that in the case of 150 kW RHR and 0 cm standoff distance, the heat flux measured at ch:141 (Fig. 1) is significantly higher than that at ch:142. This is well reflected by the char pattern shown in Fig. 1(c), which indicates that there are hot spots on the walls in the top corner. We will denote this phenomenon as 'hot spots' in the following text.

The 'hot spots' is well reproduced by the CFD calculation. Fig. 2(a) clearly shows that in the case of 150 kW RHR and 0 cm standoff distance, the calculated heat flux on the walls in the top corner is much higher than that on the walls at the middle height. In this simulation, the burning of gypsum board is ignored, thus no calculated char pattern is available. However, the wall temperature was calculated and we can reasonably regard the wall surface temperature distribution as a rough and qualitative representative of the char pattern. Fig. 4 shows the wall surface temperature distribution. In order to have a better contrast effect of the graphic presentation, Fig. 4 was drawn by using a proper threshold value. The heat flux below the threshold value is omitted in the figure. Fig. 4 corresponds to Fig. 2(a) and agrees quite well with the experimental observation shown in Fig. 1(c).

The calculation also shows that the heat flux on the side wall at the top corner is a little higher than that on the side wall at the middle height in the cases at 10 cm standoff distance (Fig. 2(b) and Fig. 3(b)). According to the calculation itself, this is attributed to the relative higher convection heat transfer at the top corner. Since the heat flux difference is quite small, even if it exists in reality, it could be difficult to be seen from the char pattern.

No explanation for the 'hot spots' was given by the experiment. Perhaps, the CFD calculation can present some insight. According to the calculation, in the case at 0 cm standoff distance and 150 kW RHR, some gas fuel was left to the top corner, and due to the impingement of the plume on the ceiling and the geometry effect, the turbulence in the top corner was strong. The strong turbulence consequently promoted the combustion, as indicated by the calculation shown in Fig. 5. Possibly, the promoted combustion in the top corner is an important reason for the 'hot spots'. Certainly, some other factors could also be responsible.

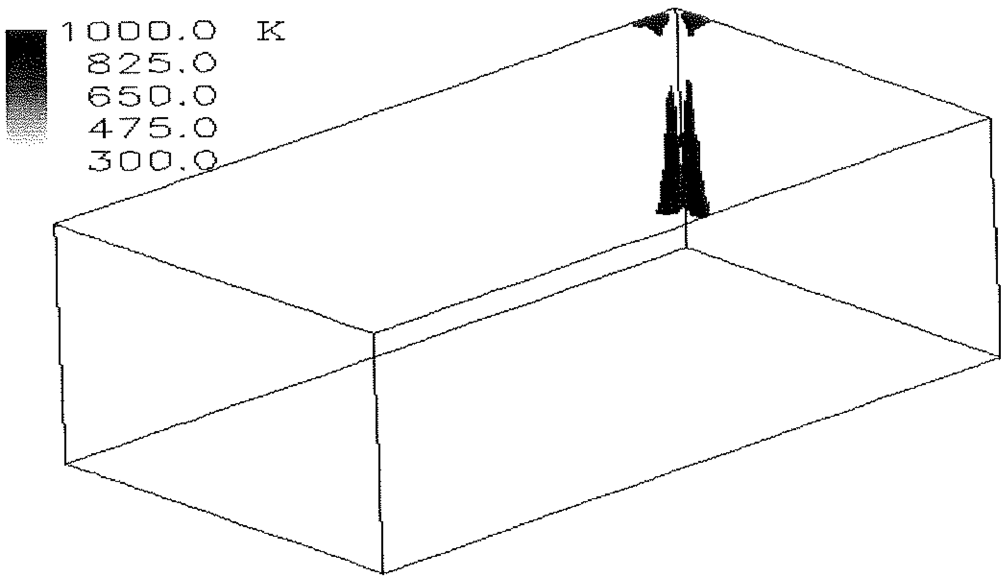


Figure 4 Calculated wall surface temperature, RHR=150 kW, 0 cm standoff, threshold value=810K (to show 'hot-spots')

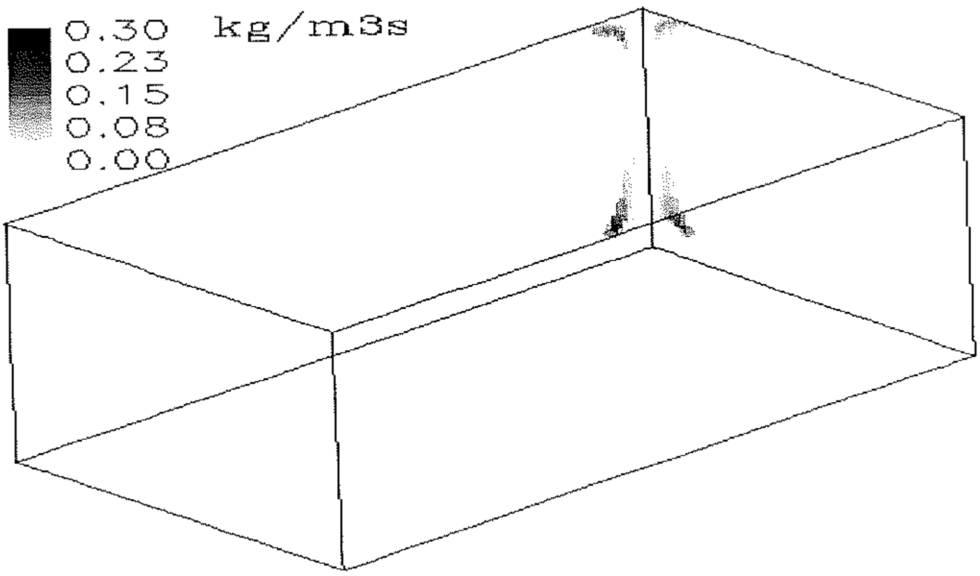


Figure 5 Fuel consumption rate, RHR=150 kW, 0 cm standoff

CONCLUSIONS

The large scale corner fires were simulated and the heat flux from flame and hot gas was calculated by using CFD method, with the important radiation component presented by the DT method and the convection heat transfer considered by the wall function. Several cases were studied by varying the burner output and the standoff distance. The results were compared with the measurements. In all the six studied cases, the simulation agrees generally well with the experimental measurements, although some significant disparity exists. The fairly good agreement indicates that the CFD method can be expected to be used as an important tool to predict the heat flux in fire and the disparity implies that further research in this area is necessary.

The over-predicted gas temperatures suggest that the entrainment was likely to be under-predicted, although the gas temperature also depends on other factors such as the heat loss through the boundary. Since the entrainment has a significant effect on the gas temperature, thus on the heat transfer, more study on the entrainment is necessary.

The heat flux on the solid surface includes two components: radiation and convection. Both radiation and convection are very complex. It is therefore essential for them to be verified separately against the experimental data.

Radiation from soot is important in fire. Consideration of soot needs to be improved. Therefore, a practical and good soot model is very desirable.

REFERENCE

1. J. G. Quintiere and T. G. Cleary, Fire Technol., Second Quarter: 209-231(1994)

2. Zhenghua Yan and Göran Holmstedt, First European Symposium on Fire Safety Science, Zurich, 1995, pp. 95-96

3. Zhenghua Yan and Göran Holmstedt, CFD and Experimental Studies of Room Fire Growth on Wall Lining Materials, submitted to Fire Safety Journal, 1995

4. R. B. Williamson, et al., Fire Safety Science-Proceeding of the Third International Symposium, IAFSS, 1991, pp. 657-666

5. S Kumar and G Cox, 5th International Symposium on the Aerodynamics and Ventilation of Vehicle Tunnels, 1985, pp. 61-76

6. G. Cox and S. Kumar, Comb. and Technol., 52: 7-23 (1987)

7. B. F. Magnussen and B. H. Hjertager, Sixteenth Symposium (International) on Combustion, The Combustion Institute, Pittsburgh, 1976, pp. 719-729

8. F.C. Lockwood and N. G. Shah, Eighteenth Symposium (International) on Combustion, The Combustion Institute, Pittsburgh, 1981, pp. 1405-1414

9. Ashok T. Modak, Fire Research, 1 (1978/1979), 339-361

10. H.C. Hottel and H.G. Mangelsdorf, Trans. Am. Inst., Chem. Eng., 31:517 (1935)

11. H.C. Hottel and R.B. Egbert, Trans. Am. Inst., Chem. Eng., 38:531 (1942)

12. Archibald Tewarson, the SFPE Handbook of Fire Protection Engineering, Second Edition Section 3/Chapter 4