Evaluation of the Field Model, Fire Dynamics Simulator, for a Specific Experimental Scenario

G. W. ZOU

Department of Building Engineering
Harbin Engineering University
Harbin, Heilongjiang, China

W. K. CHOW*

Department of Building Services Engineering
The Hong Kong Polytechnic University
Hong Kong, China

ABSTRACT: The new version of the fire dynamics simulator, FDS version 3.01, is verified by a series of full-scale fire tests. Experiments are carried out in a compartment similar in size to the ISO-9705 room calorimeter. Gasoline pool fires of different diameters are used to give different heat release rates. The ventilation factors of the compartment are adjusted to produce flashover. Fire-induced flow, temperature, and pressure fields in these different physical scenarios are calculated using the FDS software. The predictions for temperature and radiative heat flux of a post-flashover fire in the specific configuration tested are in reasonably good agreement with experimental results even when the flame occupies most of the room volume.

KEY WORDS: large eddy simulation, computational fluid dynamics, flashover fires.

INTRODUCTION

FIELD MODELS OF a building fire, involving the application of computational fluid dynamics (CFD) or numerical heat transfer (NHT), are well established in fire technology (see, e.g. [1]). One of these models, the fire dynamics simulator (FDS) [2,3] was released recently by the National Institute of Standards and Technology (NIST) in the USA.

*Author to whom corresponding should be addressed. E-mail: bewkchow@polyu.edu.hk

1042-3915/05/02 0077–16 $10.00/0 DOI: 10.1177/1042391505047304

The result of years of research on new subroutines, such as combustion chemistry, thermal radiation, and fire suppression, it can simulate the environment induced by a fire [2–5]. As FDS is a large eddy simulation, there are not as many empirical, adjustable parameters as in Reynolds Averaging Navier–Stokes Equations (RANS) [e.g. 1,6]. The realism of the FDS simulation, however, is still open to question. This is the subject of the current paper.

Engineering professionals must be able to evaluate a fire model through comparisons with experimental data. Existing data available for such an evaluation [e.g. 1,4–7] are reported in the literature, though these data are quite old in some cases; in other cases, the heat release rates of the fire were not measured by the oxygen consumption method. Further, most of the fire tests were intended to validate fire zone models [e.g. 1] or to extract measured data for deriving empirical correlation expressions such as airflow rates through doorways. Very few tests were specifically designed for verifying fire field models. As reported by McGrattan et al. [4], the results of experiments on the doorway flow in a small room with different fire sizes, and at different positions by Steckler et al. [8,9] were compared with the predicted results of FDS and another CFD/NHT model based on RANS. This set of tests has been used by many other researchers to assess their CFD/NHT models over the past 15 years. In 1994, smoke transport experiments were carried out [4] by NIST at the Fire Research Institute, Japan. New experimental studies might be useful in confirming the performance of different fire models, both field and zone.

It is difficult to carry out full-scale burning tests [10] in densely populated urban areas such as Hong Kong. Collaborative studies were established between Harbin Engineering University (HEU) at Harbin, Hailongjiang, China and The Hong Kong Polytechnic University (PolyU). A full-scale burning facility known as the PolyU/HEU Assembly Calorimeter was developed. Heat release rates of burning combustibles up to 5 MW can be measured by the oxygen consumption method.

Many full-scale burning tests have been carried out on flashover fires with different gasoline pools. Details of the experiments and preliminary analyses of the results have been reported in the literature [11,12]. A typical set of measured results was selected to compare with those predicted by FDS.

EXPERIMENTAL SETUP

A set of flashover experiments carried out at the PolyU/HEU Assembly Calorimeter [10,12] was selected. A room of length 3.6 m, width 2.4 m, and height 2.4 m was constructed of brick with a cement finish as in Figure 1. The thickness of the brick wall was 0.22 m, with the cement finish

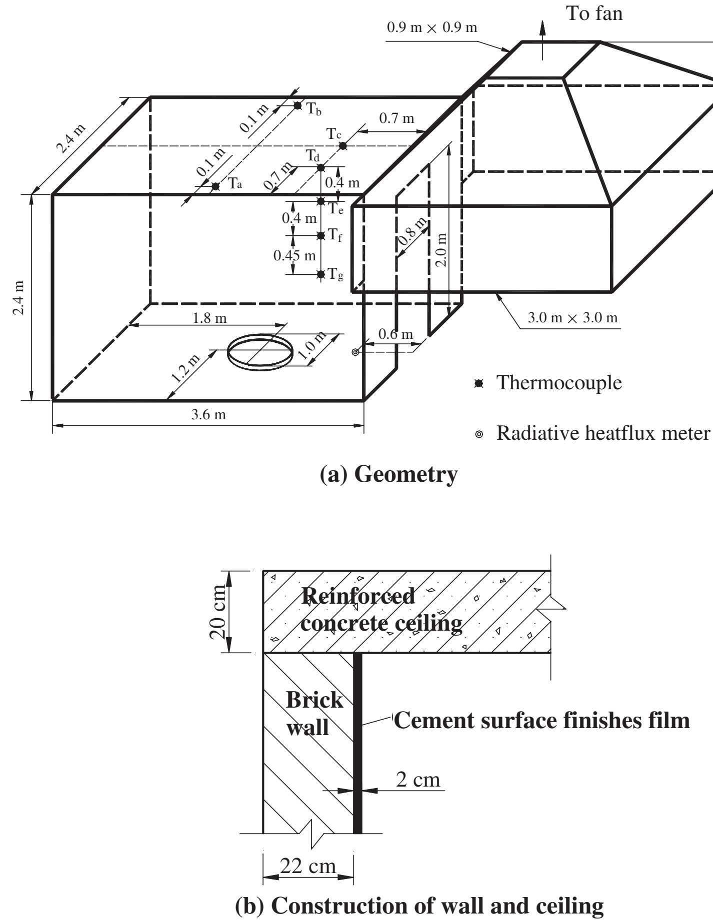


Figure 1. *Full-scale burning facility.*

of thickness 20 mm. The thermal conductivity of brick and cement is 0.93–1.12 $W\,m^{-1}\,K^{-1}$, density is 1560–1860 $kg\,m^{-3}$, and specific heat capacity is 920–1050 $J\,kg^{1}K^{-1}$ [13]. The ceiling was constructed with 0.2 m reinforced concrete with a thermal conductivity of 1.51–1.74 $W\,m^{-1}\,K^{-1}$, density of 2300–2500 $kg\,m^{-3}$, and specific heat capacity of 920 $J\,kg^{-1}\,K^{-1}$. There was a door of height 2.0 m and width 0.8 m. An exhaust hood of length 3 m, width 3 m, and height 2 m was constructed. The volume flux flow rate of the exhaust hood was set to 4 $m^3\,s^{-1}$. The fan-duct system installed can measure heat release rates of up to 5 MW.

A gasoline pool fire of 1 m diameter was placed at the center of the room. Three thermocouples T_a, T_b, and T_c were placed at 30 mm below the ceiling, and a thermocouple tree with four thermocouples T_d, T_e, T_f, and T_g were placed inside the room, 0.7 m from the door and 0.7 m from the left wall. T_d was placed 30 mm below the ceiling. Type K thermocouples of diameter 1 mm were used in locations T_a–T_g as shown in Figure 1(a).

A heat flux meter (model H-201 by Schmidt-Boelter) was placed 0.6 m from the door as in Figure 1(a) to measure the radiative heat flux received at the floor level.

NUMERICAL CALCULATIONS

In the simulations with FDS, the computing domain was extended 3.4 m out of the chamber along the x-direction. In this way, free boundary conditions could be specified properly so that the neutral plane height in the doorway could be predicted. The physical properties of the brick wall and cement are quite similar, and were taken as thermal conductivity $1.0\,W\,m^{-1}\,K^{-1}$, density $1700\,kg\,m^{-3}$, specific heat capacity $1000\,J\,kg^{-1}K^{-1}$ and thermal diffusivity $5.8 \times 10^{-7}\,m^2\,s^{-1}$. The physical properties of the ceiling were also taken as constant with a thermal conductivity of $1.6\,W\,m^{-1}\,K^{-1}$, density of $2400\,kg\,m^{-3}$, and specific heat capacity of $920\,J\,kg^{-1}K^{-1}$. The pool fire was approximated by a polygon as in Figure 2, having a roughly similar area to the circular container in the experiments. A three-dimensional Cartesian coordinate system was used

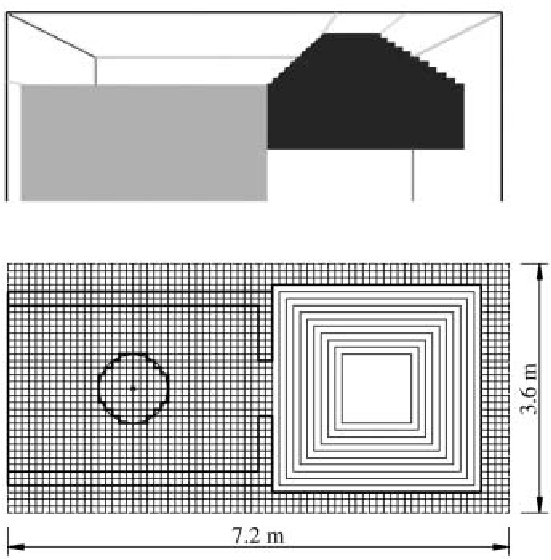


***Figure 2.** Computing domains.*

with length along the x-direction, width along the y-direction, and height along the z-direction as in Figure 2. The exhaust hood was approximated by a series of hollow square walls as in the figure. The computing domains are 72 by 36 by 36, i.e. 93,312 cells, distributed uniformly. The initial air temperature was 15°C.

The Smagorinsky subgrid scale model was used. In this scale, the boundary layer is not so important. There are two coefficients, the Smagorinsky constant C_s of values varying from 0.1 to 0.25; and the subgrid scale turbulent Prandtl number Pr_t varying from 0.2 to 0.9 (e.g., see [14,15]). Values of the Smagorinsky constant C_s would affect the results related to the grid refinement predicting indoor airflow. For coarser grids, a larger value of C_s would give better results. For simulations in this paper, C_s was selected as 0.2. The results are not very sensitive to changes in Pr_t [15] and Pr_t was taken as 0.5 [16] for simulating the indoor airflow.

The heat release rate curve measured in this set of experiments is shown in Figure 3. The peak heat release rate was about 2.6 MW and the burning duration was about 3 min. This curve is taken as the input function to the FDS model.

A fire can be taken as a boundary condition in FDS. It is modeled by ejecting pyrolyzed fuel from a solid surface or vent, which then burns upon mixing with oxygen. The species associated with combustion can be

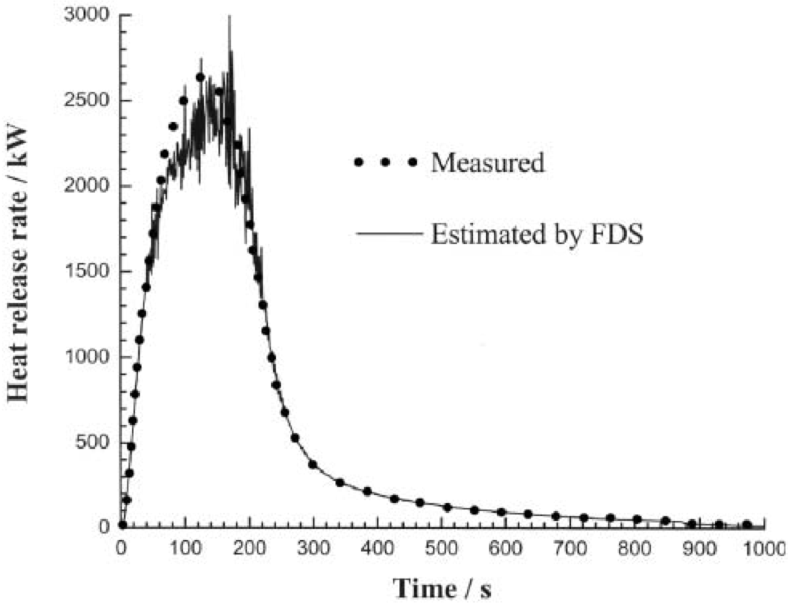


Figure 3. *Heat release rate curve.*

described by the mixture fraction variable. The fire can be described either by the heat release rate per unit area (for a fire of given size) or heat of vaporization at the fuel surface (if the fuel burning rate is estimated from thermal feedback). In this case, the burning rate depends on the thermal feedback from the fire through convection and radiation, and conduction through the solid or liquid fuel. As radiative heat flux from the fire to the fuel surface is difficult to estimate, the heat of vaporization model does not work very well. In this paper, the fire is described by the heat release rate per unit area with the heat release rate curve measured in the experiment taken as an input curve.

Simulations for one set of experiments were carried out on a personal computer with a processor of 1.9 GHz, 512 MB RAM, and hard disk 80 GB. The computing time required was up to 28 h. The transient predicted results can be visualized by the post-processor program Smokeview. The results were saved as data files and analyzed by commercial graphics processors for CFD/NHT.

RESULTS

The fire is described by the heat release rate per unit area. The fuel is injected into the room at a mass flow rate estimated from the measured (input) heat release rate and heat of combustion. The local heat release rate is then computed from the oxygen consumption rate at the flame surface using the oxygen consumption method. The total heat release rate $\dot{Q}$ is estimated by integrating the local heat release rate per unit volume $\dot{q}'''$ over the computational domain Ω as in Equation (1):

$$\dot{Q} = \oiiint_{\Omega} \dot{q}''' dV \tag{1}$$

The total heat release rate computed in this manner is shown in Figure 3. There are slight deviations from the input experimental curve. Perhaps there are limitations on using the mixture fraction combustion model. Note that the fuel and oxygen are assumed to burn instantaneously upon mixing. This is a valid assumption in simulating large-scale well-ventilated fires, but might not hold good for other fires.

The results for the fire environment denoted by the velocity vectors and temperature contours induced by the pool fires across the central y-plane at 4, 10, 100, 140, and 260 s are shown in Figure 4. This set of curves shows the transient development of the flow and temperature field when the heat release rates started to increase upon igniting the gasoline, and then

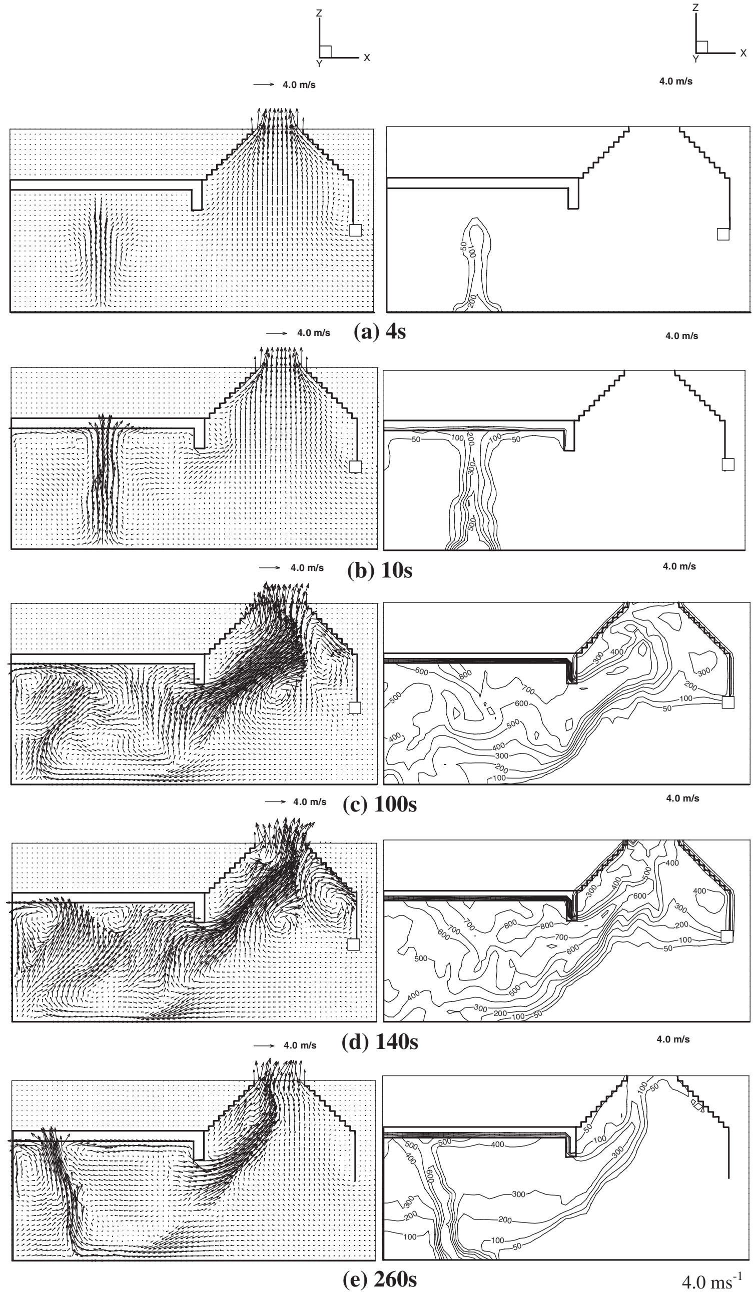


Figure 4. *Predicted fire environment.*

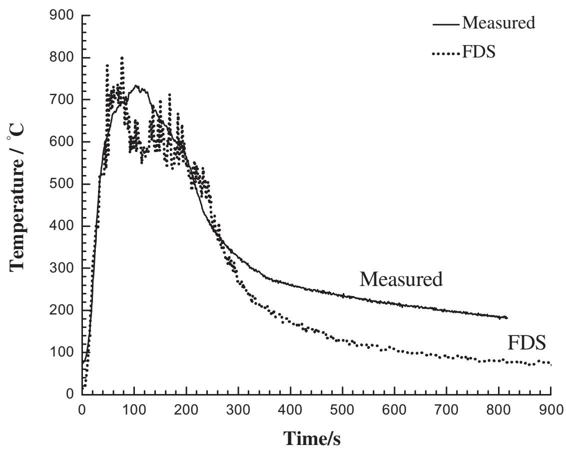
decreased when the fuel was used up. A plume was simulated first as shown by the velocity vectors at 4 s in Figure 4(a). Basically, there are two air circulations in this vertical plane, one between the fire and the wall, and the other between the door and the fire. Later, at 100 s, the circulation between the fire and the wall disappeared and combined with the other one as in Figure 4(c). Air flows in, heats up and rises, then moves out of the room. But when the fuel begins to be used up, the flow structures become quite complicated at 140 and 260 s as seen in Figure 4(d) and (e).

Note that flashover occurred in the experiment. The flame moves out of the door, as observed at the test site. This is also confirmed by numerical experiments as the gas temperature next to the ceiling was higher than 600°C. The thermal radiative heat flux at the floor level, 0.6 m from the door, was also measured in the experiment. The results predicted by FDS are compared with the results of the experiments in Figure 7. The radiative heat flux increased to over 20 kW m^{-2}, suggesting that flashover occurred. The maximum predicted value is 29.3 kW m^{-2}, quite close to the maximum measured value of 29.7 kW m^{-2}. The predicted curve agrees well with the measured curve within a 10% deviation as in Figure 8(b), except during the growth stage.

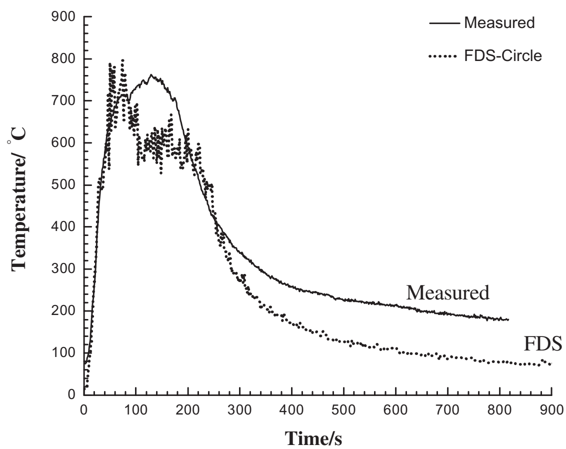
The transient temperatures measured at the seven thermocouples, three isolated at the ceiling, and four hooked up together as a thermocouple tree, are shown in Figures 5 and 6, respectively. The results predicted by FDS are also shown in these figures. The general trends of the measured and predicted temperature–time curves are similar, though deviations of more than 100°C during late stages of the fire are observed. This is still acceptable as the peak temperature might go up to 800°C over a burning duration of about 3 min. However, predicted temperatures do not agree very well with the experiments at the decay stage of the fire. The measured temperature curves appeared to be smooth because of the relatively longer response time constant of the thermocouples. Only the general trend of temperature variation was measured.

During the growth stage of the room fire, temperatures predicted by FDS agree well with measured data at the four thermocouples T_a, T_b, T_c, and T_d placed near the ceiling, as can be seen in Figures 5(a)–(c) and 6(a). At this stage, the percentages of deviations are within 10% as in Figure 8(a). This would also give similar time to flashover if the flashover criterion were judged from the gas temperature adjacent to the ceiling, i.e. 550–600°C. During the decay stage of the fire, deviations of temperature up to 100°C are observed, giving percentages of deviations of 40%. The measured temperatures at T_e, T_f, and T_g are higher than the predicted temperature at points near the ceiling. As the empirical near-wall turbulence model was not used when applying the Smagorinsky subgrid-scale model to simulate the buoyant flow, the predictions deviate more in regions near the floor and the ceiling. This agrees with earlier studies by Emmerich et al. [13] and Zhang et al. [15].

At the growth stage of the fire, the temperatures predicted at lower levels do not agree well with the measured values at thermocouples T_e, T_f, and T_g. The percentage, deviations took values of 30% at 50–90 s. On the other hand, predicted temperatures agree with experiments at the decay stage of the fire. The values of deviations between the measured and predicted values

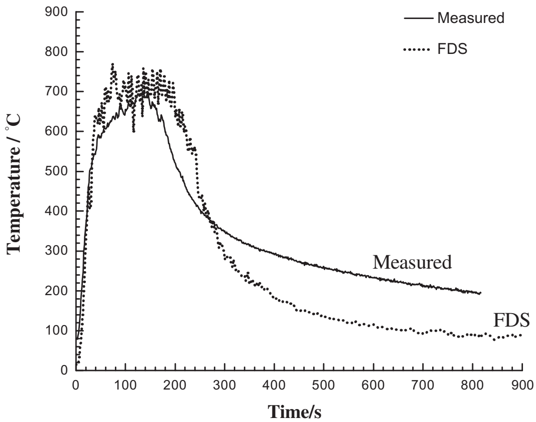


(a) At point T_a



(b) At point T_b

Figure 5. *Thermocouples at the ceiling.*



(c) At point T_c

Figure 5. *Continued.*

are 20–30°C, and the percentage deviations are 14–18%. This is clearly illustrated in Figure 6(b)–(d).

During the decay stage of the fire, the predicted temperatures agree better with the values measured at thermocouples T_e, T_f, and T_g than those at thermocouples T_a, T_b, T_c, and T_d. Greater discrepancies are found near the ceiling.

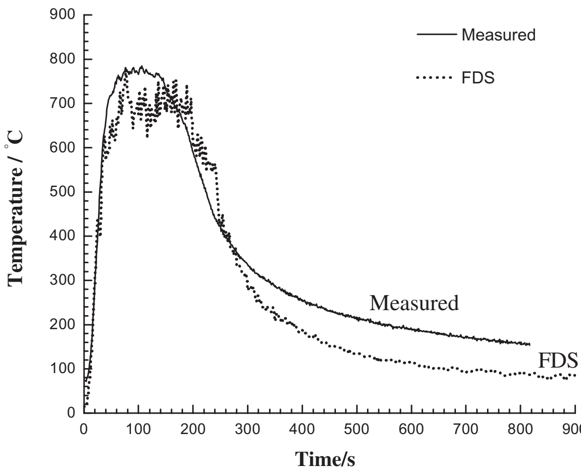
The predicted results on radiative heat flux are shown in Figure 7. The lower radiative heat fluxes were measured from 0 to 40 s, though bigger values were predicted by FDS at the beginning. One possible reason is that the combustion model does not take the ignition temperature into account when fuel and oxygen burn when mixed. Combustion reaction occurred immediately when fuel was injected into the room from the fire boundary. In this way, heat was released into the surroundings. In real incidence, the fire grew slowly at the beginning before having adequate heat to vaporize the liquid fuel. A part of the heat was absorbed by the adjacent fuel and solid. The two curves indicate that flashover occurred, with radiative heat flux of up to 20 kWm^{-2}. During the decay stage of the fire, the two curves almost concide.

The percentages of deviations (PD) for excess temperature and radiative heat flux (denoted by ϕ) between the predicted value ϕ_p and the measured value ϕ_m were calculated to quantify the deviations,

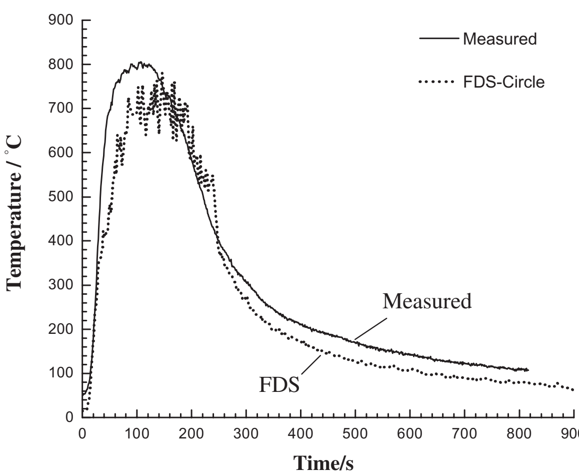
$$PD = \frac{\phi_p - \phi_m}{\phi_m} \times 100\% \tag{2}$$

Note that the excess of temperature above the ambient value is approximated here by the temperature in degrees Centigrade.

The values of PD for temperatures at T_a–T_g varied from −20 to +20%; and radiative heat flux from −10 to +10%. Typical results for T_a and radiative heat flux are shown in Figure 8.

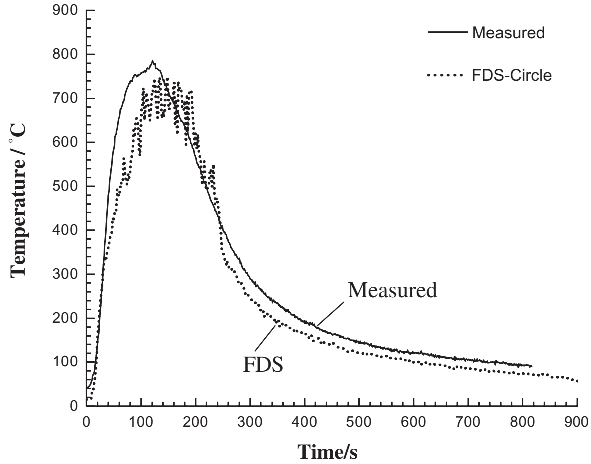


(a) At point T_d

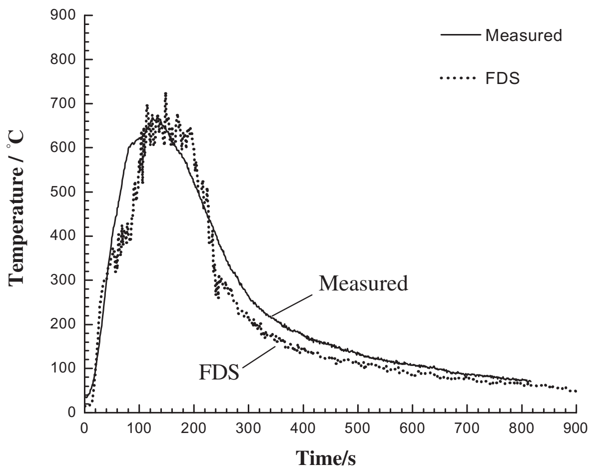


(b) At point T_e

Figure 6. *The thermocouple tree.*



(c) At point T_f



(d) At point T_g

Figure 6. *Continued.*

DISCUSSION

Based on this study of room fires that include the flashover phenomenon with burning liquid pools, it is observed that the model FDS version 3.01 can give good predictions on the fire-induced temperature fields for this specific scenario. Airflow could not be measured. As observed in the tests, the flame filled the entire room and then emerged from the doorway. There

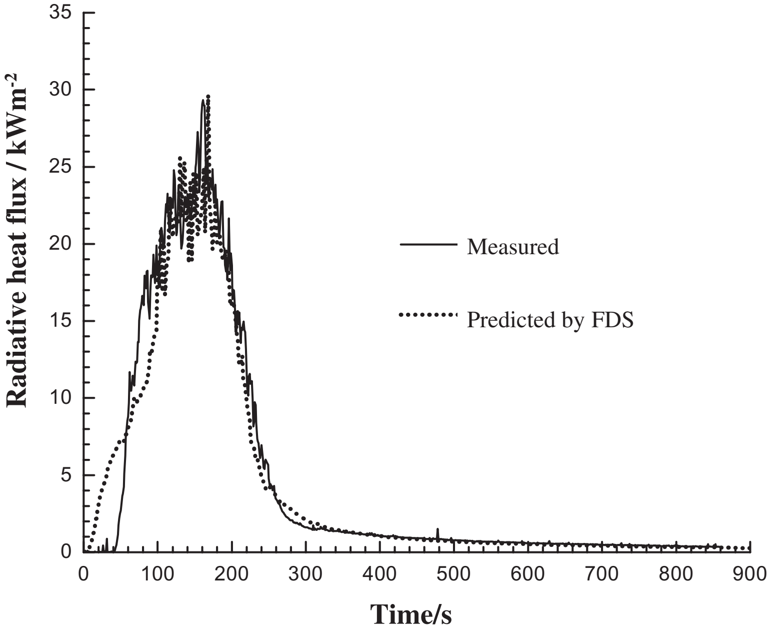
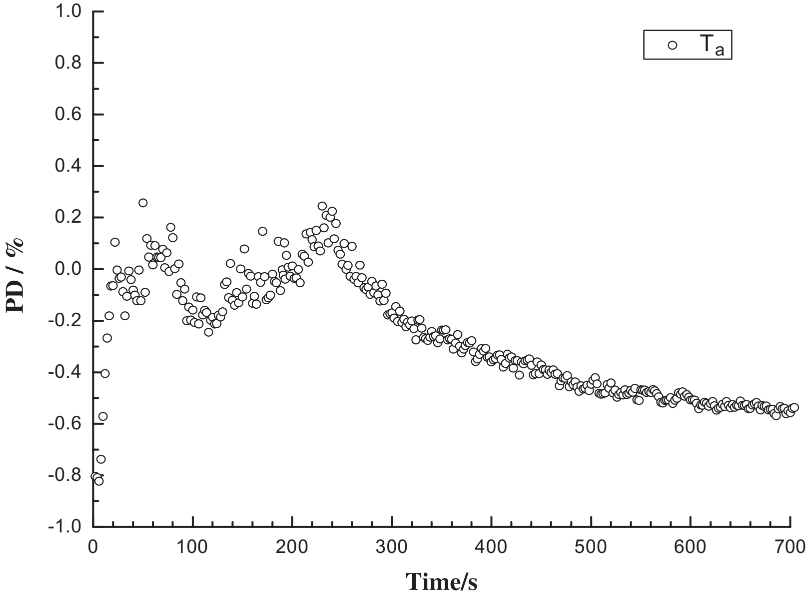


Figure 7. *Radiative heat flux.*

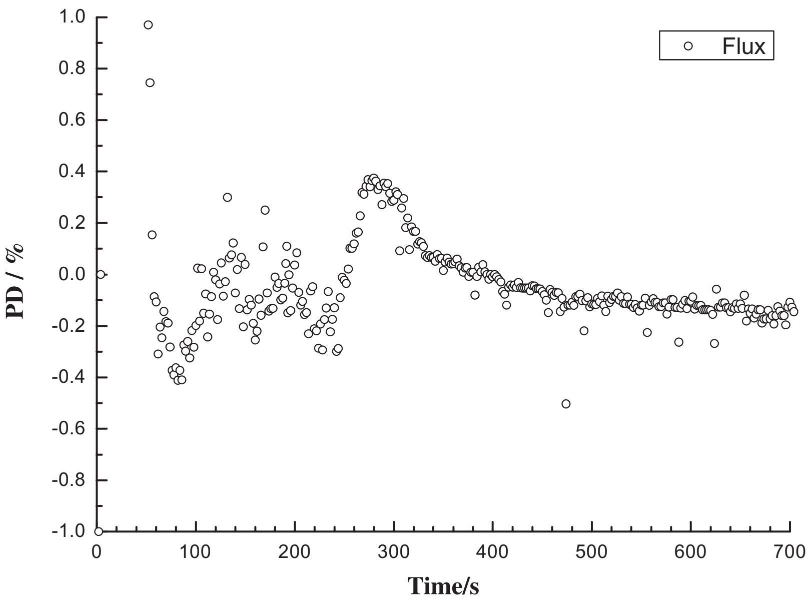
were concerns about simulating the longer chemical times with a large eddy simulation, but no problems were encountered in the simulations. Results for the predicted temperature–time curves agree fairly well with the experiments. Although deviations from measured data at ceiling levels and lower levels are not quite the same at different stages of the room fires, the general trend of the time history of temperature is still acceptable.

CONCLUSIONS

Figure Dynamics Simulator was applied to simulate flashover fires and the results are compared with those obtained from full-scale burning tests. The experimental curve on heat release rate was taken as an input function. The predicted temperature and radiative heat flux agree reasonably well with the experiments reported in [12]. The temperatures predicted near the ceiling agree within a 10% deviation from the measured data during the growth stage of the room fire. This is very useful for judging whether flashover has occurred. The predicted results on radiative heat flux are acceptable for most engineering applications. This demonstrates that FDS can be used to simulate this specific type of flashover fire.



(a) Temperature at point T_a



(b) Radiative heat flux

Figure 8. *Percentage deviation of temperature and radiative heat flux.*

To summarize, it appears that FDS gives quite good predictions, and may be suitable for use as a practical tool in fire safety designs and other engineering applications.

ACKNOWLEDGMENT

This paper is partly supported by the PolyU president under research account A-078 for Areas of Strength in Fire Safety Engineering; it is also partly supported by the Research Centre for Fire Engineering, and by the departmental earnings account, Department of Building Services Engineering, which is made up of funds earned from the running of the self-financed MSc degree programme in Fire and Safety Engineering.

REFERENCES

1. Cox, G., "Combustion Fundamentals of Fires," Academic Press, London, 1995.
2. McGrattan, K.B., Baum, H.R., Rehm, R.G., Hammins, A., Forney, G.P., Floyd, J.E., Hostikka, S. and Prasad, K., "Fire Dynamics Simulator (Version 3) – Technical Reference Guide," NISTIR 6783, National Institute of Standards and Technology, US Department of Commerce, November 2001.
3. McGrattan, K.B., Forney, G.P., Floyd, J.E., Hostikka, S. and Prasad, K., "Fire Dynamics Simulator (Version 3) – User's guide," NISTIR 6784, National Institute of Standards and Technology, US Department of Commerce, November 2002.
4. McGrattan, K.B., Baum, H.R. and Rehm, R.G., "Large Eddy Simulations of Smoke Movement," Fire Safety Journal, 1998, Vol. 30, pp. 161–178.
5. Baum, H.R., "Large Eddy Simulations of Fires – from Concepts to Computations," Fire Protection Engineering, 2000, No. 6, pp. 36–38, 40, 42.
6. Nam, S. and Bill Jr., R.G., "Numerical Simulation of Thermal Plumes," Fire Safety Journal, 1993, Vol. 21, No. 3, pp. 231–256.
7. Chow, W.K. and Leung, W.M., "Solid-wall Boundary Effect on a Building Fire Field Model," Combustion Science and Technology, 1990, Vol. 71, No. 1–3, pp. 77–93.
8. Steckler, K.D., Quintiere, J.Q. and Rinkinen, W.D. "Flow Induced by a Fire in a Compartment," 19th International Symposium on Combustion, The Combustion Institute, Pittsburgh, 1982, pp. 913–920.
9. Steckler, K.D., Baum, H.R. and Quintiere, J.Q., "Fire Induced Flows Through Room Openings – Flow Coefficients," 20th International Symposium on Combustion, The Combustion Institute, Pittsburgh, 1984, pp. 1591–1600.
10. Chow, W.K., "Support on Carrying Out Full-scale Burning Tests for Karaokes," International Journal on Engineering Performance-Based Fire Codes, 2001, Vol. 3, No. 3, pp. 104–112.
11. Chow, W.K., Gao, Y., Dong, H., Zou, G. and Meng, L., "Will Water Mist Extinguish a Liquid Fire Rapidly," Architectural Science Review, 2003, Vol. 46, No. 2, pp. 139–144.
12. Chow, W.K., Gao, Y., Dong, H., Zou, G., Luo, Z. and Meng, L., "Experimental Studies on Minimum Heat Release Rates for Flashover with Oxygen Consumption Calorimetry," Architectural Science Review, 2003, Vol. 46, No. 3, pp. 291–296.

13. ASHRAE Handbook – Fundamentals, American Society of Heating, Refrigerating and Air-Conditioning Engineers (ASHRAE), Atlanta, Ga. 2001.
14. Emmerich, S.J. and McGrattan, K.B., "Application of a Large Eddy Simulation Model to Study Room Airflow," ASHRAE Transactions, 1998, Vol. 104, pp. 1128–1137.
15. Zhang, W., Hamer, A., Klassen, M., Carpenter, D. and Roby, R., "Turbulence Statistics in a Fire Room Model by Large Eddy Simulation," Fire Safety Journal, 2002, Vol. 37, pp. 721–752.
16. Murakami, S., "Overview of Turbulence Models Applied in CWE-1997," Journal of Wind Engineering and Industrial Aerodynamics, 1998, Vol. 74–76, pp. 1–24.