J. of Fire Prot. Engr., 6 (4), 1994, pp. 137–152

A CRITICAL COMPARISON OF A PHOENICS BASED FIRE FIELD MODEL WITH EXPERIMENTAL COMPARTMENT FIRE DATA

R.N. Mawhinney, E.R. Galea*, N. Hoffmann, and M.K. Patel
Fire Safety Engineering Group
Centre For Numerical Modelling and Process Analysis
The University of Greenwich
London SE18 6PF
U.K.

SUMMARY

This paper describes the application of a fire field model based on the PHOENCIS CFD software to the simulation of fire induced flows in domestic-sized rooms. Several scenarios are examined consisting of various fire sizes, fire locations, and door sizes. Comparisons are based on upper-layer room temperatures, mass fluxes in and out of the fire compartment, and door-way vertical and horizontal temperature and velocity profiles. For most cases, the model agrees reasonably well with the observed trends. However, significant smearing is observed in predicted temperature profiles in the vicinity of the confining walls. A close examination of the horizontal doorway velocity profiles highlights the need for careful modelling and experimental practices.

INTRODUCTION

The use of mathematical models for the simulation of fire phenomena within compartments dates back some 30 years[1], to a time before the common use of electronic computers. These early models were the precursors of the now commonly accepted zone models[2]. The past 30 years of zone model evolution has seen considerable development in their sophistication and the establishment of a convincing battery of comparisons between model predictions and experimental data.

The systematic comparison of prediction with experiment (commonly termed "validation") is a crucial step in the general acceptance of model predications and in determining the scope of their application. While no degree of successful validation will prove a fire model correct, confidence in the technique is established the more frequently it is shown to be successful in as wide a range of applications as possible.

In contrast to zone models, fire field models[2] have been under development for only about 15 years[3,4] and consequently, considerably less "validation" has been completed. However, the "validation" of fire field model predictions is essential to the continued development and acceptance of this complex procedure.

Furthermore, as the number and variety of fire field models increases[3-11], it becomes necessary to provide a discriminating basis for comparison. Success at a wide range of standard "validation" exercises provides one means to this end. However, little effort has been invested in the systematic comparison of various fire models with common experimental data.

This is due for the most part to the lack of suitable experimental benchmark fire data. The majority of fire experiments are not conducted for model validation purposes, and a significant number of those that are, are specifically designed for the "validation" of the less demanding zone models.

* Author for correspondence.

In most of these cases, insufficient data is recorded to allow a detailed "validation" of field models.

The "validation" process is a non-trivial task as even the simplest steady-state field model typically involves in excess of 40,000 degrees of freedom with many more for transient models. A thorough validation would require each degree of freedom to be compared with a corresponding measurement. Clearly, this is not practical and so in practice any validation exercise involves a series of compromises.

The authors have undertaken a detailed comparison of two CFD codes commonly used for fire field modelling—FLOW3D[12] and PHOENICS[13]—with experimental data. In this paper a comparison is made of field model predications based on PHOENICS with experimental data. The ability of the code to model a series of room fire experiments[14] is examined and a further comparison is made with earlier studies produced using JASMINE,[11] and the Hadjisophocleous and Cacambouras code[10]. The results of the FLOW3D prediction are the subject of another publication[15].

The Experiments

A series of 45 experiments were conducted by Steckler *et.al.* [14] to investigate fire induced flows in a compartment measuring 2.8 m × 2.8 m in plane and 2.18 m in height. The walls were 0.1 m thick, and the walls and ceiling were covered with a lightweight ceramic fibre insulation board. The series of experiments consisted of a gas burner placed systematically in eight different floor locations with a variety of single compartment openings ranging from small windows to wide doors. The 0.3 m diameter burner was supplied with commercial grade methane at a fixed rate producing constant fire strengths of 31.6, 62.9, 105.3, and 158 kW. Near steady-state conditions were achieved within 30 minutes.

An array of bi-directional velocity probes and bare-wire thermocouples was placed within the room opening to measure velocities and temperatures within the centre of the door jamb. In addition, a stack of aspirated thermocouples was placed in the front corner of the room to measure the gas temperature profile.

The Numerical Simulations

A number of simulations were performed based on three fire locations (Figure 1), three door widths (0.24, 0.74, and 0.99 m), and two fire sizes (31.6 and 62.9 kW). The door openings used in these simulations measured 1.83 m in height. The simulations were performed using the PHOENICS (Version 1.6) software.

The starting point of the analysis is the set of three-dimensional, partial differential equations that govern the phenomena of interest here. This set consists, in general, of the following equations: the continuity equation; the three momentum equations that govern the conservation of momentum per unit mass in each of the three space dimensions; the equation for conservation of energy; and, the equations for a turbulence model (in this case, the k-epsilon model with buoyancy modifications). Compressibility is assumed, and the perfect gas law is used to describe the equation of state. The precise formulation of

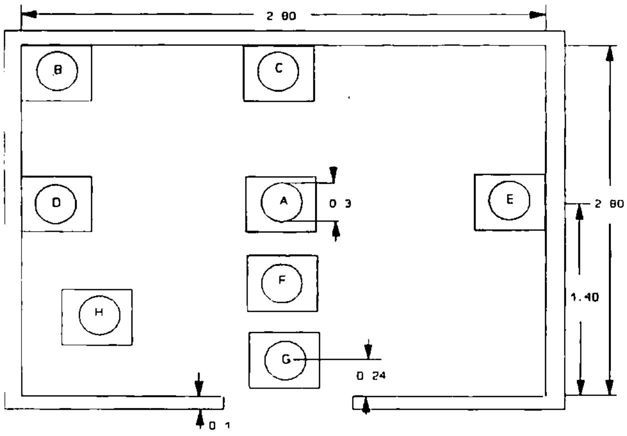


Figure 1. The eight fire locations studied in the experiments. Fires in locations A, C and D were simulated.

the differential equations describing the model will not be given here as they may be found elsewhere[1,12,13,16]. Combustion and radiation are ignored in these simulations.

The initial temperature was set to the measured ambient value while the walls of the compartment were modelled with no-slip conditions for the velocities and adiabatic conditions for the temperature. The usual "wall functions"[17] were used to compute shear stresses at the wall. In order to correctly model the flow through the open door, the numerical grid was extended by 1.4 m to include a region outside the fire compartment. A fixed pressure boundary condition was used on all external boundaries.

As adiabatic wall boundary conditions were used in the model, the quoted experimental heat source was not used in the simulations. In order to account for heat lost to the walls during each experiment, the heat convected out of the room was determined and used as the prescribed heat source. In the majority of cases examined, the heat loss to the walls was negligible, accounting for approximately 3 kW. The maximum value of 14 kW was achieved for the fire located in the room corner with the narrow door opening. In an earlier reported account of the centre fire simulations[18], the heat source was not adjusted sufficiently to take full account of this heat loss.

A mesh of 8280 cells in total was used to discritise the geometry (6480 internal and 1800 external cells). The mesh consisted of 23 cells in length, 20 cells in width and 18 cells in height for all cases examined. The mesh was non-uniformly distributed with refinements in the wall, floor, ceiling, fire and doorway regions. Only minor modifications to the mesh distribution were used for the various fire locations.

The PHOENICS code was run in steady-state mode, typically requiring 1500 sweeps to achieve convergence. This was deemed to be attained when mass and enthalpy balance into and out of the computational domain and mass balance through the door were all achieved to within 1 percent, residuals fell to less than 1 percent of average variable values, and spot values differed by less than 0.1 percent over the last 100 sweeps. Using a 40 Mhz SUN SPARC 10 workstation—rated at 13.7 MFLOPS on the standard LINPACK benchmark—a steady-state simulation required 1.63 hrs. of CPU.

Results and Discussion

Accuracy of Model Predictions

In comparing numerical predictions with experimental results, we are primarily concerned with their overall level of agreement and in ascertaining whether the model is capable of predicting the observed trends. Table 1 summarises the numerical results while Figure 2 graphically depicts the level of agreement between measured and predicted results. Figures 3 to 9 depict comparisons of model predictions with the observed trends.

The upper layer temperature was determined experimentally by averaging the temperature values in a column through the upper layer as measured by the thermocouple stack located in the corner of the room. The predicted values represent a mean temperature determined in a similar manner.

Predictions of average upper layer temperature (Figure 2a) are in good agreement with measured results for most of the cases studied. The cases where the fire was located adjacent to the rear wall display the worst correlation, with the predicted values being as much as 44 percent greater than the measured values.

The position of the neutral plane is determined by calculating the approximate location of the zero velocity line within the doorway. As such, it is subject to errors in the measured (and predicted) velocity profile. For the measured values these are of the order of 10 percent, bringing all of the predictions within the range of experimental errors (Figure 2b).

TABLE 1: Comparison of PHOENICS predictions of Neutral Plane Height/Door Height (N/H_o), Mass Flow Rates and Average Upper Layer Temperature with experimental results.

Scenario	N/H_o	Mass Flow Rate kg/sec		Average Hot Layer Temp(C)
		IN	OUT	
A, 0.24m, 62.9kW				
Numerical	0.484	0.282	0.282	210
Experimental	0.499	0.255	0.247	190
A, 0.74m, 62.9kW				
Numerical	0.504	0.587	0.587	133
Experimental	0.561	0.554	0.571	129
A, 0.99m, 62.9kW				
Numerical	0.512	0.698	0.698	111
Experimental	0.582	0.653	0.701	109
A, 0.74m, 31.6kW				
Numerical	0.511	0.500	0.500	96
Experimental	0.569	0.430	0.461	86
C, 0.24m, 62.9kW				
Numerical	0.508	0.236	0.236	301
Experimental	0.528	0.238	0.248	209
C, 0.74m, 62.9kW				
Numerical	0.596	0.398	0.398	201
Experimental	0.579	0.479	0.498	152
C, 0.99m, 62.9kW				
Numerical	0.614	0.445	0.445	188
Experimental	0.593	0.576	0.593	134
C, 0.74m, 31.6kW				
Numerical	0.604	0.350	0.350	126
Experimental	0.613	0.387	0.396	94
D, 0.74m, 62.9kW				
Numerical	0.558	0.487	0.486	155
Experimental	0.565	0.470	0.486	153

The measured mass fluxes into and out of the room are subject to errors of 10–13 percent. Predictions of this quantity for fires located away from the rear wall show good agreement with experiment; however for fires located adjacent to the rear wall differences of up to 25 percent occur (Figure 2c). This trend was also observed in the JASMINE predictions[11] albeit with a slightly coarser mesh (6270 cells).

Differences Between Model Predictions and Observations

There are several possible explanations which may account for these differences relating to the nature of the experiments and the modelling. The mass flow measurements (and predictions) rely on accurate velocity measurements throughout the area of the open doorway. In order to achieve meaningful results, the bidirectional ve-

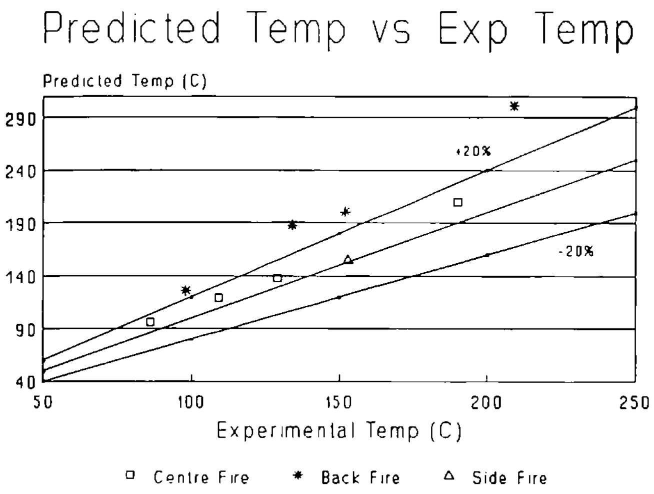


Figure 2a. Calculated and measured upper layer temperature (°C) for centre, back and side positioned fires. Model predictions use adiabatic boundary conditions.

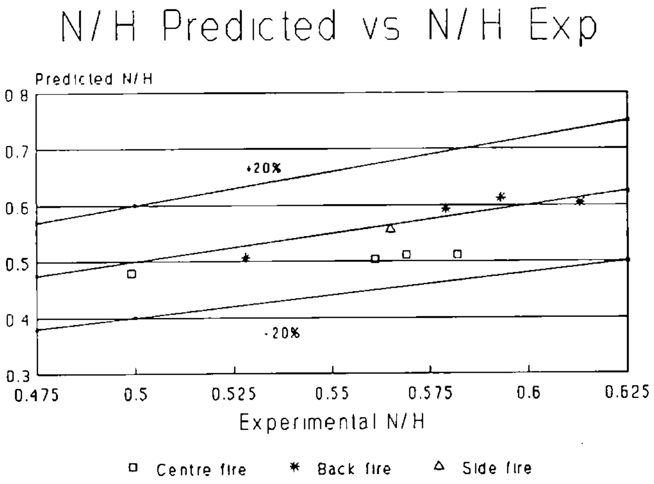


Figure 2b. Calculated and measured neutral plane height/door height for centre, back and side positioned fires. Model predictions use adiabatic boundary conditions.

locity probes must be aligned parallel to the velocity streamlines. The velocity probes in the experiment were placed with their axes horizontal (*i.e.*, parallel to the floor). However, in the vicinity of the door edges —particularly the top edge where the hot buoyant gases exit the room—the streamlines intercept the door plane at acute angles, leading to the possibility of substantial measurement errors in the velocity and hence mass flux.

The burner was modelled as a heat source of the appropriate heat release rate which was released entirely within the confines of the burner volume. Furthermore, the burner was considered to be square in section with a volume identical to the circular burners used in the experiment. For the fires located well away from the walls, this appears to be a reasonable approximation. However, in the experiment the circular burners located in the corner and in the middle of the back wall were separated from the walls by at least 0.06 m, this being the size of the lip around the burner perimeter. In the model, the burner was located immediately adjacent to the walls. Williamson, *et.al.* [19], noted that for 150 kW corner fire tests, small wall stand off distances (0.05 m) resulted in a reduction of average ceiling layer temperatures

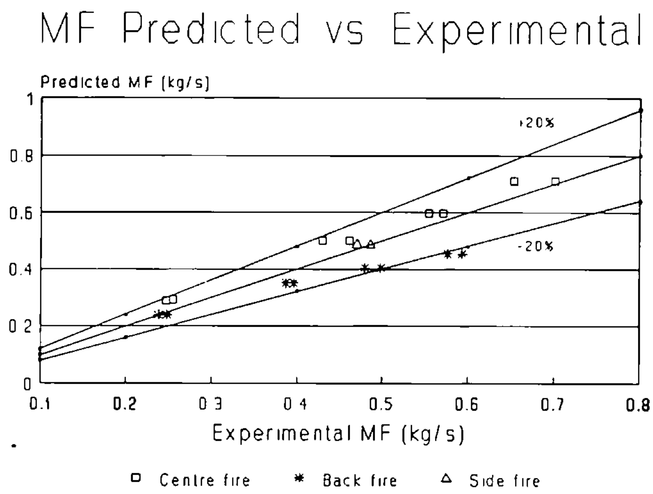


Figure 2c. Calculated and measured mass flux in and out (kg/s) for centre, back and side positioned fires. Model predictions use adiabatic boundary conditions.

of approximately 40°C (approximately 16 percent) compared to a zero stand off distance. These differences in temperature may also influence the mass entrainment into and out of the room.

The wall boundary conditions are also expected to have a greater influence in the cases where the burner is located adjacent to the confining walls. In these cases, the simplistic treatment of the walls as perfectly insulating boundaries may account for the significantly higher temperatures predicted. Clearly, the nature of the heat loss to the wall plays a significant role in these cases, and a full treatment utilising wall material properties is required.

The exclusion of radiation from the model description is expected to have an influence on the predicted results. The JASMINE[11] model included a description of radiation for a selection of the simulations. In those simulations, the predicted mass flux more closely matched the measured mass fluxes for the cases where the fire was located adjacent to the walls; an unchanged or worse agreement was obtained for those cases where the fire was located away from the walls.

In both the experiment and model predictions, we find that for those cases in which the fire is located adjacent to the walls, the average upper layer temperature is higher than in equivalent situations where the fire is located away from the walls. In these situations, we expect the effects of radiation to be more important and hence the exclusion of radiation from these models to be of greater significance.

The representation of the gas burner by a prescribed heat source rather than by a combustion model appears to be a reasonable approximation in this application. In comparing mass flux results for a similar model which excluded radiation effects but included the effects of a simple one-step chemical reaction combustion model[11] there appeared to be little difference. This may be expected since the generation of combustion products is not of major importance in this example.

Trends in Model Predictions

The overall trends observed in the experiment have been captured by the fire field model (see Table 1). As the door width is increased from 0.24 m to 0.99 m, the height of the neutral plane is observed to increase (Figure 3a), the temperature of the hot layer to decrease (Figure 3b), and the mass flux in and out of the room to increase (Figures 3c and 3d). Furthermore, as the fire strength is increased from 31.6 to 62.9 kW, the height of the neutral plane decreases (Figure 4a), while the temperature of the hot layer (Figure 4b) and the mass flux in and out (Figures 4c and 4d) increase.

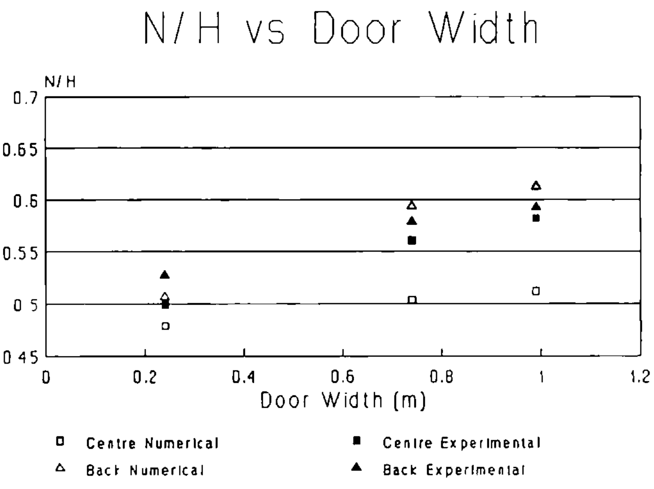


Figure 3a. Calculated and measured neutral plane height/ door height as a function of door width (m) for centre and back positioned 62.9 kW fires.

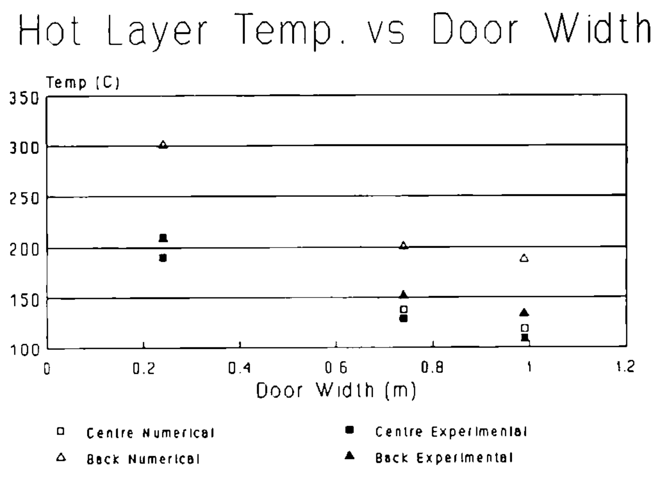


Figure 3b. Calculated and measured hot layer temperature (°C) as a function of door width (m) for centre and back positioned 62.9 kW fires.

J. of Fire Prot. Engr., 6 (4), 1994, pp. 137–152

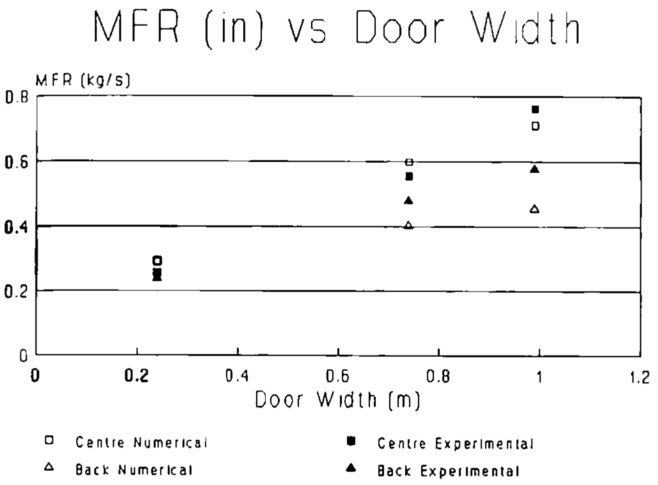


Figure 3c. Calculated and measured mass flux in (kg/s) as a function of door width (m) for centre and back positioned 62.9 kW fires.

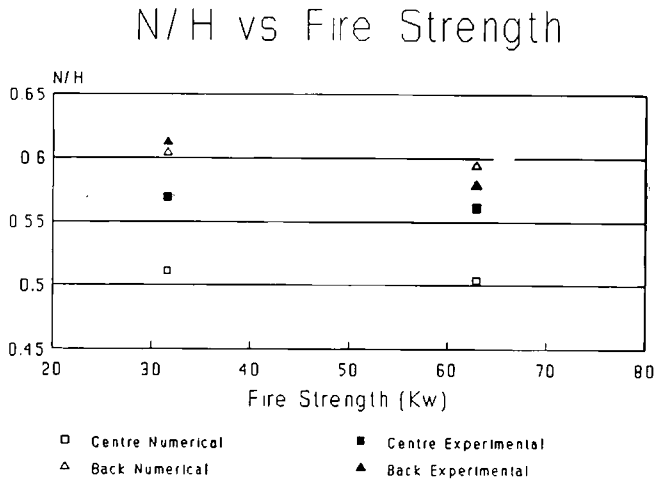


Figure 4a. Calculated and measured neutral plane height/ door height as a function of fire size (kW) for centre and back positioned fires with 0.74 m wide door.

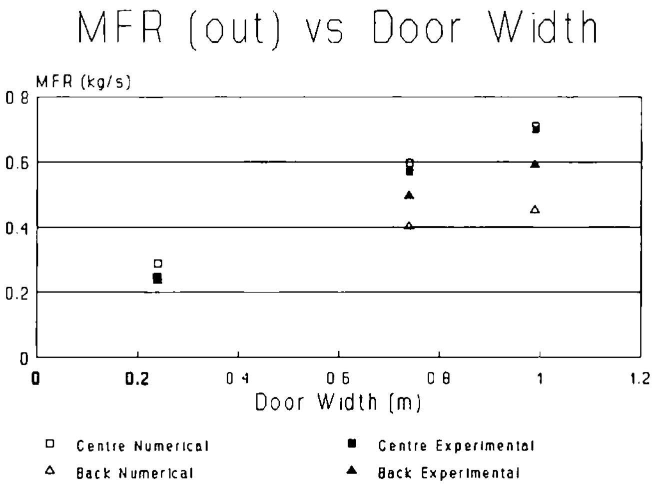


Figure 3d. Calculated and measured mass flux out (kg/s) as a function of door width (m) for centre and back positioned 62.9 kW fires.

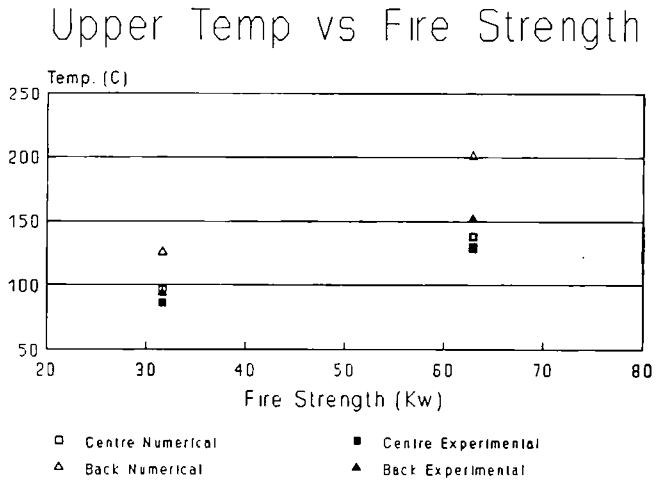


Figure 4b. Calculated and measured upper layer temperature (°C) as a function of fire size (kW) for centre, and back positioned fires with 0.74 m wide door.

Predictions of doorway centreline velocities as a function of door width are depicted in Figures 5a and 6a for the centre and rear fires, respectively. In both cases, model predictions produce good agreement with experimental data; however they both tend to under-predict velocities in the upper most portion of the door. This behaviour was also observed in the JASMINE predictions with and without radiation.

The accuracy of the experimental results in this region is questionable. As discussed above, the velocity measurements in the vicinity of the door edges are prone to errors if the bidirectional probes are not aligned parallel to the flow.

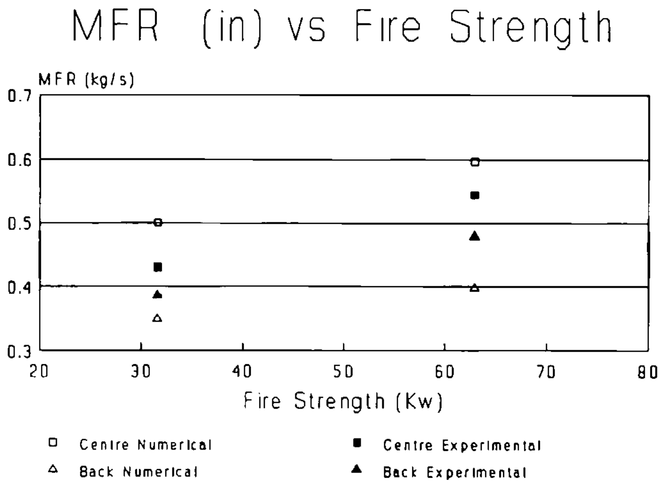


Figure 4c. Calculated and measured mass flux in (kg/s) as a function of fire size (kW) for centre and back positioned fires with 0.74 m wide door.

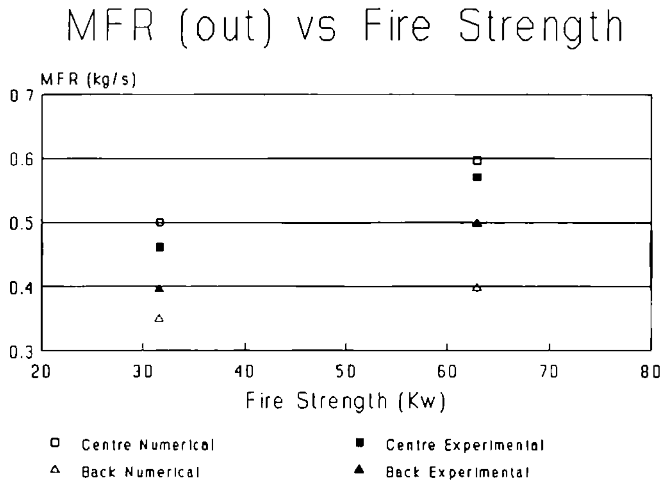


Figure 4d. Calculated and measured mass flux out (kg/s) as a function of fire size (kW) for centre and back positioned fires with 0.74 m wide door.

Predictions of doorway centreline temperatures as a function of door width are depicted in Figures 5b and 6b for the centre and rear fires, respectively. In both cases, model predictions produce reasonable agreement with experimental data, and are capable of predicting the observed trends as door widths decrease. For the two widest doors, the model overpredicts the upper layer temperatures for the rear fire (Figure 6b), while more closely predicting those for the centre fire case. This is consistent with the above discussion.

For the rear fire, the narrow door case produced the poorest levels of agreement. While the overall predicted temperature profile matches that of the experiment, temperatures are greatly overpredicted in the hot layer (Figure 6b). This could be due to poor mesh refinement (as only 2 cells were used across the width of the door, compared with 8 cells in the wide door case).

The poorest level of overall agreement was observed for the corner stack temperatures (Figures 5c and 6c). Thus, for the middle fire, the model appears to predict the upper layer temperature with a good level of accuracy (less than 11 percent) while for the rear fire, upper layer temperatures are overpredicted by as much as 44 percent. Further, the model smears the transition between the hot and cold layers in the room. For the centre fire case (Figure 5c), this results in a failure to predict the existence of a distinct cold layer. For the rear fire case, (Figure 6c) the smearing is less pronounced, and the model predicts the formation of a cold layer (albeit with a smaller depth than that observed in the experiment).

Predictions of doorway centreline velocities and temperatures as a function of fire location for the 0.74m door are depicted in Figures 7a and 7b respectively. Model predictions of doorway velocity (Figure 7a) are in good agreement with experimental data. However, as noted above, the velocity near the top of the door is underpredicted for the centre, back and side located fires. Near the top of the doorway, the measured velocity for all three cases tends to converge to a single common value. This is also observed in the predicted profiles. The closest level of overall agreement between measured and predicted velocity profiles occurs for the side located fire.

Figure 5. Predicted and measured (a) door centre vertical velocity profiles, (b) door centre vertical temperature profiles and (c) corner stack vertical temperature profiles for the 0.24 m, 0.74 m and 0.99 m wide doors. In all cases, the 62.9 kW centrally located fire was modelled.

Figure 5a.

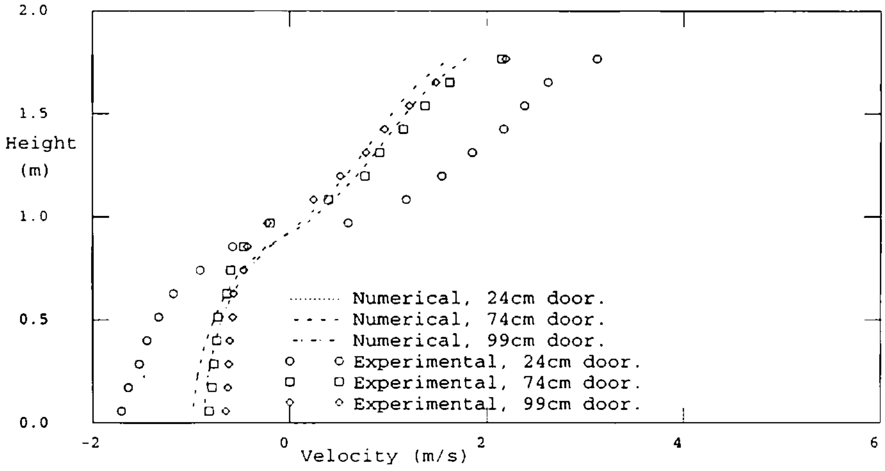


Figure 5b.

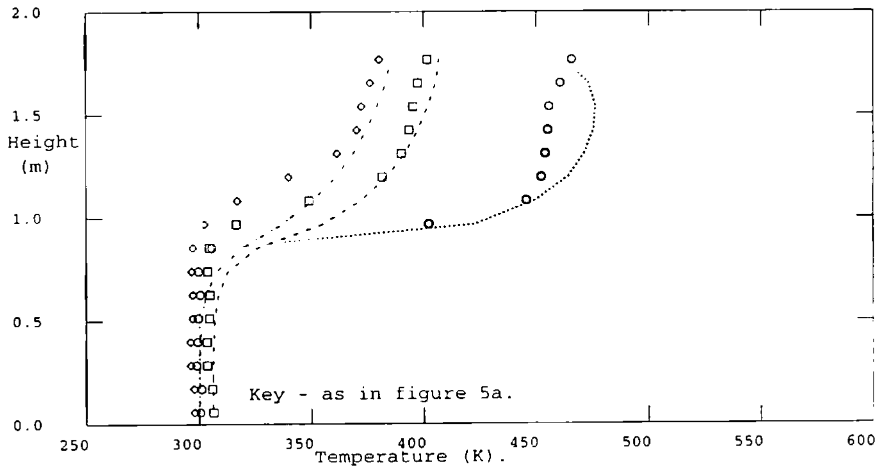


Figure 5c.

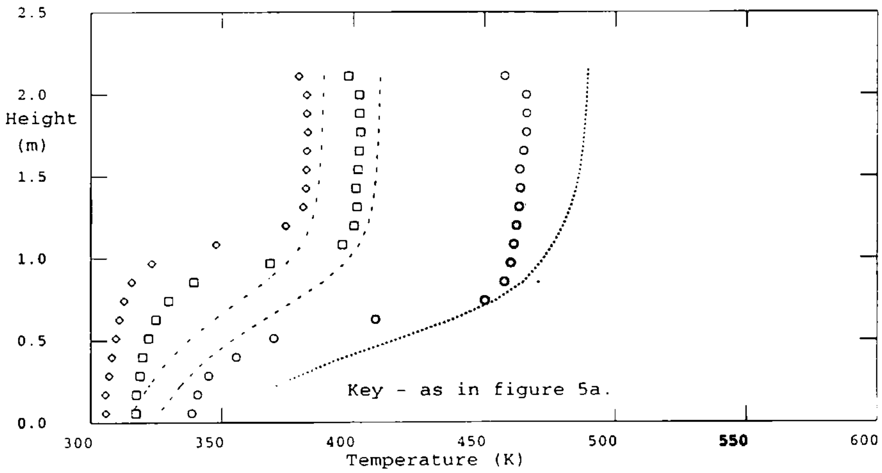


Figure 6. Predicted and measured (a) door centre vertical velocity profiles, (b) door centre vertical temperature profiles and (c) corner stack vertical temperature profiles for the 0.24m, 0.74m and 0.99m wide doors. In all cases, the 62.9 kW fire positioned ajacent to the rear wall was modelled.

Figure 6a.

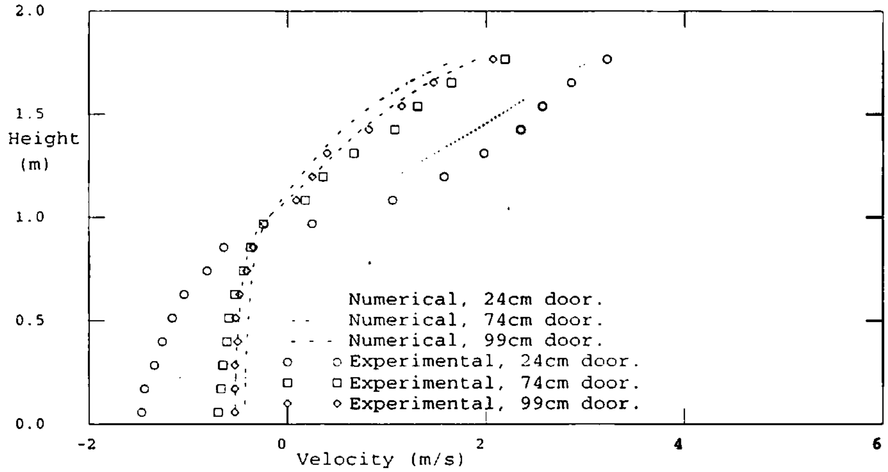


Figure 6b.

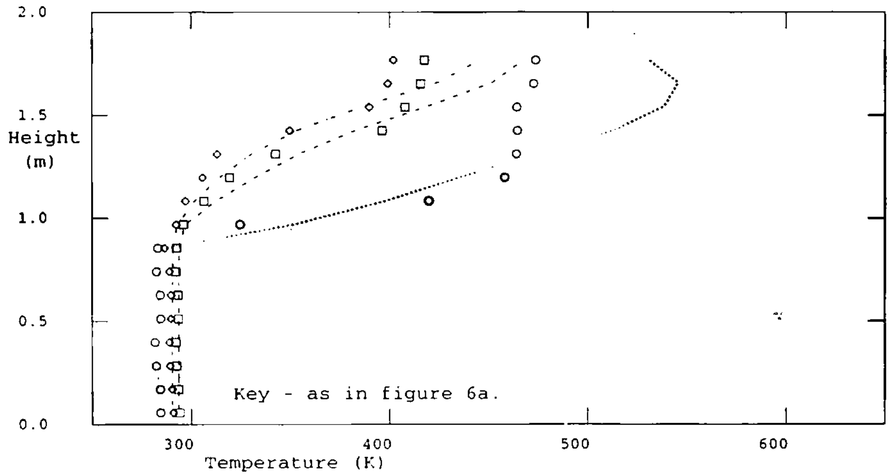


Figure 6c.

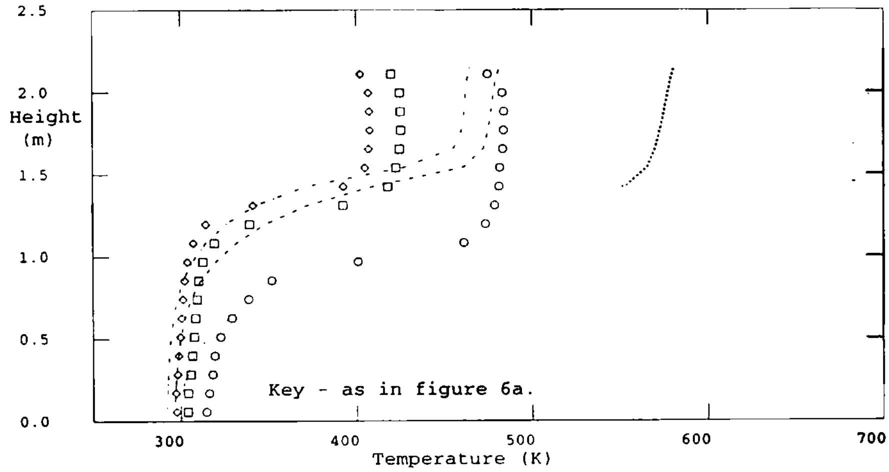


Figure 7. Predicted and measured (a) door centre vertical velocity profiles and (b) door centre vertical temperature profiles for the 0.74 m wide door for the centre, rear wall and side wall located 62.9 kW fires.

Figure 7a.

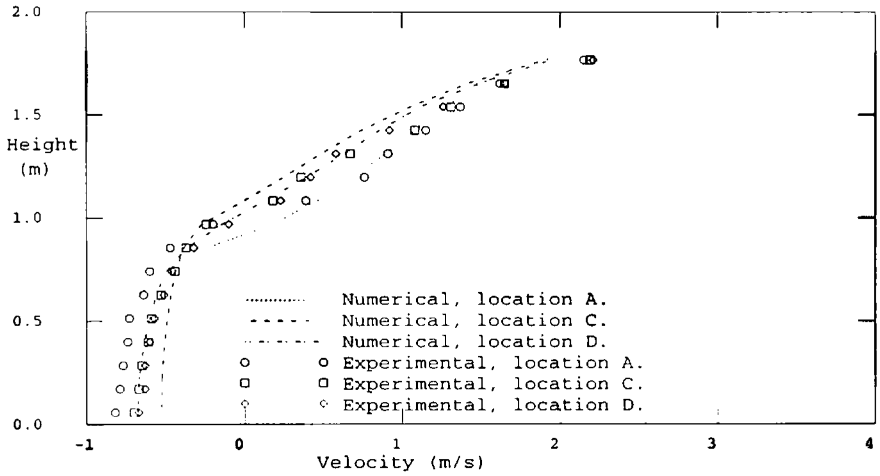
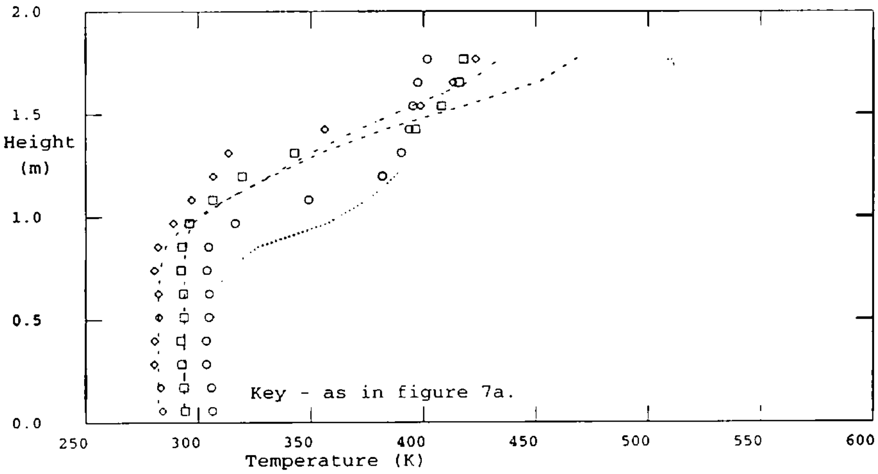


Figure 7b.



Model predictions of doorway centreline temperatures (Figure 7b) are in good agreement with measured data. However temperatures in the upper region of the door are overpredicted in all three cases. The measured temperature profiles suggest that while the side and rear located fires produce a similar temperature distribution within the doorway, this is different to that observed for the centre fire location. These observations are also reflected in the predicted temperature profiles.

Horizontal Velocity Distribution

A significant observation to emerge from the experimental work concerned the nature of the horizontal velocity distribution within the door jamb. In most of the cases, the velocity profile was observed to peak at the door sides with a minimum in the middle. Steckler *et.al.*[20] have argued that these results are consistent with potential flow theory, if the flow is assumed to be irrotational and inviscid. However, in their JASMINE simulation, Kumar *et.al.*[11] noted completely the opposite result, *i.e.*, peak velocities were predicted in the centre of the door. Similar trends were also noted by Hadjisophocleous *et.al.*[10]. In the JASMINE simulation, the door wall appears to have been modelled through the use of a thin surface.

Using this approach, the high faces of the last cells within the room are closed off, thereby defining a soffit and door jamb of zero thickness. As PHOENICS uses a staggered velocity mesh, velocity vectors are located at the cell faces in line with the zero thickness soffit (see Figure 8a). However, the pressures which are used to determine the velocity are obtained from the neighbouring cell centres located just outside and inside the fire compartment. In most applications, this is a fair approximation. However, if detailed comparisons with experimental data derived from within the door jamb are required, then this may prove too crude an approximation.

To gauge the significance of this approximation, a series of simulations was performed using the PHOENICS code in which the soffit was modelled as a thin surface of zero thickness, a single cell 0.1 m thick, and two cells measuring 0.05 m each. The case presented here relates to the 31.6 kW central fire with 0.74 m door.

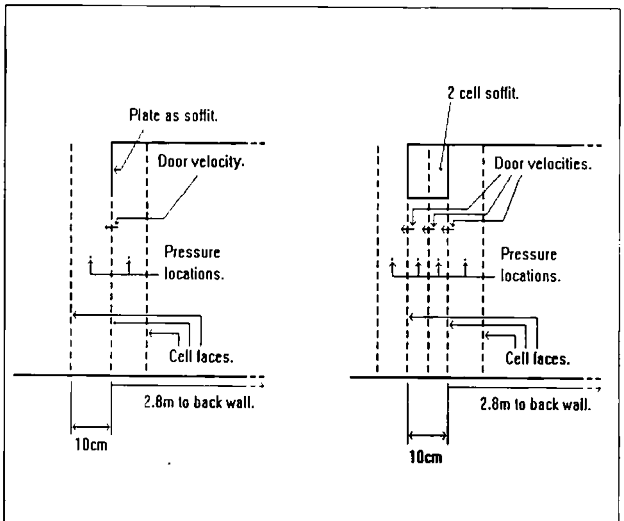


Figure 8. Velocity and pressure locations within doorway for PHOENICS models describing soffit as a zero thickness plate and a two cell thick soffit.

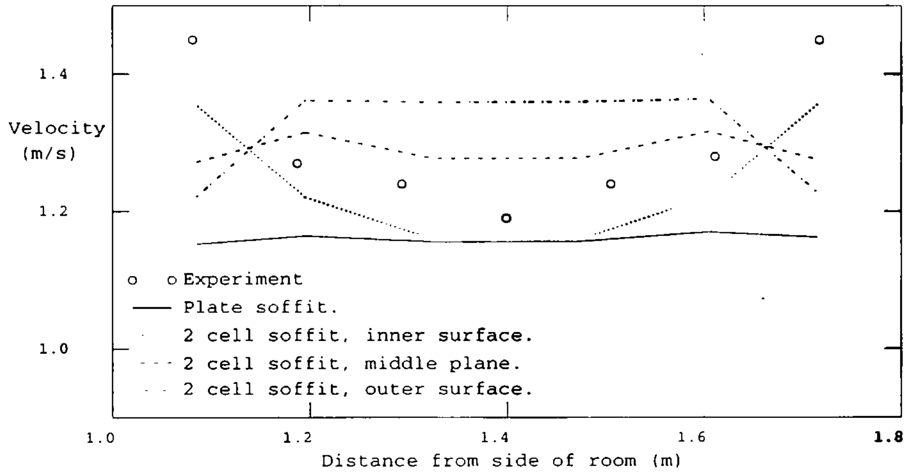
With two cells defining the door soffit three velocities are produced within the door jamb: one at its centre plane, one at its inside plane, and another at its outside plane (see Figure 8b). All three velocities have at least one driving pressure point located within the door jamb.

Figure 9 depicts horizontal velocity profiles at 1.66 m above the floor for the experiment, the thin surface soffit representation, and the two cell soffit at the inside plane, centre plane, and outside plane locations. The experimental velocity curve peaks towards the door sides, producing a concave profile. The thin surface soffit curve is virtually flat with a slight decrease at the sides. This is similar to the velocity profile in the centre of the two cell soffit; however in this case there appears to be a greater downturn towards the door sides. The velocity profile at the inside plane of the jamb agrees most closely with the trends revealed in the experiment, producing a concave profile while the profile at the outside plane appears inverted with the velocity attaining minimum values towards the door sides.

When the door jamb is represented by a single cell, two velocity distributions are produced within the door jamb. One velocity distribution is at its inside plane which peaks towards the door sides; another is at its outside plane which peaks towards the centre of the door. These distributions are similar to those generated by the two cell soffit at the same locations. The velocity distribution in the centre plane of the jamb can be determined by averaging these velocities. An almost flat velocity distribution results, which is similar to that found in both the plate soffit and in the centre plane of the two cell soffit.

Clearly, a velocity gradient exists through the thickness of the door jamb which is impossible to detect using a thin surface representation of the soffit. If a two cell

Figure 9. Measured and predicted horizontal velocity distribution at various locations within the doorway for the plate and two cell thick soffit.



soffit representation is used, the numerical velocity profile matches the measured profile, but its location appears to be displaced by as much as 0.05 m towards the inside edge of the jamb.

With regard to the experimental results, there are two issues of accuracy to be considered. First, as previously discussed, the velocity measurements in the vicinity of the door edges are prone to large errors if the bi-directional probes are not aligned parallel to the flow. Secondly, the velocity probes function by converting a measured pressure difference across a small but finite distance—typically of the order of 0.02 m —into a velocity. As the door jamb is 0.1 m thick, the exact positioning of the probe becomes crucial if detailed comparisons with field models are to be attempted.

Conclusions

Based on comparisons of upper-layer temperature, mass fluxes into and out of the compartment and neutral plane height the PHOENICS based fire field model produced reasonable agreement with measured room fire data (within +/-20percent for most of the important measured variables). The PHOENICS predicted doorway vertical velocity and temperature profiles are also in good agreement with the measured profiles.

The poorest agreement was achieved for the upper layer temperatures when the fire was located adjacent to the rear wall. In these cases the predicted temperatures were as much as 44 percent higher than those measured. The model also had difficulty in predicting the corner temperature stratification with severe smearing occurring in the predicted temperatures for the centre fire location cases. These differences are expected to be due to the simplicity of the treatment of the wall boundary conditions.

An investigation of the doorway horizontal velocity profile reveals that careful modelling of this region is essential in order to correctly predict measured trends. Furthermore, the predicted profile appears to be sensitive to the exact location within the door jamb, suggesting the need for extremely accurate and careful experimental practices.

Further work in fire model validation is currently underway at the University of Greenwich. The continued development of fire field modelling technology and its general acceptance as a tool for practical applications would however benefit greatly from a systematic effort in their "validation", and this will rely on the establishment of a coherent data base of test data.

ACKNOWLEDGEMENTS

Professor E. Galea is indebted to the U.K. CAA for their financial support of his personal chair in Mathematical Modelling at the University of Greenwich. Mr. R.N. Mawhinney acknowledges the SERC and AEA Technology for financial support. Finally, the authors are indebted to CHAM for allowing the use of PHOENICS.

REFERENCES

1. Thomas, P.H., Hinkley, P.L., Theobald, C.R., and Simms, D.L., "Investigations into the Flow of Hot Gases in Roof Venting," Fire Research Tech Paper No. 7, HMSO, 1963.

2. Galea, E.R, "On the Field Modelling Approach to the Simulation of Enclosure Fires," *Journal of Fire Protection Engineering*, Vol. **1**, No. 1, pp. 11-22, 1989.

3. Satoh, K., Lloyd, J.R, Yang, K.T., and Kanury, A.M., "A Numerical Finite-Difference Study of the Oscillatory Behavior of Vertically Vented Compartments," *Proceedings of the 2nd National Symposium on Numerical Methods in Heat Transfer*, USA, 1981.

4. Markatos, N.C., Malin, M.R., and Cox, G., "Mathematical Modelling of Buoyancy Induced Smoke Flow in Enclosures," *International Journal of Heat Mass Transfer*, Vol. **25**, No. 1, 1982, pp. 63.

5. Galea, E.R. and Markatos, N.C., "The Mathematical Modelling and Computer Simulation of Fire Development in Aircraft," *International Journal of Heat Mass Transfer*, Vol. **34**. No. 1, 1991 pp. 181–197.

6. Simcox, S., Wilkes, N.S., and Jones, I.P., "Fire at Kings Cross Underground Station," 18th November 1987; Numerical Simulation of the Buoyant Flow and Heat Transfer, U.K. Atomic Energy Authority Harwell Report, 1988, AERE-G 4677.

7. Galea, E.R., Hoffmann, N. and Berhane, D., "Large-Scale Fire Field Modelling—the Route to General Use via Parallel Processing," *Interflam '93*, Interscience, 1993, pp. 307–319.

8. Than, C-F and Savilonis, B.J., "Modeling Fire Behavior in an Enclosure with a Ceiling Vent," *Fire Safety Journal*, Vol. **20**, 1993, pp. 151–174.

9. Chow, W.K. and Fong, N.K., "Numerical Studies on the Sprinkler Fire Interaction Using Field Modelling Technique," *Proceedings of the 5th International Conference of Fire Safety Science*, Interflam, 1990, pp. 25–33.

10. Hadjisophocleous, G.V. and Cacambouras, M., "Computer Modelling of Compartment Fires," *Journal of Fire Protection Engineering*, Vol. **5**, No. 2, 1993, pp. 39–52.

11. Kumar S., Gupta, A.K. and Cox, G., "Effects of Thermal Radiation on the Fluid Dynamics of Compartment Fires," *Proceedings of the 3rd International Symposium on Fire Safety Science*, Elsevier Applied Science, 1991, pp. 345–354.

12. Burns, A.D. and Wilkes, N.S., "A Finite-Difference Method for the Computation of Fluid Flows in Complex Three-Dimensional Geometries," U.K. Atomic Energy Authority Harwell Report, AERE-R 12342.

13. Spalding, D.B., "A General Purpose Computer Program for Multidimensional One- and Two-Phase Flow," *Mathematics and Computers in Simulations*, North Holland (IMACS), Vol. **23**, 1981, pp. 267.

14. Steckler, K.D., Quintiere, J.G. and Rinkinen, W.J., *Flow Induced by Fire in a Compartment*, NBSIR 82-2520, National Bureau of Standards, Washington, September 1982.

15. Kerrison, L.N., Galea, E.R., Hoffmann, N., and Patel, M.K., "A Comparison of a FLOW3D based Fire Field Model with Experimental Room Fire Data," To appear in *Fire Safety Journal*.

16. Patankar, S.V., *Numerical Heat Transfer and Fluid Flow*, McGraw Hill, New York, USA, 1980.

17. Patankar, S.V. and Spalding, D.B., "A Calculation Procedure for Heat, Mass and Momentum Transfer in Three-Dimensional Parabolic Flows," *Inter-*

national Journal of Heat Mass Transfer, Vol. **15**, 1972, pp. 1787–1806.

18. Kerrison, L., Mawhinney, N., Galea, E.R., Hoffmann, N. and Patel, M.K., "A Comparison of Two Fire Field Models with Experimental Room Fire Data," *Proceedings of the 4th International Symposium on Fire Safety Science*, 1994, pp. 13-17

19. Williamson, R.B., Revenaugh, A. and Mowrer, R.W., "Ignition Sources in Room Fire Tests and Some Implications for Flame Spread Evaluation," *Proceedings of the 3rd Int ernational Symposium on Fire Safety Science*, Elsevier Applied Science, 1991, pp. 657–666,

20. Steckler, K.D., Baum H.R., and Quintiere, J.G., "Fire Induced Flows Through Room Openings—Flow Coefficients," *Twentieth Symposium on Combustion*, The Combustion Institute, 1984, pp. 1591–1600.