A Comparison of a FLOW3D Based Fire Field Model with Experimental Room Fire Data

L. Kerrison, E. R. Galea, N. Hoffmann & M. K. Patel

Fire Safety Engineering Group, Centre for Numerical Modelling and Process Analysis,
The University of Greenwich, London SE18 6PF, UK

(Received 13 December 1993; revised version received 14 July 1994; accepted 15 September 1994)

ABSTRACT

This paper describes the application of a fire field model based on the FLOW3D CFD software to the simulation of fire induced flows in domestic sized rooms. Several scenarios are examined consisting of various sized fires, 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 the results suggest that significant mesh refinement is required to produce results in quantitative agreement with experimental results. A close examination of the horizontal door-way velocity profiles highlights the need for careful modelling and experimental practices in this region.

INTRODUCTION

The systematic comparison of fire field model predictions with experimental data (often called 'validation') is an essential step in the continual development and acceptance of this complex and expensive procedure. 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 addition, as the number and variety of fire field models increase it becomes essential 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 zonal models. 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 and PHOENICS) with experimental data. In this paper a comparison is made of field model predictions based on FLOW3D with experimental data. The ability of the code to model a series of room fire experiments is examined and a further comparison is made with an earlier study produced using the JASMINE code. The results of the PHOENICS prediction will be the subject of a later publication.

THE EXPERIMENTS

A series of 45 experiments was conducted by Steckler, et al. to investigate fire induced flows in a compartment measuring 2.8 m x 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 light weight 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, 105.3 and 158 kW. Near steady-state conditions were achieved within 30 minutes.

A two-dimensional array of bi-directional velocity probes and bare-wire thermocouples was placed in a vertical plane located midway between the inner and outer edges of the door jamb. In this way temperatures and velocities could be measured throughout the door.
way across its width and through its height. In addition, a stack of aspirated thermocouples were placed in the front corner of the room to measure the gas temperature profile (see Fig. 1).

**THE NUMERICAL SIMULATIONS**

A number of simulations was performed based on four fire locations (Fig. 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 FLOW3D (Version 2.3.3) 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\textsuperscript{10,11,17}. Compressibility is assumed and the perfect gas law is used to describe the equation of state. The precise formulation of the differential equations describing the model will not be given here as they may be found elsewhere.\textsuperscript{10-12,18} Combustion and radiation are ignored in these simulations, the fire being simulated as a simple source of heat appropriate to the case under consideration.

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 or isothermal conditions for the temperature. In the isothermal cases the wall temperatures were set to ambient values. The usual ‘wall functions’\textsuperscript{10,11,18} 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.

For the majority of cases, a mesh of 8280 cells in total (6480 internal and 1800 external cells) was used to discretise the geometry. The mesh consisted of 23 cells in length, 18 cells in width and 20 cells in height. 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.

In each case the heat source representing the fire was placed in the appropriate location and described as a rectangular burner of the same surface area as the round burners used in the experiment. In the adiabatic cases, the heat release rate was modified in order to account for heat lost to the walls and via radiation during the experiment. This was achieved by calculating from the provided experimental data, for each configuration examined, the heat convected out of the room during the experiment and using this value as the heat release rate for the numerical calculations. In the majority of cases the calculated heat loss was small, accounting for approximately 3 kW. A maximum value of 14 kW was achieved for the corner fire with the narrow door opening. For the isothermal cases, the full heat release rate was used in the numerical calculations, losses due to radiation being ignored.

FLOW3D was run in transient mode, typically requiring 200 time steps before reaching steady-state conditions. Only the steady-state results are considered in this paper. Within each time step, convergence was assumed if the mass source residual fell below $1 \times 10^{-0.4}$ (and the other key residual measures, e.g. enthalpy fell by corresponding amounts). If the time step iteration was not stopped by these measures then a maximum of 100 iterations would be performed during the time step. Steady-state was considered to be achieved when the maximum
change between spot values fell to less than 1% between time steps. Using a SPARC 10 workstation, run times were of the order of 26 CPU hours for a typical transient simulation.

The hybrid differencing scheme was used throughout and the Stone method was used to solve the momentum and enthalpy equations. The pressure-correction equation was solved using the conjugate gradient method and the turbulence quantities were solved using a line method. The fully implicit backward differencing scheme was used for the time discretisation.

RESULTS AND DISCUSSION

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 Fig. 2 graphically depicts the level of agreement between measured and predicted results and Figs 3–10 depict comparisons of model predictions with the observed trends.

The upper layer temperature was determined experimentally by averaging the temperature values in the upper layer as measured by

![Graph showing temperature predictions vs experimental results]

**Fig. 2(a).** Calculated and measured upper layer temperature (°C) for centre, corner, back and front positioned fires. Model predictions use isothermal boundary conditions. Central curve depicts line of equivalence (i.e. y = x).
Fig. 2(b). Calculated and measured upper layer temperature (°C) for centre, corner, back and front positioned fires. Model predictions use adiabatic boundary conditions. Central curve depicts line of equivalence (i.e. $y = x$).

Fig. 2(c). Calculated and measured neutral plane height/door height for centre, corner, back and front positioned fires. Model predictions use isothermal boundary conditions. Central curve depicts line of equivalence (i.e. $y = x$).
FLOW3D based fire field model

N/H Predicted vs N/H Exp
(adiabatic conditions)

Fig. 2(d). Calculated and measured neutral plane height/door height for centre, corner, back and front positioned fires. Model predictions use adiabatic boundary conditions. Central curve depicts line of equivalence (i.e. \( y = x \)).

MF Predicted vs Experimental

Fig. 2(e). Calculated and measured mass flux in and out (kg/s) for centre, corner, back and front positioned fires. Model predictions use isothermal boundary conditions. Central curve depicts line of equivalence (i.e. \( y = x \)).
Fig. 2(f). Calculated and measured mass flux in and out (kg/s) for centre, corner, back and front positioned fires. Model predictions use adiabatic boundary conditions. Central curve depicts line of equality (i.e. \( y = x \)).

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%, bringing the majority of the predictions (both adiabatic and isothermal) within the range of experimental errors [Figs 2(c) and 2(d)]. This suggests that the neutral plane predictions are not as sensitive as the temperature predictions to the thermal boundary condition specification.

The measured mass flux into and out of the room is subject to
errors of between 10\% and 13\%.\textsuperscript{13} Predictions of this quantity for fires located away from the walls show good agreement with experimentation, however for fires located adjacent to walls differences of up to 40\% occur [Fig 2(e) and 2(f)]. This trend was also observed in the JASMINE predictions\textsuperscript{15} all-be-it with a slightly coarser mesh (6270 cells). These results also suggest that the mass flux predictions are not as sensitive as are the temperature predictions to the wall thermal boundary condition specification.

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 bi-directional velocity probes must be aligned parallel to the velocity streamlines.\textsuperscript{13,14} 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 measurement errors in the velocity and hence the mass flux.

The burner was modelled as a heat source of the appropriate heat output with a rectangular rather than a circular section. For the fires located well away from the walls this appears to be a reasonable approximation. 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. However, in the model, the burner was located immediately adjacent to the walls. Williamson \textit{et al.}\textsuperscript{20} 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 of approximately 40\degree C (approximately 16\%) 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 corner and back wall located fire tests. In cases where the walls were treated as perfectly insulating, significant differences in temperatures were observed. These differences were greatly reduced by using an isothermal condition. Clearly, heat losses to the wall play a significant role in these cases and a full treatment utilising wall material properties is required.

Finally, the exclusion of radiation from the model description is expected to have an influence on the predicted results. The JASMINE\textsuperscript{15} 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, with an unchanged or worse agreement for those cases where the fire was located away from the walls. The average upper layer temperature is higher and hence the effects of radiation are expected to be more significant for those cases in which the fire is located adjacent to the walls.

The overall trends observed in the experiment have been captured by the fire field model (see Table 1). With isothermal boundary conditions, as the door width is increased from 0.24 m to 0.99 m, the height of the neutral plane is observed to increase [Fig. 3(a)], the temperature of the hot layer decreases [Fig. 3(b)] and the mass flux in and out of the room also increases [Figs. 3(c) and (d)]. Furthermore, as the fire strength is increased from 31.6 to 61.9 kW, the height of the neutral plane decreases [Fig. 4(a)] while the temperature of the hot layer [Fig. 4(b)] and the mass flux in and out [Figs 4(c) and (d)] increases. Similar trends are observed for the model with the adiabatic boundary conditions. Unless otherwise stated, the remaining numerical predictions refer to the model with the isothermal boundary conditions.

Predictions of doorway centreline velocities as a function of door width are depicted in Figs 5(a) and 5(b) for the centre and corner 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 62.9 kW centre fire with 0.74 m door scenario was repeated with a refined mesh (Fig. 6). The mesh was increased to an equivalent of 61,916 cells in total using a mirror plane of symmetry along the centre axis of the compartment passing through the centre of the fire and open doorway. While the mesh refinement has improved the overall level of agreement [see Fig. 6(a)], the under-prediction in the maximum velocity while reduced, still persists. The accuracy of the experimental results in this region are questionable. As discussed above, the velocity measurements in the vicinity of the door edges may be prone to errors if the bi-directional probes are not aligned parallel to the flow. The nature of this flow also has the potential to cause numerical errors due to false diffusion. A finer local mesh refinement in the vicinity of the upper regions of the doorway may be necessary to further reduce these differences.

An interesting feature of the corner fire test doorway velocity profiles is a kink which appears near the hot–cold interface in the experimental
<table>
<thead>
<tr>
<th>Scenario</th>
<th>N/H₀</th>
<th>Mass flow rate (kg/sec)</th>
<th>Average hot layer temp (°C)</th>
</tr>
</thead>
<tbody>
<tr>
<td>A, 0.24 m, 62.9 kW</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Numerical†</td>
<td>0.476</td>
<td>0.237</td>
<td>0.246</td>
</tr>
<tr>
<td>Numerical‡</td>
<td>0.483</td>
<td>0.245</td>
<td>0.248</td>
</tr>
<tr>
<td>Experimental</td>
<td>0.499</td>
<td>0.255</td>
<td>0.247</td>
</tr>
<tr>
<td>A, 0.74 m, 62.9 kW</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Numerical†</td>
<td>0.505</td>
<td>0.576</td>
<td>0.584</td>
</tr>
<tr>
<td>Numerical‡</td>
<td>0.493</td>
<td>0.604</td>
<td>0.610</td>
</tr>
<tr>
<td>Experimental</td>
<td>0.561</td>
<td>0.554</td>
<td>0.571</td>
</tr>
<tr>
<td>Fine Mesh†</td>
<td>0.547</td>
<td>0.501</td>
<td>0.529</td>
</tr>
<tr>
<td>Fine mesh‡</td>
<td>0.571</td>
<td>0.510</td>
<td>0.546</td>
</tr>
<tr>
<td>A, 0.99 m, 62.9 kW</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Numerical†</td>
<td>0.515</td>
<td>0.732</td>
<td>0.741</td>
</tr>
<tr>
<td>Numerical‡</td>
<td>0.508</td>
<td>0.761</td>
<td>0.769</td>
</tr>
<tr>
<td>Experimental</td>
<td>0.582</td>
<td>0.653</td>
<td>0.701</td>
</tr>
<tr>
<td>A, 0.74 m, 31.6 kW</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Numerical†</td>
<td>0.510</td>
<td>0.477</td>
<td>0.485</td>
</tr>
<tr>
<td>Numerical‡</td>
<td>0.507</td>
<td>0.470</td>
<td>0.495</td>
</tr>
<tr>
<td>Experimental</td>
<td>0.569</td>
<td>0.430</td>
<td>0.461</td>
</tr>
<tr>
<td>B, 0.24 m, 62.9 kW</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Numerical†</td>
<td>0.575</td>
<td>0.161</td>
<td>0.163</td>
</tr>
<tr>
<td>Numerical‡</td>
<td>0.556</td>
<td>0.168</td>
<td>0.168</td>
</tr>
<tr>
<td>Experimental</td>
<td>0.523</td>
<td>0.201</td>
<td>0.205</td>
</tr>
<tr>
<td>B, 0.74 m, 62.9 kW</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Numerical†</td>
<td>0.634</td>
<td>0.284</td>
<td>0.287</td>
</tr>
<tr>
<td>Numerical‡</td>
<td>0.671</td>
<td>0.288</td>
<td>0.288</td>
</tr>
<tr>
<td>Experimental</td>
<td>0.566</td>
<td>0.435</td>
<td>0.462</td>
</tr>
<tr>
<td>B, 0.99 m, 62.9 kW</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Numerical†</td>
<td>0.694</td>
<td>0.298</td>
<td>0.299</td>
</tr>
<tr>
<td>Numerical‡</td>
<td>0.685</td>
<td>0.299</td>
<td>0.299</td>
</tr>
<tr>
<td>Experimental</td>
<td>0.586</td>
<td>0.513</td>
<td>0.491</td>
</tr>
<tr>
<td>B, 0.74 m, 31.6 kW</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Numerical†</td>
<td>0.641</td>
<td>0.234</td>
<td>0.237</td>
</tr>
<tr>
<td>Numerical‡</td>
<td>0.656</td>
<td>0.251</td>
<td>0.253</td>
</tr>
<tr>
<td>Experimental</td>
<td>0.580</td>
<td>0.343</td>
<td>0.346</td>
</tr>
<tr>
<td>G, 0.74 m, 62.9 kW</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Numerical†</td>
<td>0.487</td>
<td>0.602</td>
<td>0.639</td>
</tr>
<tr>
<td>Numerical‡</td>
<td>0.492</td>
<td>0.638</td>
<td>0.675</td>
</tr>
<tr>
<td>Experimental</td>
<td>0.580</td>
<td>0.550</td>
<td>0.601</td>
</tr>
<tr>
<td>C, 0.74 m, 62.9 kW</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Numerical†</td>
<td>0.653</td>
<td>0.331</td>
<td>0.332</td>
</tr>
<tr>
<td>Numerical‡</td>
<td>0.673</td>
<td>0.440</td>
<td>0.328</td>
</tr>
<tr>
<td>Experimental</td>
<td>0.579</td>
<td>0.474</td>
<td>0.476</td>
</tr>
</tbody>
</table>

† Isothermal conditions.
‡ Adiabatic conditions.
Fig. 3(a). Calculated and measured neutral plane height/door height as a function of door width (m) for centre, corner, back and front positioned fires.

Fig. 3(b). Calculated and measured hot layer temperature (°C) as a function of door width (m) for centre, corner, back and front positioned fires.
MFR (in) vs Door Width

Fig. 3(c). Calculated and measured mass flux in (kg/s) as a function of door width (m) for centre, corner, back and front positioned fires.

MFR (out) vs Door Width

Fig. 3(d). Calculated and measured mass flux out (kg/s) as a function of door width (m) for centre, corner, back and front positioned fires.
Fig. 4(a). Calculated and measured neutral plane height/door height as a function of fire size (kW) for centre, corner, back and front positioned fires.

Fig. 4(b). Calculated and measured upper layer temperature (°C) as a function of fire size (kW) for centre, corner, back and front positioned fires.
**MFR (in) vs Fire Strength**

![Graph](image)

**Fig. 4(c).** Calculated and measured mass flux in (kg/s) as a function of fire size (kW) for centre, corner and back positioned fires.

**MFR (out) vs Fire Strength**

![Graph](image)

**Fig. 4(d).** Calculated and measured mass flux out (kg/s) as a function of fire size (kW) for centre, corner and back positioned fires.
Fig. 5(a). Numerical and experimental door centre vertical velocity profiles for the 0.24 m, 0.74 m and 0.99 m wide doors. The 62.9 kW fire was centrally located.

Fig. 5(b). Numerical and experimental door centre vertical velocity profiles for the 0.24 m, 0.74 m and 0.99 m wide doors. The 62.9 kW fire was located in the corner of the room.
results for the two widest doors. This feature has also been detected in the FLOW3D predictions [Fig. 5(b)]. In the JASMINE$^{15}$ simulations, this was only observed for cases which included radiation.

Predictions of doorway centreline temperatures as a function of door width are depicted in Figs 7(a) and 7(b) for the centre and corner 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. Here again a kink in the measured temperature profile occurs at the hot-cold layer interface for the corner fire which is also detected by the FLOW3D predictions [Fig. 7(b)]. For the two widest doors, the model over predicts the upper layer temperatures for the corner fire [Fig. 7(b)] while correctly predicting those for the centre fire case. This is consistent with the above discussion. For the centre fire scenario, the results for the refined mesh show an improved agreement with measured results [Fig. 6(b)] in the lower portions of the room and little or no improvement in the higher regions.

The narrow door case consistently produced poor levels of agreement. Temperatures are over-predicted in the hot layer [Figs. 7(a) and
Fig. 6(b). Numerical and experimental door centre vertical temperature profiles for the 0.74 m wide door. The 62.9 kW fire was located centrally and both coarse and fine numerical results are presented.

(b) and velocities are under-predicted in the upper and lower layers [Figs. 5(a) and (b)], particularly for the corner fire case [Fig. 5(b)]. This is probably due to poor mesh refinement in the narrow door case as only two cells were used across the width of the door compared with eight cells in the wide door case.

The poorest level of overall agreement was observed for the corner stack temperatures [Fig. 8(b)]. For the middle fire, with the exception of the narrow door case, the model appears to predict the upper layer temperature with a high level of accuracy (3–1%) with a similar level of accuracy for the lower layer temperatures [Fig. 8(a)]. However, for the corner fire [Fig. 8(b)], the upper and lower layer temperatures are over predicted by as much as 25%.

The degree to which the fire plume is deflected backwards by the induced incoming flow can be seen in Fig. 9 for the 0.74 m door. While it is extremely difficult to make accurate measurements, the inclination of the FLOW3D (Fig. 9) produced plume appears to be similar to that produced by the JASMINE15 code. These observations are consistent with the comments made by Quintiere et al.21 who observed, in an
Fig. 6(c). Numerical and experimental corner stack vertical temperature profiles for the 0.74 m wide door compartment. The 62.9 kW fire was located centrally and both coarse and fine numerical results are presented.

Fig. 7(a). Numerical and experimental door centre vertical temperature profiles for the 0.24 m, 0.74 m and 0.99 m wide doors. The 62.9 kW fire was centrally located.
Fig. 7(b). Numerical and experimental door centre vertical temperature profiles for the 0·24 m, 0·74 m and 0·99 m wide doors. The 62·9 kW fire was located in the corner of the room.

Fig. 8(a). Numerical and experimental corner stack vertical temperature profiles for the 0·24 m, 0·74 m and 0·99 m wide doors. The 62·9 kW fire was centrally located.
Fig. 8(b). Numerical and experimental corner stack vertical temperature profiles for the 0.24 m, 0.74 m and 0.99 m wide doors. The 62.9 kW fire was located in the corner of the room.

Fig. 9. FLOW3D predicted temperature distribution through the centre of the fire compartment passing through the centre of the 0.74 m wide door. The 62.9 kW fire was centrally located.
earlier series of experiments, the plume to be 'blown' over by the incoming draft of air.

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 edges with a minimum in the middle. Steckler et al.\textsuperscript{14} have argued that these results are consistent with potential flow theory if the flow is assumed to be non-rotational and inviscid. However, in their JASMINE simulation, Kumar et al.\textsuperscript{15} noted completely the opposite result, i.e. peak velocities were predicted in the centre of the door. In their simulation, the door wall appears to have been modelled through the use of a thin surface. The significance of this approximation is examined in detail in another publication currently in preparation\textsuperscript{16} in which the PHOENICS code is used.

For the FLOW3D models the depth of the soffit (distance between the inner and outer edges of the door jamb) was represented by two cells measuring 0.05 m each. The case presented here relates to the 62.9 kW central fire with 0.74 m door (Fig. 10).

In the upper reaches of the door, the experimental velocity curve peaks towards the door jamb edges producing a concave profile. The
predicted horizontal velocity profile agrees with the trends revealed in the experiment producing a concave profile. In the lower reaches of the door, the flow is directed inwards producing negative velocity values. In this case the experimental velocity curve peaks towards the centre of the door (essentially a mirror image of the higher velocity profile). These observed trends in experimental results were also reproduced by the numerical predictions.

**CONCLUSIONS**

Based on comparisons of upper-layer temperature, mass fluxes into and out of the compartment and neutral plane height, the FLOW3D based fire model produced reasonable agreement with measured room fire data. The FLOW3D predicted doorway vertical velocity and temperature profiles are also in good agreement with the measured profiles. Temperature predictions based on models with isothermal boundary conditions agreed more closely with the observed results than those models with adiabatic boundary conditions. For fires located adjacent to the confining walls a full treatment of the wall thermal boundary conditions may be necessary in order to produce more realistic results. The mass fluxes and neutral plane height were not as sensitive as the temperature to the nature of the thermal boundary condition.

The model had greatest difficulty in predicting mass fluxes into and out of the room for fires located close to the confining walls of the compartment. While most predicted quantities were within ±20% of the measured values, the differences associated with the mass flux predictions for the corner fires were within ±40%.

Through careful modelling of the doorway region it was possible to correctly predict the observed shape of the horizontal velocity distribution.

Finally, the thorough 'validation' of fire field models requires the establishment and general availability of a data base of experimental results produced for the express purposes of field model validation. As fire field modelling encompasses areas in fire sciences, mathematics and software engineering, the range of data gathered for such a data base also needs to target issues raised by these disciplines. To establish such a data base will require the cooperation of both modellers and experimentalists. Further work in fire model validation is currently underway at the University of Greenwich and the authors welcome cooperation with other interested groups.
ACKNOWLEDGEMENTS

Professor E. Galea is indebted to the CAA for their financial support of his personal chain in Mathematical Modelling at the University of Greenwich. L. Kerrison acknowledges the EPSRC for financial support. Finally, the authors are indebted to the UKAEA for allowing the use of FLOW3D.

REFERENCES


