VALIDATION OF COMPUTATIONAL FLUID DYNAMICS MODEL FOR PREDICTING ROOM FIRES

Jojo S.M. Li
Department of Building Services Engineering, The Hong Kong Polytechnic University, Hong Kong, China

(Received 27 February 2003; Accepted 31 March 2003)

ABSTRACT

This paper describes the application of Computational Fluid Dynamics (CFD) to predict air flow and temperature distributions induced by a fire in a compartment. The theory behind this is to solve a system of partial differential equations describing conservation of mass, momentum and heat with the k-ε turbulence model. Predicted results on temperature are compared with experimental data measured from full-scale burning tests. For most cases, the predicted results agree reasonably well with the experimental data. An examination of the vertical temperature profiles of a fire positioned adjacent to the rear wall highlights the need for careful modelling and experimental practices. Sensitivity studies on varying numerical properties such as grid size and iteration number will also be reported.

1. INTRODUCTION

Computational Fluid Dynamics (CFD) is now extensively used in building design to help devise suitable mitigation and evacuation strategies. CFD fire simulations can provide insight and understanding for hazardous scenarios that may not be practical to investigate experimentally. For applications in the field of safety case analyses, one of the barriers to the practical use of CFD to make important decisions is validation of the model. There is argument on whether CFD can be used to demonstrate a safe design and operation basis, by making intelligent decisions on the known strengths and weaknesses of the mathematical model. Therefore, a procedure to validate the models to predict complex flows is very important, that is, a systematic comparison of model predictions with reliable experimental data.

Popular field models [e.g. 1-5] such as PHOENICS, JASMINE and CFX/FLOW3D have been evaluated for simulating building fires. The CFD model PHOENICS 3.2 [6] was used as a simulation tool in this paper. Using the FLAIR menu, air flow patterns, temperature distributions and smoke movement in enclosed spaces can be predicted. CFD results would be evaluated by comparing with the physical data on room fire by Steckler et al. [7]. That set of experiments was commonly used for validating CFD fire models.

2. NUMERICAL SIMULATIONS OF STECKLER ROOM FIRE EXPERIMENT

Results of the compartment fire experiment conducted by Steckler et al. [7] can be used to verify the CFD results. Those experiments were conducted to investigate fire-induced flows through the opening in a compartment of size 2.8 m × 2.8 m × 2.18 m (height). A gas burner was placed at different floor locations with various compartment openings. The parameters in FLAIR for simulating this room fire are shown in Table 1.

A number of simulations were performed with the fire at two different locations, A and C as shown in Fig. 1. The door width and height were 0.74 m and 1.83 m respectively and the fire size was 62.9 kW. The simulations were performed with a uniform mesh with two different grid finenesses, and the main parameters are summarised in Figs. 2 and 3. The effect of grid refinement was examined by refining the coarse grid, giving 9 cells at the fire source instead of one, and approximately 2 times refinement in volume elsewhere. Apart from the variation in grid spacing, simulations were set to run in 50 and 100 iterations. The air temperatures for different number of iterations and different grid size are compared.

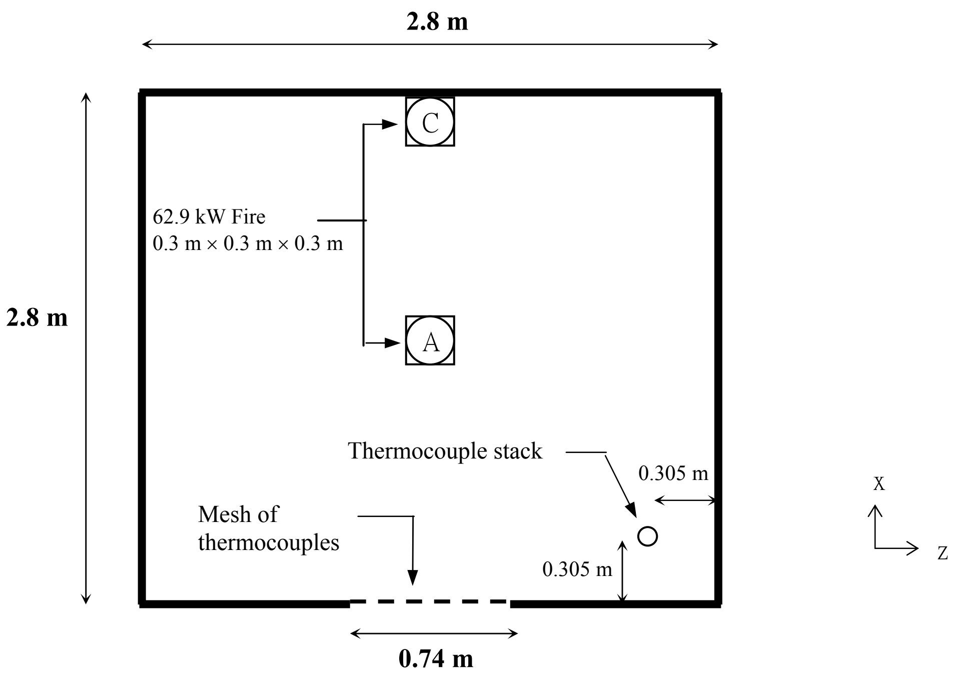
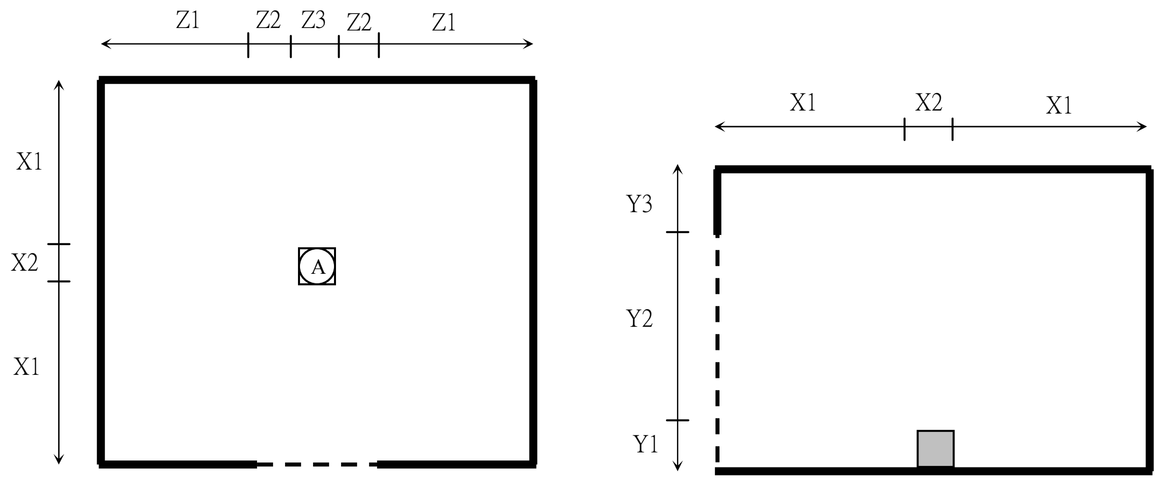


Fig. 1: Configuration of experiment by Steckler et al.



(a) In x-z plane

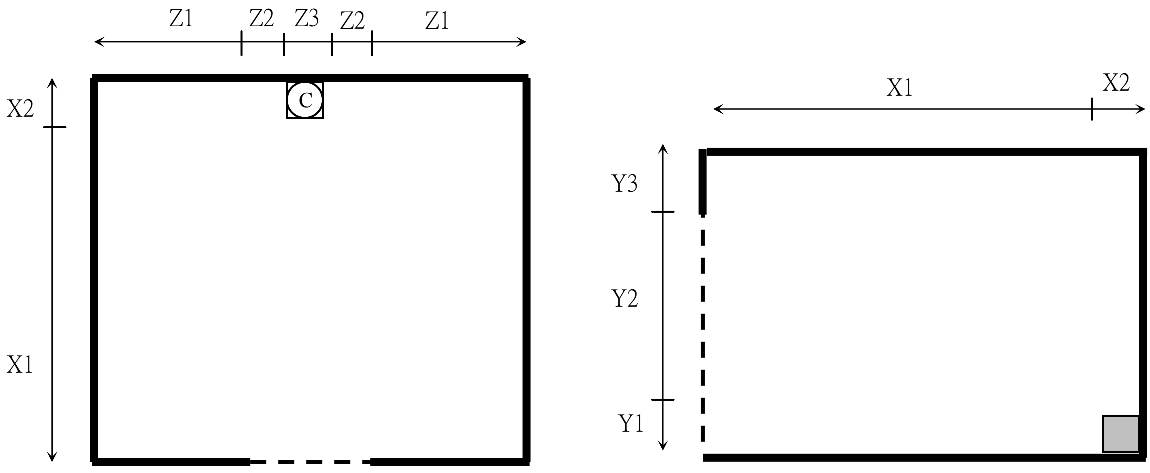
(b) In x-y plane

Grid	No. of grids in x direction		No. of grids in y direction			No. of grids in z direction			Total no. of grids
	X1	X2	Y1	Y2	Y3	Z1	Z2	Z3	
Coarse	10	1	1	12	3	8	2	1	7056
Fine I	13	3	3	16	4	10	3	3	19343
Fine II	13	1	1	16	4	10	3	1	15309

Fig. 2: Mesh distribution of centrally located fire in experiments by Steckler et al.

Table 1: Controlled Numerical Simulation Setup

Time dependence	Steady					
Gravitational forces	On					
Buoyancy model	Density difference					
Gravitational acceleration	X = 0		Y = -9.81 ms^{-2}		Z = 0	
Reference density	1.189 kgm^{-3}					
Coefficient for auto wall functions	Log-law					
Global wall roughness	0					
Turbulence model	k-ε					
Radiation model	Off					
Combustion model	Volumetric heat source					
Domain material	Air using Ideal Gas Law					
Reference pressure	1×10^5 Pa					
Reference temperature	273 K					
Initial values	U = 0 ms^{-1}	V = 0 ms^{-1}	W = 0 ms^{-1}	Differential pressure=0, Pa	T = 293 K	KE = 0.01 m^2s^{-2}



(a) In x-z plane

(b) In x-y plane

Grid	No. of grids in x direction		No. of grids in y direction			No. of grids in z direction			Total no. of grids
	X1	X2	Y1	Y2	Y3	Z1	Z2	Z3	
Coarse	20	1	3	12	3	8	2	1	7938
Fine I	25	3	3	16	4	10	3	3	18676
Fine II	25	1	3	16	4	10	3	1	16146

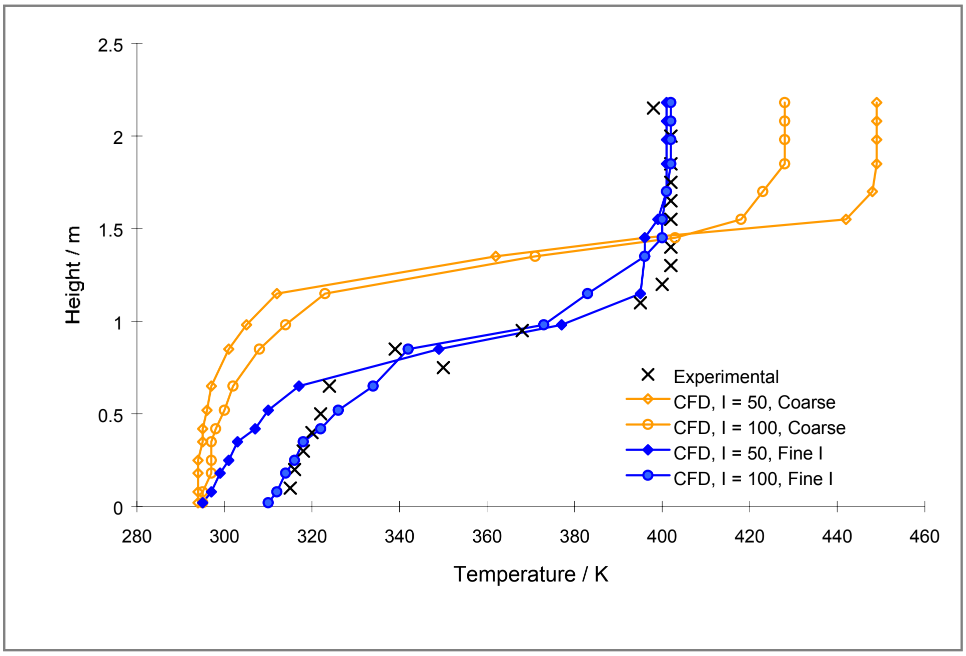
Fig. 3: Mesh distribution of fire located at rear wall in experiments by Steckler et al.

3. COMPARISON OF STECKLER ROOM FIRE MODELLING AND EXPERIMENTAL RESULTS

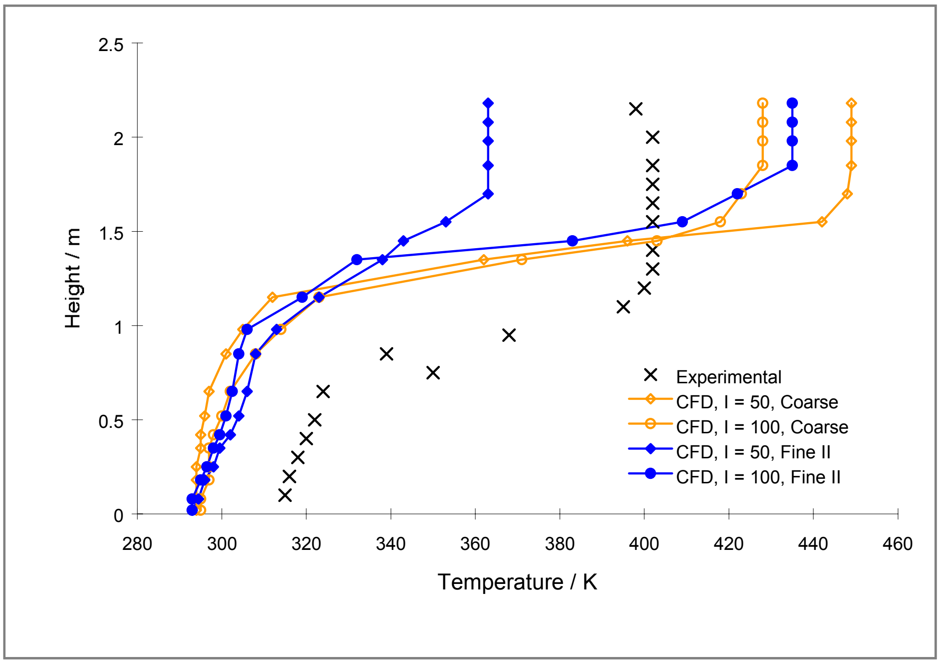
- Fire at room center

Comparisons of the modelling results and experimental results for the natural ventilation tests are shown in Figs. 4 to 7. On the basis of uniform grid spacing, predictions of corner stack and doorway temperatures as a function of different grid finenesses and number of iterations for the centrally located fire are depicted in Figs. 4(a) and 5(a). In both cases, it is found that no matter the number of iterations used, results from coarse grid did not agree very well. On the other hand, the level of agreement with experimental data was largely improved when the grids were refined and higher number of iterations was used. As observed from Fig. 4(a), predictions of corner stack temperature agreed well with experiment. For doorway temperature predictions as shown in Fig. 5(a), the model appeared to predict the upper layer temperature with a high level of accuracy, while results for the lower layer deviated from experiment data by about 10 K.

Comparison of temperature predictions with no local refinement on the fire source is shown in Figs. 4(b) and 5(b). Poorer level of overall agreement is observed for corner stack and doorway temperatures. None of the trials gave predictions in temperature profiles as close as shown in Figs. 4(a) and 5(a), even when fine grids and high iteration number were used. The results disagreed with the experimental data particularly in the region around the height of 0.8 to 1.3 m, where rapid changes of velocity and temperature were experienced. Although the predicted temperatures deviated from the experimental data by about 60 K in maximum, the results seemed to follow the pattern of the measured data.

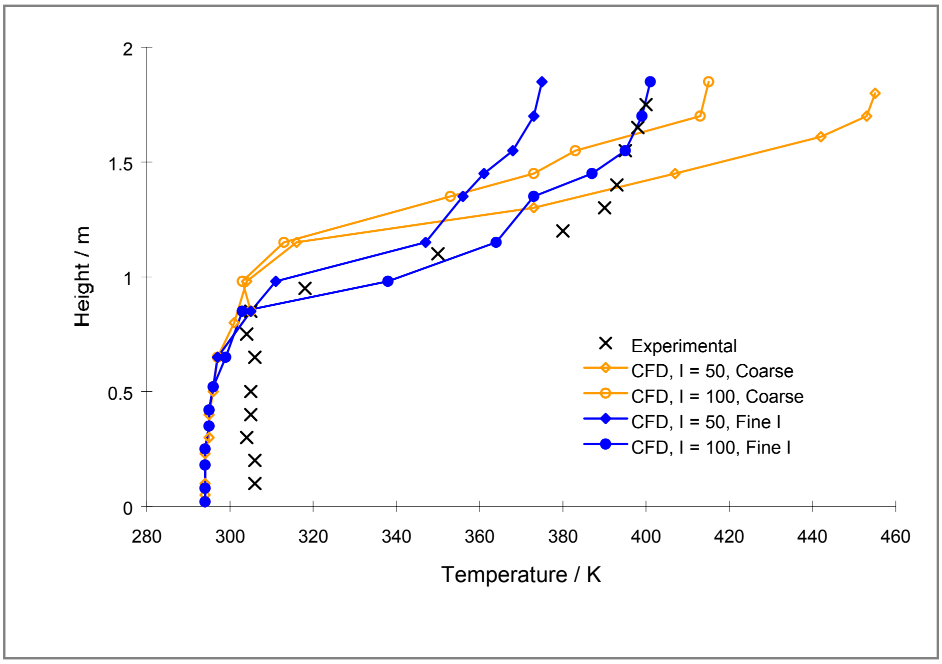


(a) With cell refinement at fire source

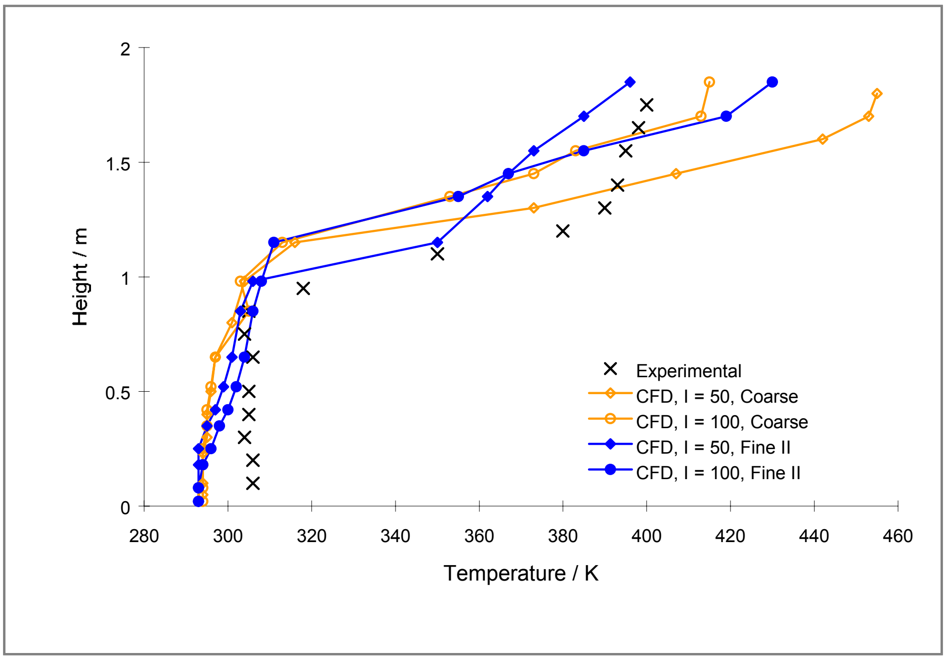


(b) Without cell refinement at fire source

Fig. 4: Predicted and measured vertical temperature profiles at corner stack for centrally located fire



(a) With cell refinement at fire source



(b) Without cell refinement at fire source

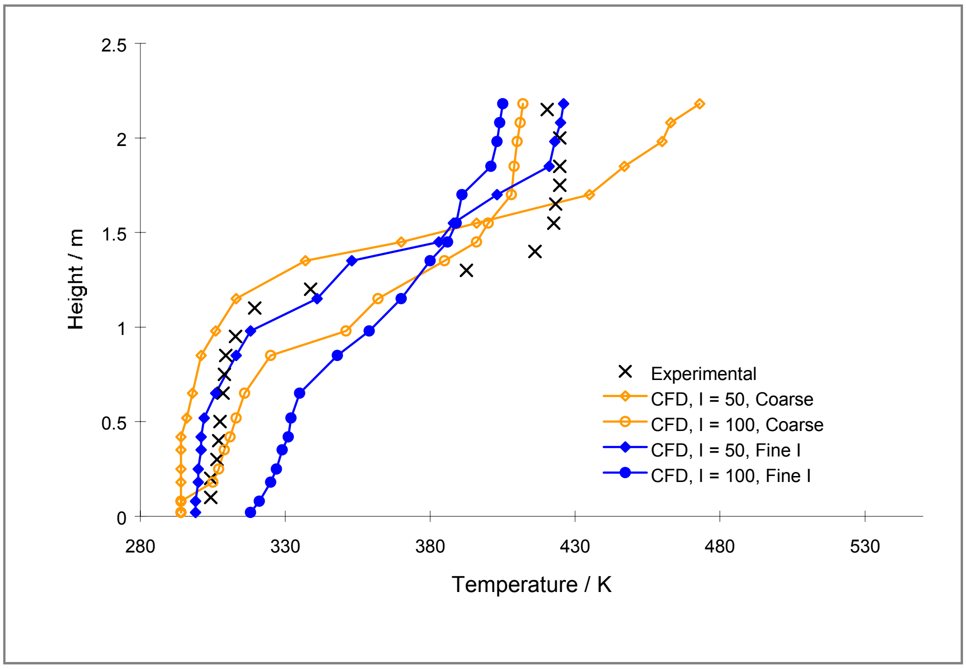
Fig. 5: Predicted and measured vertical temperature profiles at door center for centrally located fire

- Fire located at rear wall

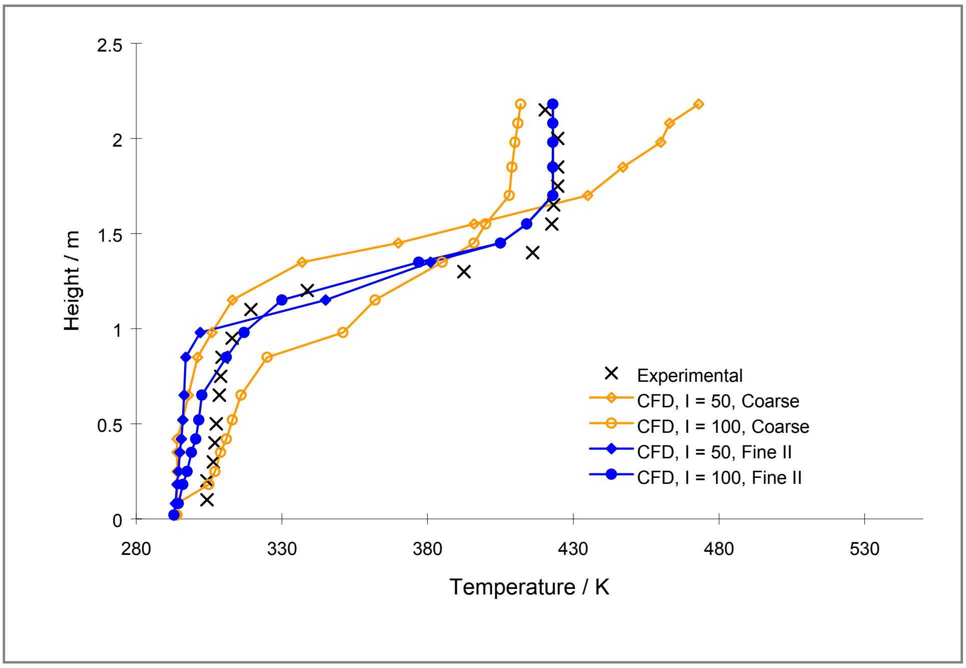
For the fire located at rear wall, the predicted vertical temperature profiles are compared with experimental results in Figs. 6 and 7. Comparisons of corner stack temperature profiles with cell refinement at the fire source are shown in Figs. 6(a) and 7(a). Most of the predicted values from simulations of different iteration number and grid density are scattered around the experimental data. The agreement can be seen as comparatively reasonable at "I = 50, Fine I" for corner stack temperature prediction in Fig. 6(a). While the results at "I = 50, Coarse" for lower layer temperature prediction and "I = 100, Coarse" for upper layer temperature prediction agreed better.

Comparisons between the predicted vertical temperature profiles without local refinement at the fire source and experimental data are shown in Figs. 6(b) and 7(b). Again, for the predictions at corner stack and door center, simulations with higher number of iterations and grid density agreed well with experimental data. On the contrary, local refinement at the fire source could not improve the agreement level when the fire was positioned adjacent to the rear wall.

From both the experiment and model predictions, it is found that the average upper layer temperature is higher where the fire was centrally located. The likely reason is that the effect of radiation is important and hence neglecting radiation in these models would be of greater significance.

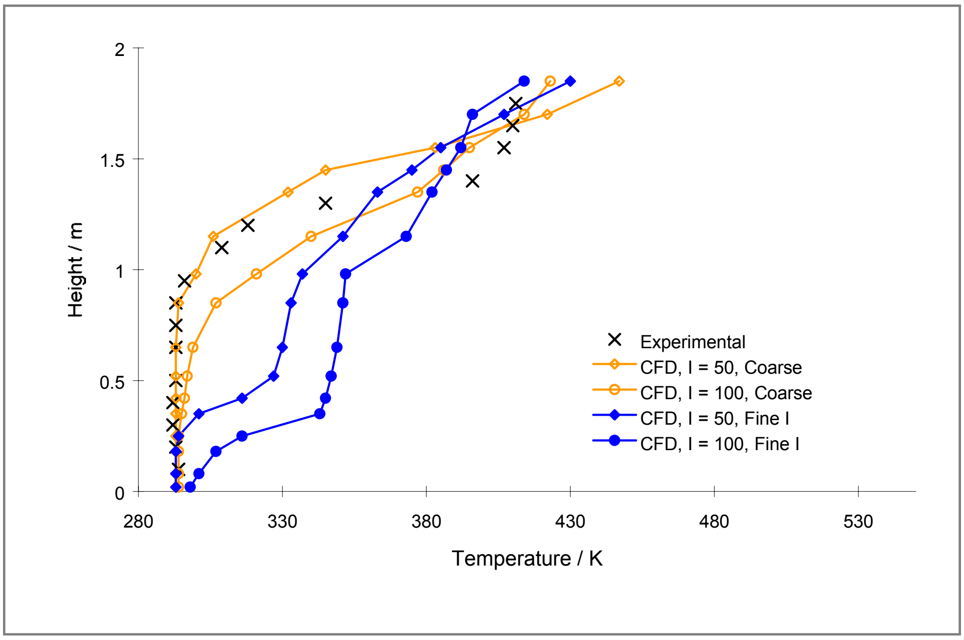


(a) With cell refinement at fire source

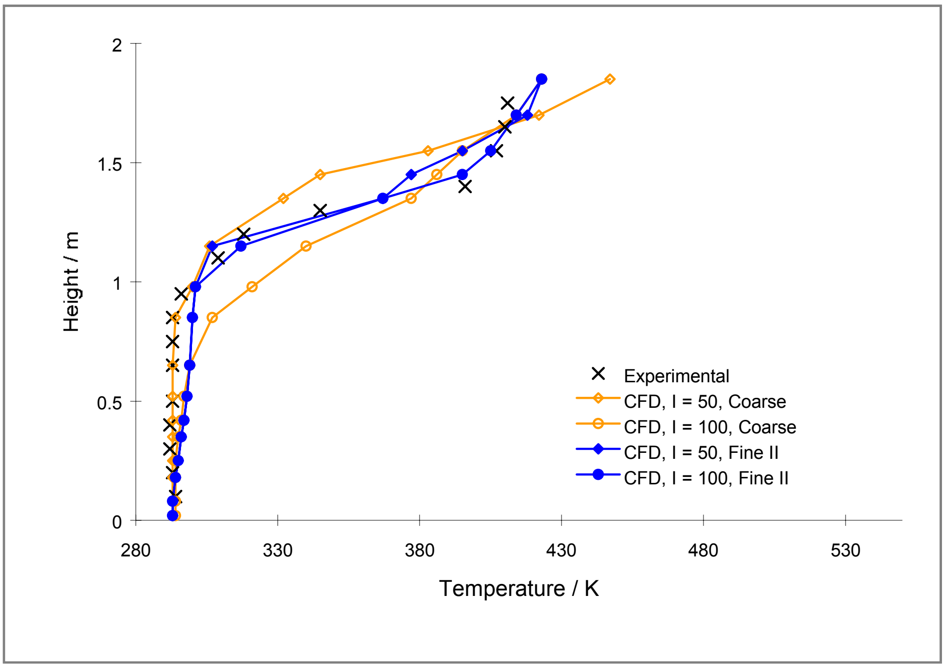


(b) Without cell refinement at fire source

Fig. 6: Predicted and measured vertical temperature profiles at corner stack for fire located at rear wall



(a) With cell refinement at fire source



(b) Without cell refinement at fire source

Fig. 7: Predicted and measured vertical temperature profiles at door center for fire located at rear wall

4. DIFFERENCES BETWEEN MEASURED AND PREDICTED TEMPERATURES

There are many reasons for the differences between predicted and measured temperatures. Experimental results might not be reliable. There are limitations in solving the mathematical equations numerically.

- Experimental uncertainties

Numerical simulations may give unsatisfactory results if the model is not used properly. The model uncertainty is primarily due to the assumptions made by the model, and can be quantified as a result of the validation process. Unfortunately, very few full-scale burning tests had been carried out for verifying CFD codes. There are many problems in comparing the CFD predicted results with data on full-scale fire experiments. Those tests are not performed specifically for validating CFD codes. All full-scale fire test data available have some experimental uncertainties. Discrepancies between model predictions and experimental data might be, at least partly, due to measurement errors. Even for a well-conducted experiment, the temperatures measured in two evidently identical tests could absolutely have a difference up to 300 °C [3].

Models of bared bead, single-shielded aspirated and double-shielded aspirated thermocouples were developed to characterize the uncertainties associated with temperature measurements in fire environments [8]. The models indicate that thermocouples respond differently to changes in effective surroundings temperature in a hot upper layer than in a relatively cooler lower layer in a room fire. Although, errors can be reduced by the use of an shielded aspirated thermocouple, they cannot be eliminated totally.

- Numerical uncertainties

Numerical uncertainty results from the influence of discretization and iterative convergence errors. This uncertainty cannot be eliminated, but only minimized or bounded in simulation. The error is created by those terms truncated in the Taylor series representation of derivatives, or introduced by the iterative solution process. These discretization errors have definite magnitude and an assignable cause, and can all be cast in terms of two parameters – the grid size and the time step size [9].

Grid refinement is the main tool for improving the accuracy of a simulation as the truncated error can be reduced more quickly [10]. Using a finer grid will allow the computational boundary and associated turbulent boundary layer to be modelled more accurately, which may have an important bearing on the level of turbulent mixing and convective heat transfer. It was suggested that relatively large number of grids should be allocated in the direction of height in tunnel fire simulation to account for the rapid changes in velocity and temperature due to buoyancy [11]. Also, finer grids should be assigned at the heat source to resolve the fire chemistry [12]. However, the effect of grid refinement is not very significant outside the combustion zone and its thermal plume [1]. Finer grids might be more desirable but relatively longer CPU time is required. Experienced engineers can deal with that professionally.

By comparing the numerical results with analytical solutions of time-dependent problems [10], longer time step would give inaccurate and unrealistic numerical solution that oscillates about the exact solution. In spite of the fact that the implicit method, which tolerates much larger time steps, gives results that are in reasonable agreement with the exact solution, high solution accuracy can only be achieved with small time steps.

On the other hand, the order of accuracy may keep constant as the grid and time step are reduced beyond a specific threshold. Grid and time-step refinement studies often expose discretization errors and programming errors in boundary conditions, as well as in the underlying partial differential equations discretization [13]. Therefore, systematic refinement of the grid size and time step for specific problems is important in verification testing of CFD codes.

Iterative convergence is as significant as grid convergence. The numerical experiments of tunnel pool fire and room fire were computed in steady state, which required iterations over the entire domain. It was shown that the level of agreement with experimental data was largely improved when higher number of iterations was used.

- Model uncertainties

Apart from experimental uncertainties, detailed models describing turbulent flow field and its coupling with chemical kinetics, such as soot generation modelling, are still at the stage of intensive development and validation [14]. Therefore, temperature predictions were not sensitive to soot fraction, and this inefficiency is believed to contribute to the discrepancy.

5. DISCUSSION

- Behaviour of side-wall located plume

A series of numerical experiments were carried out. CFD results were compared with the room fire experiments. According to the results of room fire simulation, special attention should be paid on the mesh generation for those cases in which the fire is located adjacent to the walls. Wall or corner fire can be characterized by merging of flame and plume from the source or rising along the wall. The flame tip height is higher than that in free boundary. Under natural convection, surrounding air will flow toward the reactive zone of the pool fire, and accelerate progressively upward along the vertical burning wall due to buoyancy. Because of the interaction between the vertical burning wall and the pool fire, entrained air is flowing almost horizontally towards the vertical wall with very low upward velocities. After that, the buoyant gas column is broadened. The combustion process changes due to the reduction of air entrainment and the pressure difference between the wall side of flame and the free boundary side [15]. It is difficult to include instability due to buoyancy and the transition to turbulence at the lower levels in the numerical simulations [16].

- Quality of validation data

Fire modelling via CFD needs to include additional fire science modules for combustion, radiation, soot production, etc. The complexity of the chemistry and physics involved in fire simulation continues to pose challenges, not only in terms of the CFD model, but also in producing good quality validation data from real-fire experiments. Several cases were highlighted by Grant and Lea [17], in which there are still substantial gaps in the availability of good quality fire data for CFD validation. However, most of the experimental data, particularly with respect to full-scale fires, concerns only detailed temperature measurements; velocity and mass-flow measurements are made only to a much lesser extent. More measurements for other parameters are needed for a good quality validation exercise.

Results of uncertainty and sensitivity analyses should form an integral part of fire field model to provide data to the engineers, fire-code regulator and the others to assess the tolerance of the design for fire safety. Full documentation of real-life fire safety and fire hazard records in buildings and also the results of fire field model analyses of the design fire scenarios based on the specific design fire loads are helpful for future reference and use [18].

6. CONCLUSION

CFD results agreed reasonably well with the measured fire data in the room fire validation test in the tunnel pool fire experiment. Larger number of iterations and finer grid configuration can give more accurate simulation results. Further, the effect of cell refinement at the fire source is significant for simulations with the fire located adjacent to the wall.

Actual updating of the technique of fire field modelling is very important for satisfying the performance requirements for fire safety engineering, such as areas to develop modelling treatment of turbulent flow field and coupling with chemical kinetics. The general acceptance of which as a tool for practical applications would benefit greatly from a systematic effort in their "validation", and this will rely on the establishment of a coherent database of test data.

ACKNOWLEDGEMENT

This project is funded by a part-time PolyU student research grants under account number G-V644.

REFERENCES

1. S. Kumar, N. Hoffmann and G. Cox, "Some validation of JASMINE for fires in hospital wards", Lecture notes in engineering 18, Numerical simulation of fluid flow and heat/mass transfer processes, Springer-Verlag, New York, pp. 159-169 (1986).
2. R.N. Mawhinney, E.R. Galea, N. Hoffmann and M.K. Patel, "A critical comparison of a PHOENICS based fire field model with experimental compartment fire data", Journal of Fire protection Engineering, Vol. 6, No. 4, pp. 137-152 (1994).
3. A.N. Beard, "Fire models and design", Fire Safety Journal, Vol. 28, No. 2, pp. 117-138 (1997).
4. A.J. Grandison, E.R. Galea and M.K. Patel, Fire modelling standards/benchmark report on phase 1 simulations, Paper no. 01/IM/72, CMS Press (2001).
5. A.J. Grandison, E.R. Galea and M.K. Patel, Fire modelling standards/benchmark report on phase 2 simulations, Paper no. 01/IM/81, CMS Press (2001).
6. PHOENICS Version 3.2, Concentration, heat and mass, CHAM Co., London, UK (1999).
7. K.D. Steckler, J.G. Quintiere and W.J. Rinkinen, Flow induced by fire in a compartment, NBSIR 82-2520 (1982).
8. L.G. Blevins and W.M. Pitts, "Modeling of bare and aspirated thermocouples in compartment fires", Fire Safety Journal, Vol. 33, No. 4, pp. 239-259 (1999).
9. C.J. Freitas, "The issue of numerical uncertainty", Applied Mathematical Modelling, Vol. 26, No. 2, pp. 237-248 (2002).
10. H.K. Versteeg and M. Malalasekera, "Appendix A - Accuracy of a flow simulation", An introduction to computational fluid dynamics - The finite volume method, pp. 240-242 (1995).
11. H. Xue, J.C. Ho and Y.M. Cheng, "Comparison of different combustion models in enclosure fire simulation", Fire Safety Journal, Vol. 36, No. 1, pp. 37-54 (2001).

12. G.H. Yeoh, R.K.K. Yuen, S.M. Lo and D.H. Chen, "On numerical comparison of enclosure fire in a multi-compartment building", Fire Safety Journal, Vol. 38, No. 1, pp. 85-94 (2003).

13. W.L. Oberkampf and T.G. Trucano, "Verification and validation in computational fluid dynamics", Progress in Aerospace Sciences, Vol. 38, No. 3, pp. 209-272 (2002).

14. V. Novozhilov, "Computational fluid dynamics modeling of compartment fires", Progress in Energy and Combustion Science, Vol. 27, No. 6, pp. 611-666 (2001).

15. W. Takahashi, H. Tanaka, O. Sugawa and M. Ohtake, "Flame and plume behaviour in and near a corner of walls", Fire Safety Science Proceedings of Fifth International Symposium, pp. 463-474 (1997).

16. H.Y. Wang and P. Joulain, "Modeling on the interaction of the turbulent diffusion flames between a vertical burning wall and a pool fire", Fire Safety Science Proceedings of Fifth International Symposium, pp. 475-486 (1997).

17. G. Grant, and C.J. Lea, "A review of the modelling of fire in buildings and structures", Health & Safety Laboratory, Report no. CM/01/01 (2001).

18. K.T. Yang, "Role of fire field models as a design tool for performance-based fire code implementation", International Journal on Engineering Performance-Based Fire Codes, Vol. 1, No. 1, pp. 11-17 (1999).