International Journal on Engineering Performance-Based Fire Codes, Volume 5, Number 3, p.54-68, 2003

TREATMENT OF FIRE SOURCE IN CFD MODELS IN PERFORMANCE BASED FIRE DESIGN

R. Yau, V. Cheng and R. Yin
Ove ARUP & Partners Hong Kong Ltd, Hong Kong

(Received 18 December 2002; Accepted 13 May 2003)

ABSTRACT

With the increasing needs of performance-based fire design on advanced modern buildings, fire field modeling (Application of Computation Fluid Dynamics) has become more and more attractive as a critical design tool to meet this kind of requirements. Traditionally, instead of complex combustion model, the fire source was just taken as a volumetric heat source with prescribed heat release rate in practical fire simulation. However, it was always argued about the accuracy of CFD results with such simplification. This paper presents the comparison of simulation results of fire field models with volumetric heat source method and submodels of combustion model and radiation model, altogether with the available experimental data under different fire scenarios. Some guidance and discussion of important issues related to the application of fire field modeling on performance-based fire design, which needed special consideration and proper practical application, were provided in this paper.

1. INTRODUCTION

With the adoption of performance based fire safety engineering principles at an early stage of architectural design, the architects and designers were freed from the constraints imposed by the historical prescriptive fire safety approach, to create more cost-effective, innovative and user-friendly design of building. For the systematic design of an effective fire protection system, the fire development should be properly understood and the key elements for prescribed fire scenarios, such as smoke transport processing, temperature distribution should also be clearly identified. Due to the close coupling of these processions with any complex shape of enclosure in nature, the fire zone model is not sufficient for the practical design. The fire field modeling technique, or application of CFD has been developed to offer a means of optimizing fire safety engineering solutions for innovative architectural design.

Fire field model (CFD model) is based on numerical solutions to a series of partial differential equations associated with conservation of mass, momentum, energy and species. The unique advantage of the fire field models are that it could provide the potential to study the complicated problems encountered in fire safety engineering performance assessment of smoke and hot air movement for nearly all kinds of building fire scenarios and the ability to present the thermal environment parameters, such as smoke temperature, velocity magnitude, smoke concentration at any time and locations, providing nearly the same phenomena as in a real fire.

Although fire field model has been used as an access tool for fire safety engineering design for more than two decades [e.g. 1], there are continuous validations and verifications on the fire field model [e.g. 2-6], in which the fire source was taken as a volumetric heat source. Therefore, some arguments still exists on the reliability of CFD simulation results due to some simplifications. One of the major concerns is on the volumetric heat source, which is used to describe the fire instead of the combustion model.

The paper focuses on the proper application of CFD modeling in fire safety design. Comparing the predicted results of different treatment of fire source in CFD model with experimental data, a brief description for CFD fire modeling is provided in this paper. The advantage, capacity and limitation of different CFD fire modeling are highlighted. Results presented in this paper are useful to the further application of fire field model in the practical fire engineering approach.

2. CHARACTERIZATION OF FIRE SOURCE IN FIRE FIELD MODEL

For building fires, the unwanted fire generated a large quantity of heat. The heat released from combustion materials gave the surrounding air up to higher temperature, which can be in the range of hundreds of degrees Centigrade. Consequently, the high temperature provided the strong buoyancy that drove the fire and its productions (smoke) spreading from the fire source in the buildings. The concerned parameters in the design of fire protection system, such as hot air temperature, smoke concentration, smoke transport procession inside the building; all have strong relations to the fire strength. Therefore, fire strength or heat release rate of fire source was one of the most important parameters that should be determined in the fire engineering design. It was the key element to affect the fire phenomena in buildings and determine all the design of fire protection system. Consequently, there was a need in CFD modelling for properly estimating the fire load (or fire heat release rate) in the design fire scenario.

There are two ways to determine the fire source in fire field model:

- Combustion model
- Volumetric heat source

The first one is the application of combustion submodel. The input parameters in the CFD modelling are the consumption rate and physical property of combustible materials. Fuel and air mixing process, fuel combustion process, combustion productions generation and heat release rate were calculated in the programs. It is quite complex and difficult to deal with, since the chemistry reaction in burning materials, mixing of air and fuel due to turbulence are needed to simulate in the model, many assumptions and practical parameters are included in the combustion model to close the equations [7].

The second one is the most commonly used one, in which the fire source is directly considered as a prescribed volumetric heat source. The only parameters needed are the heat release rate and fire volume. It is simple and easy to handle. Nevertheless, there still have some arguments for the prescription of the fire itself [8]: fire source might be characterised by a known heat release rate but then associate that heat release rate with an inappropriate fuel area. Another problem can occur when unsuitable "pre-determined" volume can yield gas temperature in the source that is much too high or too low, even higher than the adiabatic flame temperature [9].

However, such problem can be solved with proper attention to the characteristics of buoyant fire source. First, the characteristic for the building fire can be represented with the non-dimensional heat release rate Q^*[8]:

$$Q^* = \frac{Q}{\rho_0 C_p T_0 D^2 \sqrt{gD}} \tag{1}$$

where Q is the heat release rate of the fire; D is a characteristic fuel dimension; ρ_0, C_p, and T_0 are the ambient air density, air specific heat and temperature respectively; and g is the acceleration due to gravity.

It has been pointed out [8] that the range of Q^* ($0.1 < Q^* < 2.5$) presented the characteristic of typical building. The relations of fire strength Q and dimension D for typical building fire must be located within the range. More recently, another relationship between heat release rate and its dimension for buoyant fire was derived [4]:

$$\ln Q = 2.69 \ln P - 3.83 \tag{2}$$

where P is the perimeter of fuel.

Following the above two equations, the fuel area of combustion materials with prescribed heat release rate could be properly determined.

The second problem is the volume of fire source. Unit mass of fuel burnt with air either as a diffusion or a premixed flame will release similar quantities of heat, but the resulting volumes of heated gas are very different. Too large or too small volume could cause wrong simulation result. With calculated fuel area from equations (1) and (2), the flame height is the only parameter needed for fire volume.

For building fire, it is obvious the diffusion flame fire with large volume of heated gas, comparing the mean energy release rate per unit volume of a typical fire with those of man-made combustion system [10]. The energy intensities of some combustion system are listed in Table 1 [11].

Table 1: Heat release intensity for different combustion system

Combustion system	Heat release intensities (MWm^{-3})
Aero gas turbine (take-off)	1500
Gas central heating boiler	200
Gasoline engine	100
Gas-fired fluidised bed	40
Pulverized coal combustor	10
Fire	0.5

It was clear that the heat release intensity of typical building fire is at the magnitude of 0.5 MWm^{-3} and it could be treated as a criteria to judge whether the presumed fire volume was acceptable or not.

For typical diffusion flame fire, the heat released from the fire was concentrated on the continuous region of diffusion flame [12]. The continuous diffusion flame height can be calculated from the following equation [13]:

$$L = \left(\frac{Q}{\rho_\infty C_p T_\infty \sqrt{g}}\right)^{2/5} \tag{3}$$

With the known flame height and fuel area, the fire volume could be obtained.

3. NUMERICAL SIMULATIONS

The commercial CFD code STAR-CD [14], which has been validated for fire simulation before [3], is applied in this paper to simulate building fire with the above two different characteristics of fire source in fire modelling. The conservation of mass, momentum and energy equation concerned and the associated numerical algorithm are described in the literature, and are not repeated here. The modified k-ε turbulence model [15] is applied in the present study.

A building-fire experiment with three rooms connected [16] was selected for the simulation. The layout was shown in Fig. 1. The height of each room was 2.5 m. The door that connected each room and corridor was 0.8 m (W) x 2.0 m (H). A sandbox burner was placed at the center of Room 102. Propane was provided with constant flow rate that liberate 300 kw of heat. The plan view of CFD model grid mesh was illustrated in Fig. 2. The space was evenly divided into 25 parts along vertical direction.

For the simulations with volumetric fire source method, three different volumes of the heat source were applied for comparison, so as to test the impact on the heat source volume to the precision of predicted CFD results. The volumes of heat source were 0.2 m (L) x 0.3 m (W) x 0.2 m (H), 0.7m (L) x 0.6 m (W) x 0.6 m (H), and 1.0 m (L) x 1.0 m (W) x 1.2 m (H), which represent the small, middle and large size of heat source respectively. The total heat release rate is 300 kw. 35 percent of heat radiated to the surrounding [17] and was not considered in the present study.

For the middle size, the diameter of fuel and height of flame were calculated from equations (2) and (3). The heat release intensity of this size is 1.2 MWm^{-3}, which is the same magnitude as mentioned in Table 1.

The computation is started on a personal computer with CPU Intel P4 2.4 GHz. The total CPU time used is about 6.1 hours.

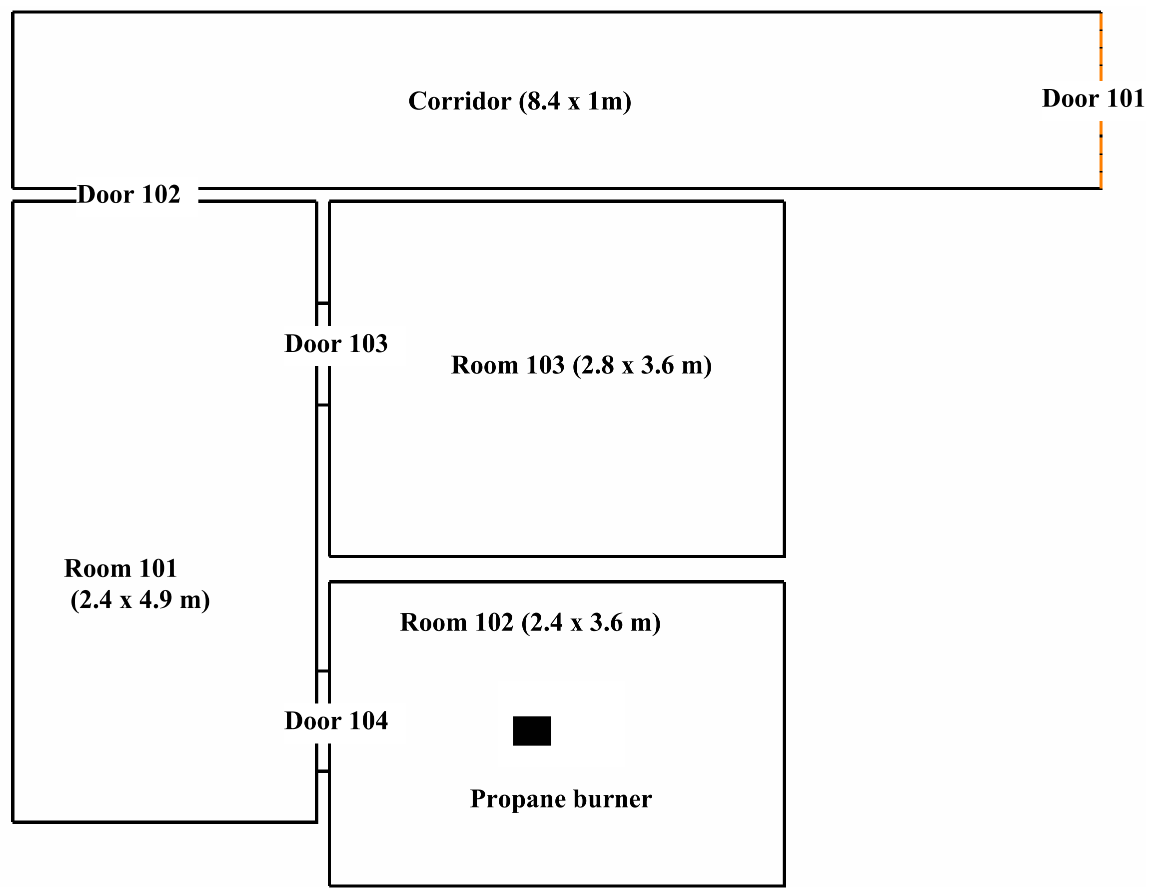


Fig. 1: Layout of the experimental facility conducted by Luo & Beck (1994)

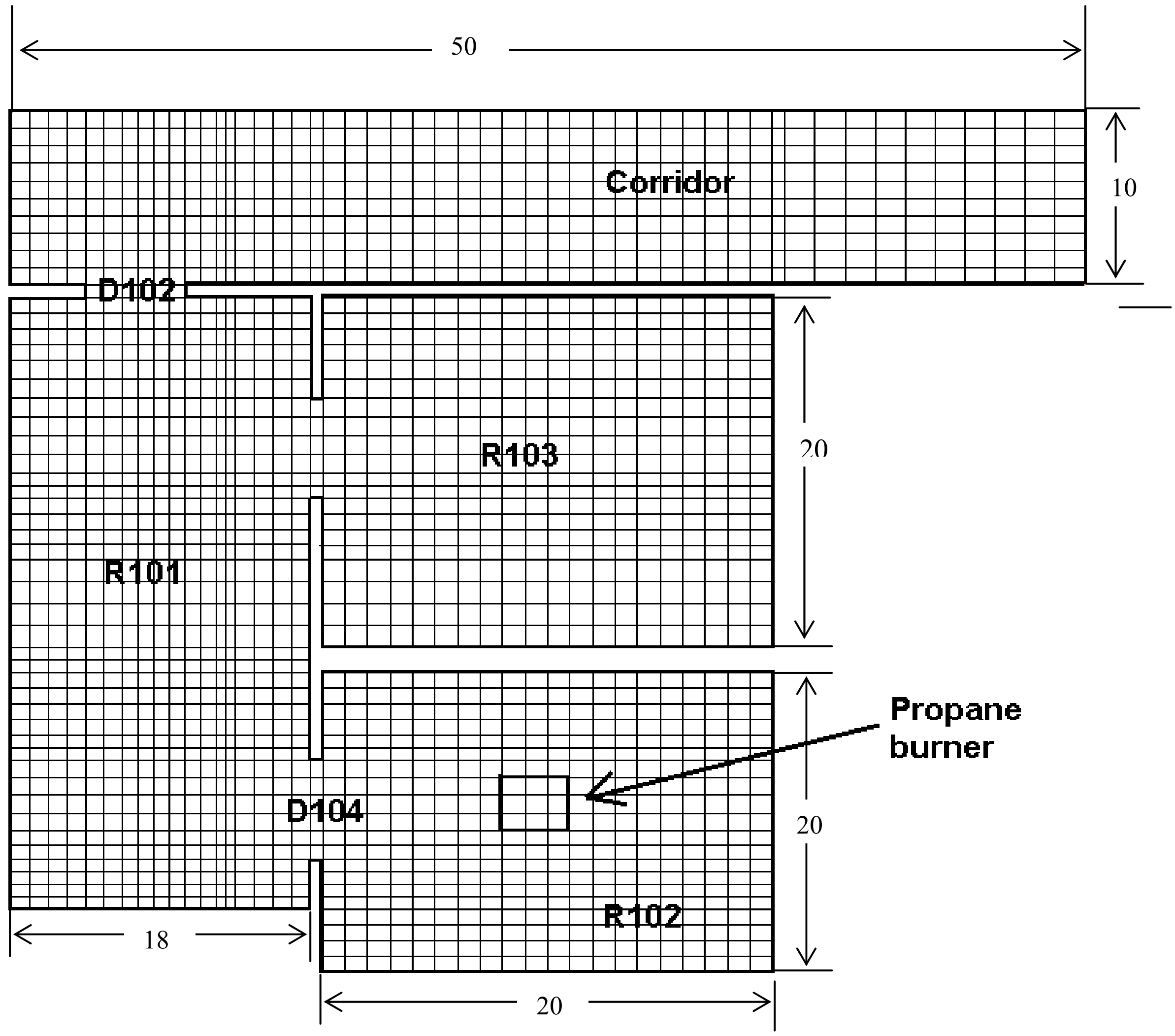


Fig. 2: Mesh layout for the simulations

For the combustion model, the three-step turbulence-controlled Eddy Break Up (EBU) model [24] was used to simulate gas combustion procession. In this model, kinetics reaction time scale was assumed much smaller than the gas fuel micro-mixing, and the combustion reaction rate mainly depended on the mixing rate of gas fuel (propane) and surrounding air. The combustion reaction for propane was:

$$C_3H_8 + 1.5O_2 \longrightarrow 3CO + 4H_2 \qquad (4)$$

$$H_2 + 0.5O_2 \longrightarrow H_2O \qquad (5)$$

$$CO + 0.5O_2 \longrightarrow CO_2 \qquad (6)$$

The burning rate of propane was given by:

$$R = -\frac{\rho\varepsilon}{k} A_{ebu} \min[m_F, \frac{m_o n_f M_f}{n_o M_o}, B_{ebu} \frac{m_p n_f M_f}{n_p M_p}] \quad \text{kgm}^{-3}\text{s}^{-1} \qquad (7)$$

where A_{ebu}, B_{ebu} are dimensionless empirical coefficients, taken as 4 and 0.5 [24] in the present study; ρ is the density of mixing gas; k is the turbulence energy; ε is the turbulence energy dissipation; M_f is molecular weight of propane; M_o is molecular weight of oxygen; M_p is molecular weight of combustion products; m_f is mass fraction of propane; m_o is mass fraction of oxygen; m_p is mass fraction of combustion products; and n_i is stoichiometric coefficient of species in the chemical reaction.

The supply rate of propane was set at the same rate as experiments and then the fire size was calculated during the simulation.

The radiation sub-model, known as 'Discrete Transfer Method' [18] was also applied in the simulation. The radiation effect was represented by many directed beam with pre-determined directions between two boundary walls. As each beam was traced inside the computational domain, its radiant intensity I in the direction Ω is absorbed and incremented by the intervening material according to the following equation:

$$\frac{dI}{ds} = -(k_a + k_s)I + \frac{k_a E_g}{\pi} \int_{4\pi} p(\underline{\Omega}, \underline{\Omega'}) i(\underline{\Omega'}) d\underline{\Omega'} \quad (8)$$

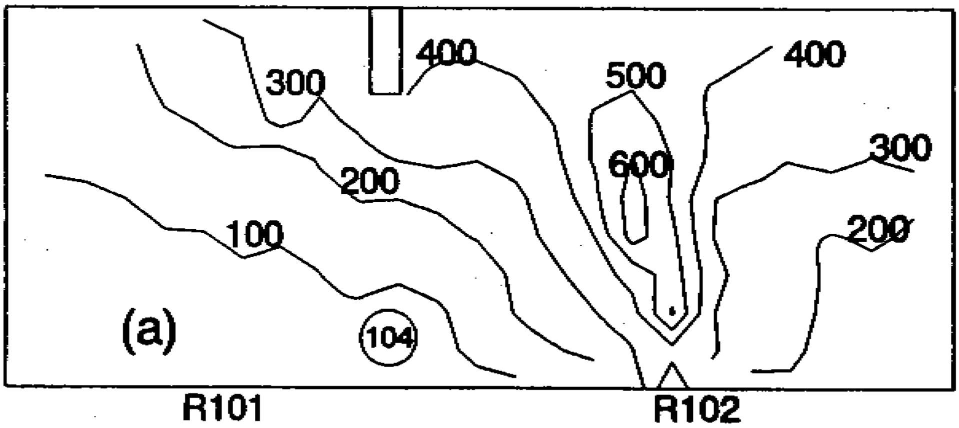
where s is distance in the $\underline{\Omega}$ direction; E_g is black body emissive power of the material at temperature T_g; K_a is absorption coefficient; K_s is scattering coefficient; $P(\underline{\Omega},\underline{\Omega}')$ is probability that the radiation incident in direction $\underline{\Omega}'$ will be scattered to within $d\underline{\Omega}$ of $\underline{\Omega}$.

The heat absorption or emission for a cell through radiation was then calculated by summing the change of intensity of each beam passing through the cell.

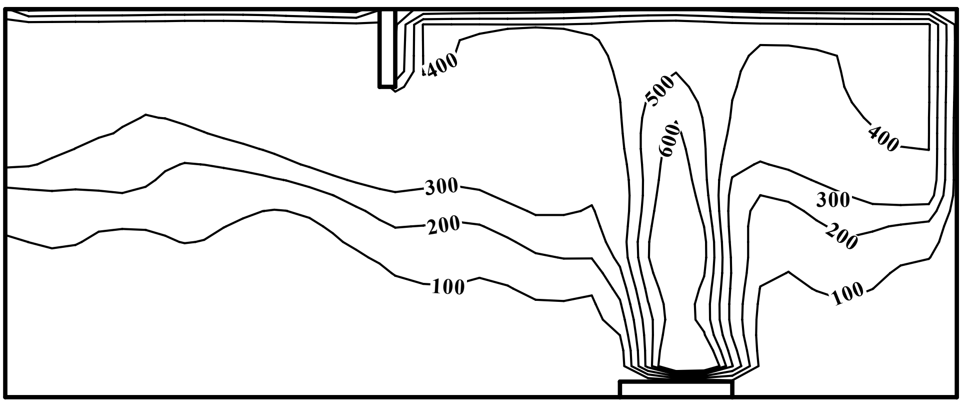
The simulations were carried out in transit mode for 20 minutes and the time step was 0.5 s. The results were considered as steady state solution, just as conducted in the experiments.

The calculation is carried out with the same computer and the CPU time used is about 15.8 hours, which is nearly twice as the time cost with heat source.

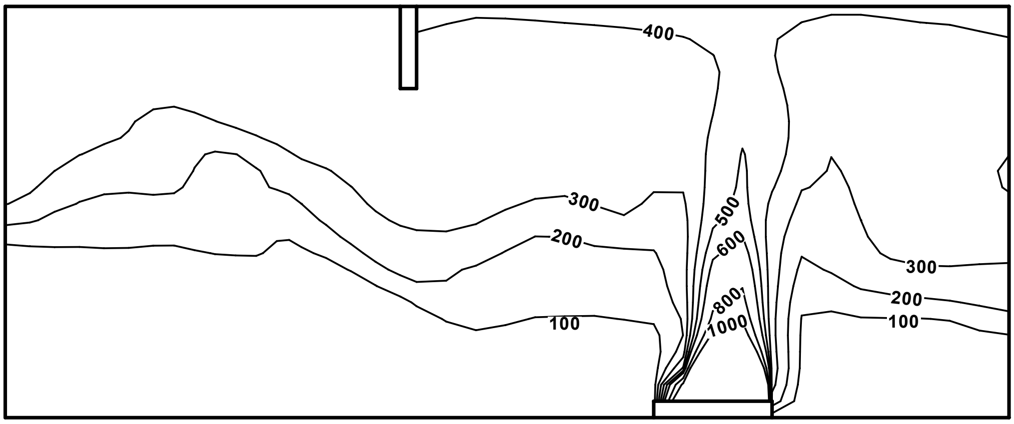
The section view of predicted and measured temperature distribution inside the Room 101 and 102 was shown in Fig. 3. The CFD results with different fire volumes were different from each other at the region of fire plume: for small fire volume case, smaller volume means higher heat intensity of fire source, resulted in higher temperature at the fuel region, which was much higher than the experimental data, even higher than 1000 °C in some locations. Meanwhile, the temperature gradient of hot smoke is larger than experimental result due to higher temperature of smoke penetrated to the upper smoke layer from the fire source.



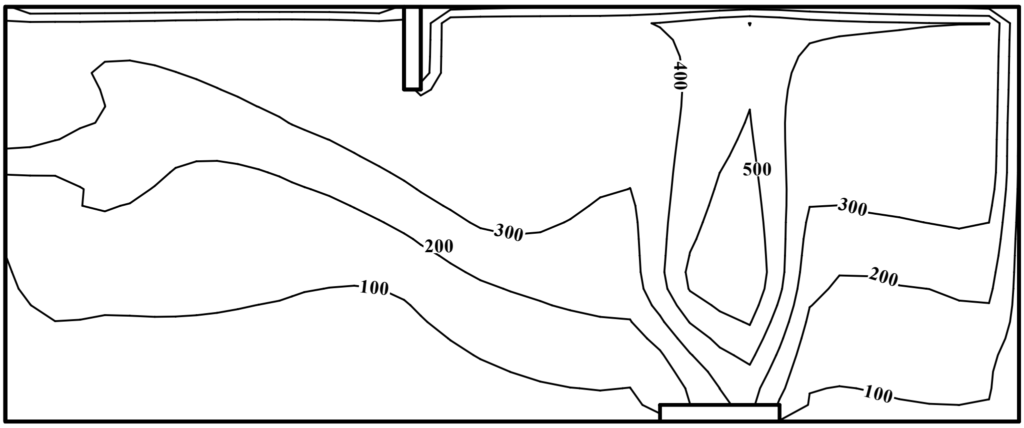
(a) Experimental result



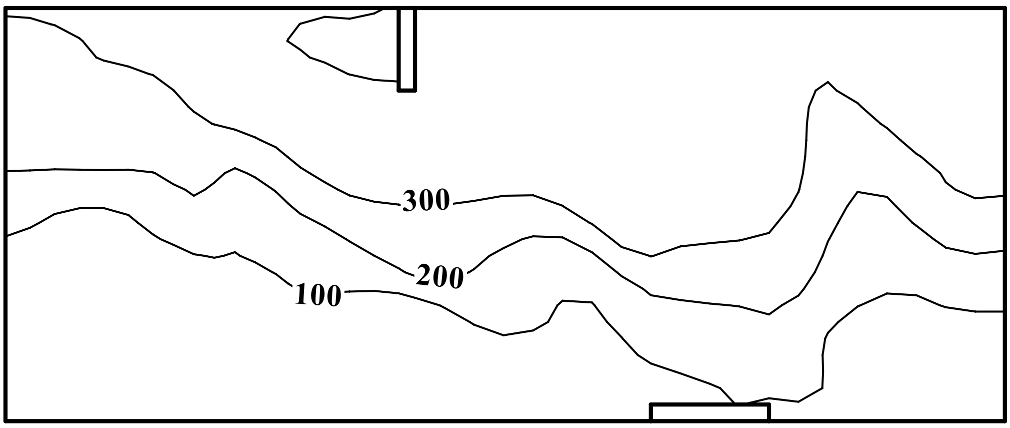
(b) Combustion model



(c) Small volumetric heat source



(d) Middle volumetric heat source



(e) Large volumetric heat source

Fig. 3: Temperature distribution in Room 102 and 101 across the door (°C)

On the contrary, the larger volume of fire source caused much lower temperature at the flame region. No particular higher temperature zone was found in the flame region as a result of relative strong convection effect.

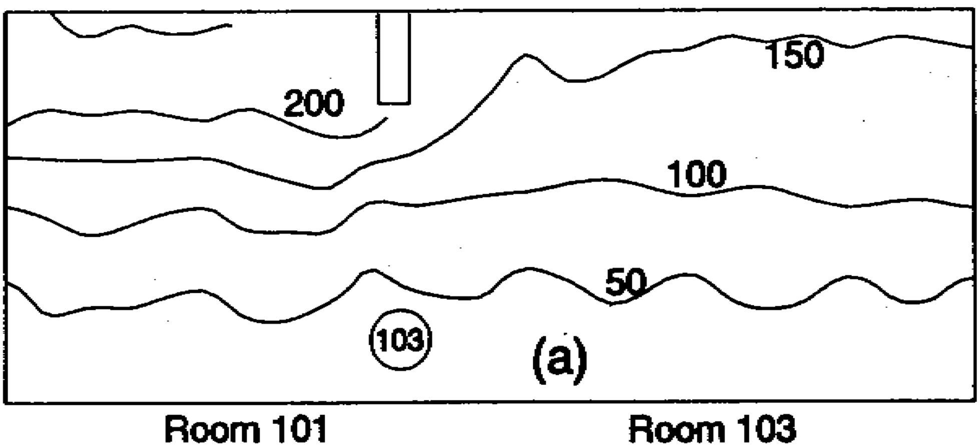
Meanwhile, CFD model with middle volume of fire source, which is determined by the criteria described above, could achieve results that agreed well with the experimental results in most area. At the region of fire flame and ceiling, the predicted temperature was a little underestimated. It might be caused by the even distribution of heat intensity inside the fire source and unconsidered radiation effect to the air. For combustion model, the predicted temperature gives good agreement with the experimental result at the region near the fire flame.

Fig. 4 showed the temperature profiles in Room 101 and Room 103 at the centreline of Door 103. For volumetric heat source method, all the predicted results generally agreed with the experimental result, except that the higher temperature was predicted at high level of smoke layer with small volume heat source.

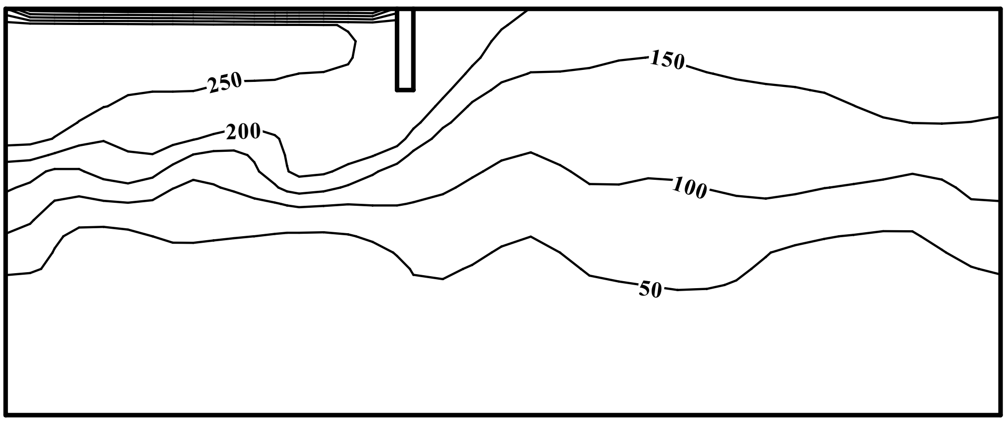
For the combustion model, the temperature contours of 200 °C and 250 °C predicted were a little higher than experimental result.

The measured and predicted temperatures on a vertical section across the centreline of Door 102 were illustrated in Fig. 5. For all simulation results, the temperature contours of 100 °C and 150 °C gave agreement with the experiments results. However, the calculated temperature with combustion model and small volume of heat source were a little over-predicted at both high temperature region (>150 °C) and low temperature region (< 50 °C).

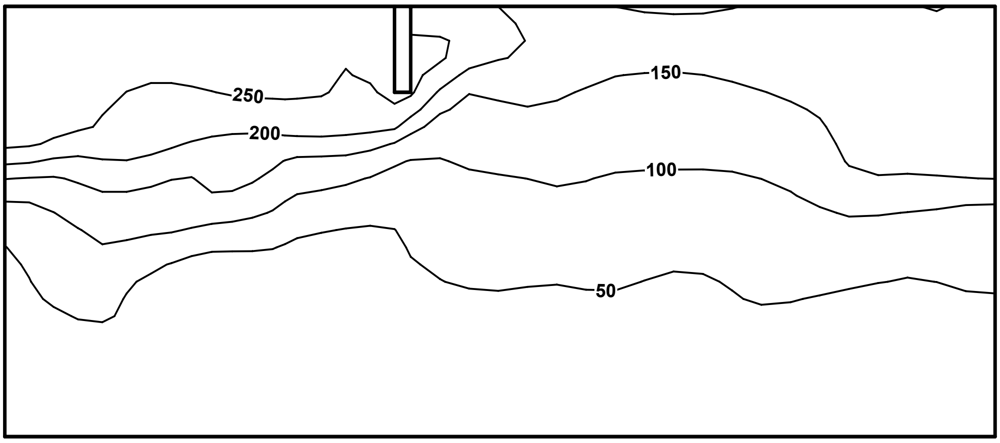
The airflow velocity at two door openings (Door 102 and Door 104) predicted by combustion model and middle volume heat source were compared in Fig. 6. Good agreement was achieved between predicted results and experimental data.



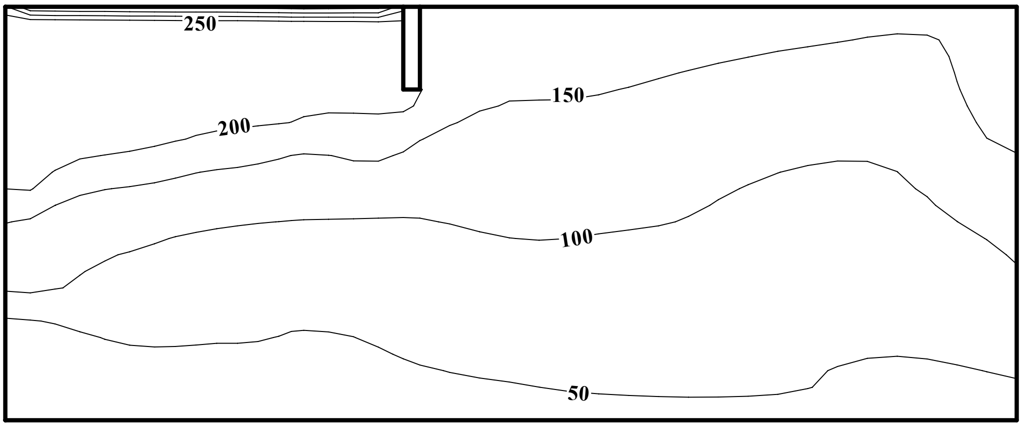
(a) Experimental result



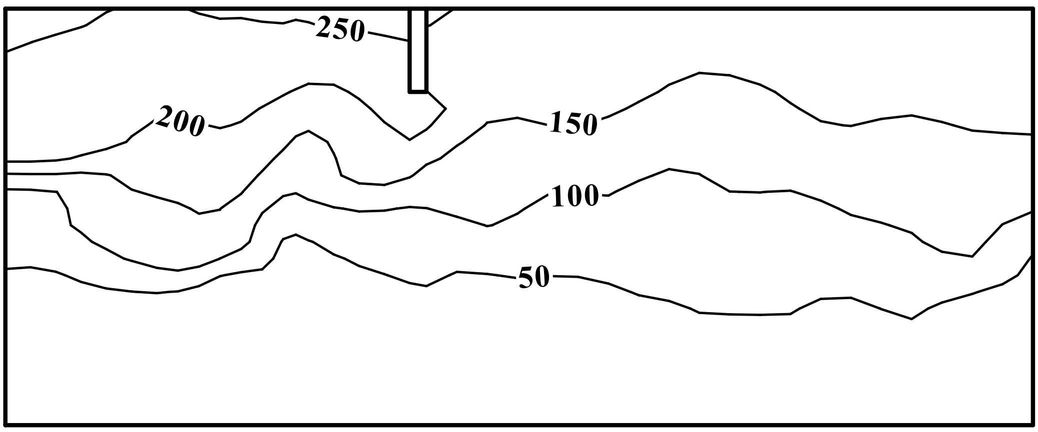
(b) Combustion model



(c) Small volumetric heat source

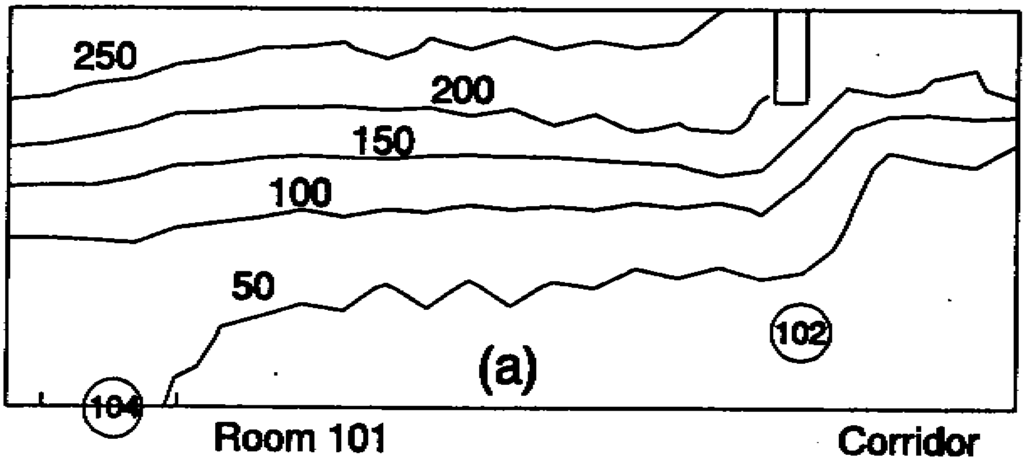


(d) Middle volumetric heat source

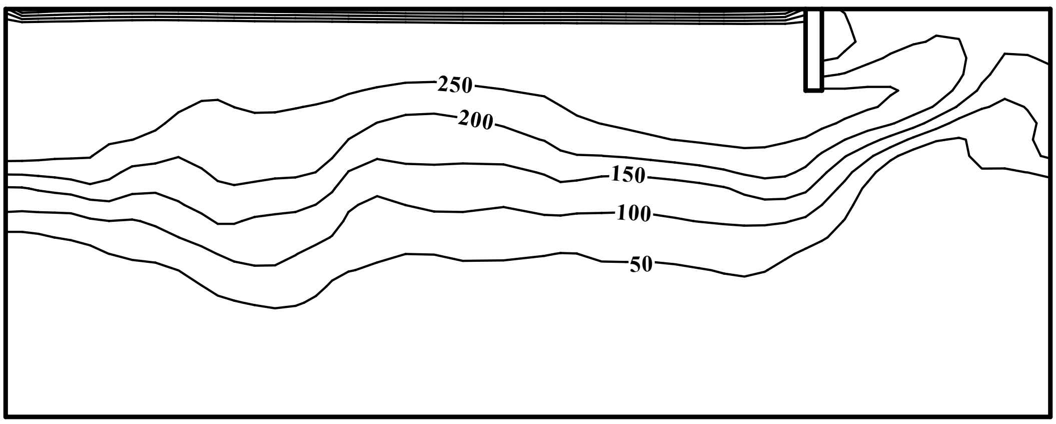


(e) Large volumetric heat source

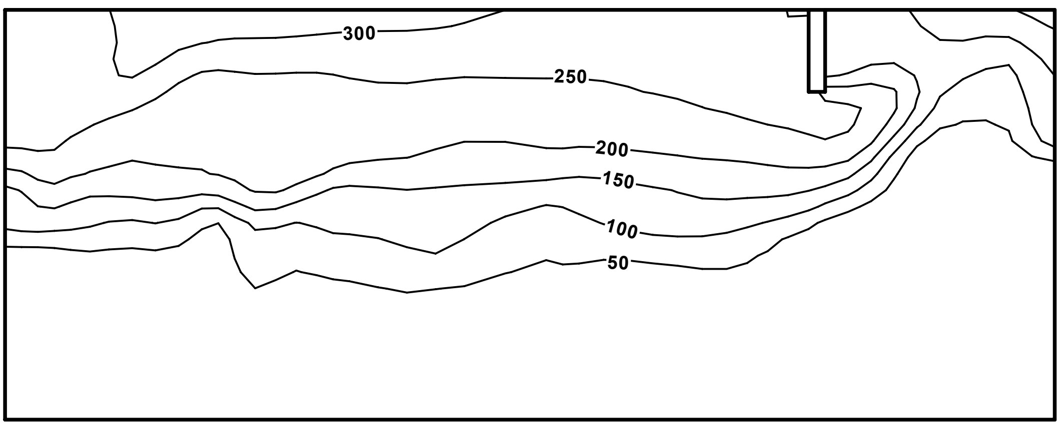
Fig. 4: Temperature distribution in Room 101 and 103 (°C)



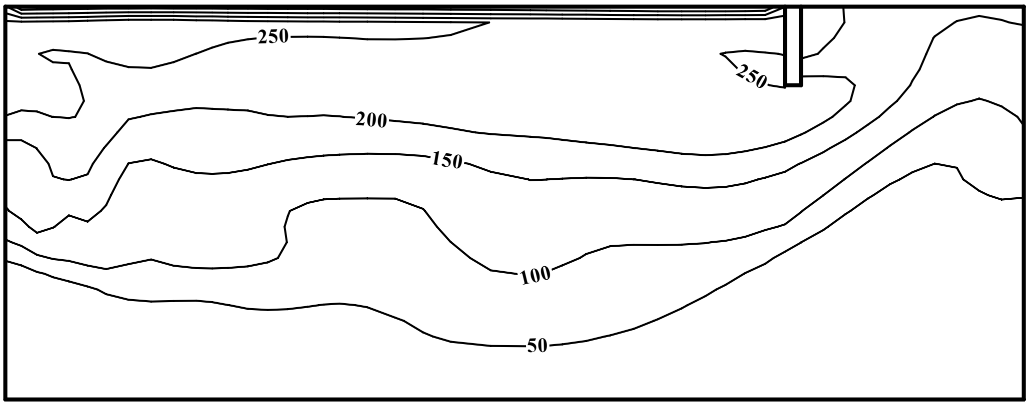
(a) Experimental result



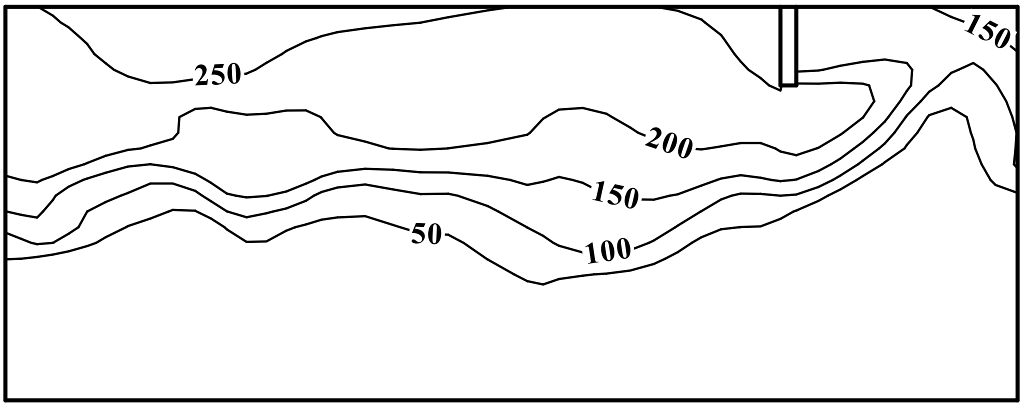
(b) Combustion model



(c) Small volumetric heat source



(d) Middle volumetric heat source



(e) Large volumetric heat source

Fig. 5: Temperature distribution in Room 101 and corridor (°C)

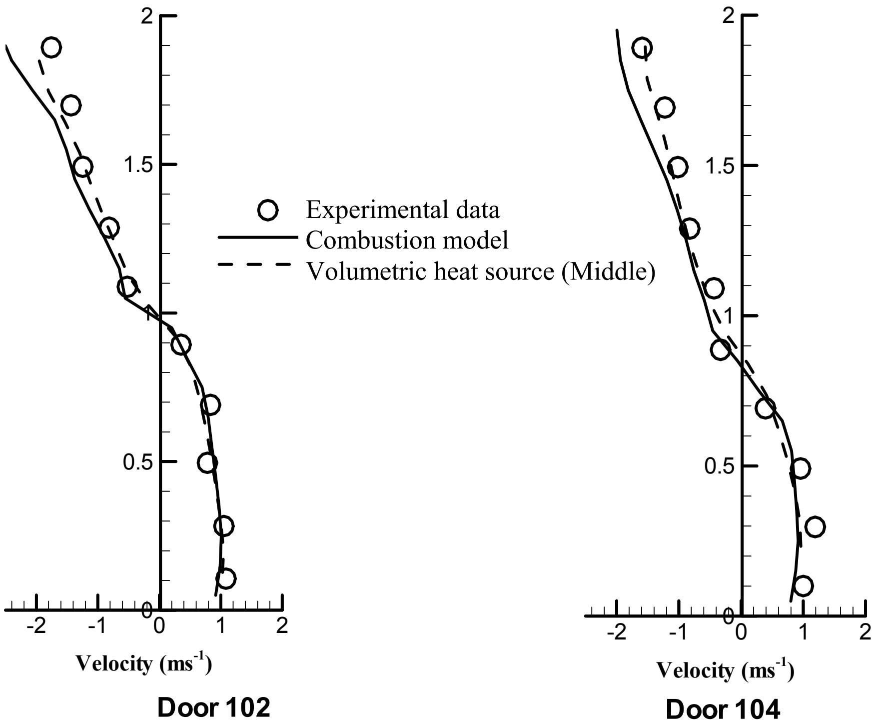


Fig. 6: Comparison of the measured and predicted velocities at Doors 102 and 104

4. DISCUSSION

The comparison shows that the prescribed volume of heat source in fire simulation acts an important role on the accuracy of predicted results, especially at the region near the fire plume. If determined properly, fire field model with volumetric heat source method could predict quite good results in most area of building fire scenarios on both smoke temperature and velocity. Further, the predicted temperature field at the flame region with combustion model has better agreement with measured results than that of volumetric heat source.

However, it is quite difficult to apply combustion model in the practical fire protection system design at the present stage.

First, application of combustion model needs the property and quantity of combustible materials as input parameters. In fire engineering approach, it is quite difficult to determine the exact type and quantity of combustible materials involved in the design fire scenario.

Second, there are thousands of probabilities for the cause of fire accident: the ignition type of combustion materials, fire transportation and development process. All of them could affect the predicted results with combustion model. It is impossible and unrealistic to simulate all the possibilities from engineering approach.

Third, for the combustion model itself, it is obvious the most important submodel, but also the most difficult one in any fire field model. Since it is needed to provide the fire heat load in fire simulation, it is not surprising that this submodel is regarded as the weakest link in the entire fire modelling. Despite the mechanisms for turbulent combustion are largely known, there is quite a controversy on how this phenomenon should be quantified described, and there is an even higher uncertainty of basic intrinsic quantities needed in the submodel, such as turbulent mixing, reaction kinetics and pyrolysis parameters, all are lacking for combustibles normally found in an inhabited buildings [19].

Since the intermediate chemistry in burning materials, mixing of air and fuel due to turbulence are difficult to model in a fire, there is a need for estimating the fire heat load (or fire heat release rate) from a fire in a design fire scenario without the use of combustion submodel, which is taken as the input parameter in most mathematical fire model. Fortunately, there is an general agreement in the field of implementing the performance-based fire codes that the fire load in a given fire scenario can be considered as a design parameter, known as the design fire load, which is in general a function of the materials normally found near a given fire source and the nature of the fire source itself.

In the analysis of a design fire scenario, where the study is mainly concerned with the smoke and toxic-gas movement in buildings or with the smoke filling of rooms, staircases refuges, or effective of smoke extraction system, or in large open space, the fire heat load could be taken as a volumetric heat source to provide accurate results.

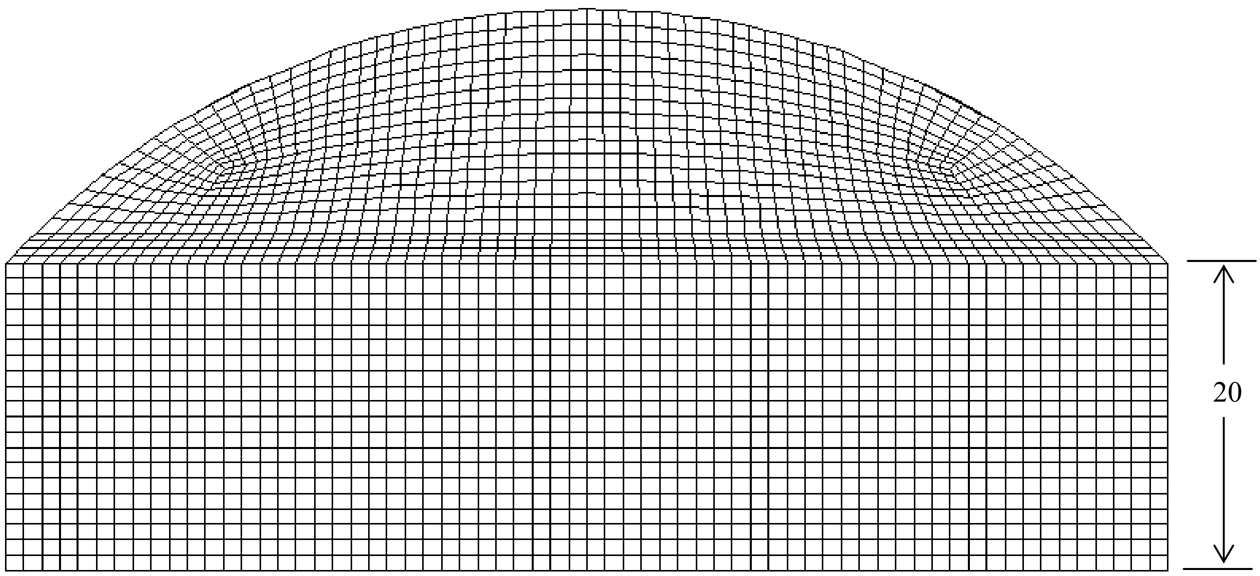
For the radiation model, it is very complex to be applied in the fire study at the present time and still needs further investigation, so it would be better to be switched off without much affecting on the prediction [20]. Even in the confined room fire, with proper consideration of fuel area and volume, it could also be utilized to predict the fire environment except the fire flame region. It is of special interest to note that with such a specification, the application of a fire field model becomes much simplified and easy to control with satisfied prediction.

5. FURTHER EXPERIMENTAL VERIFICATION

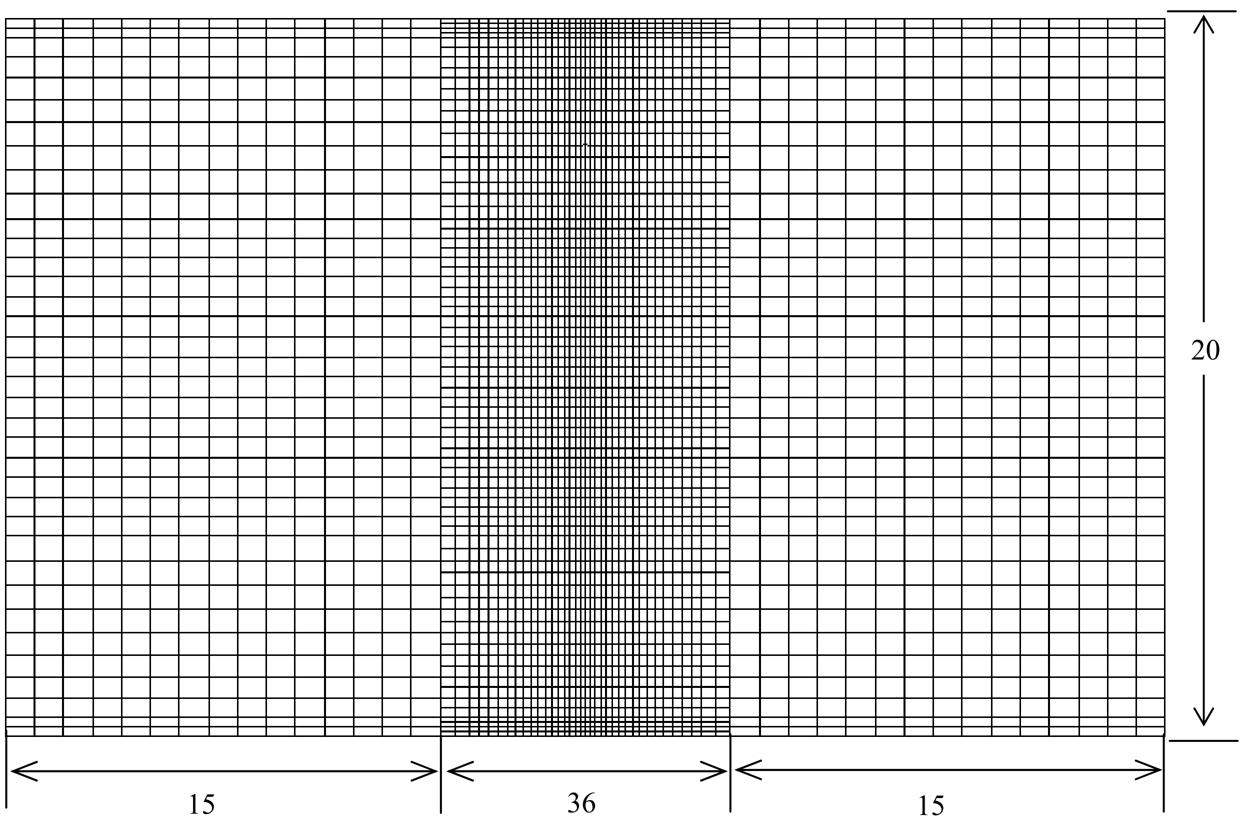
Another experiment was selected to validate the fire field modelling with volumetric heat source on the simulation of fire in large space. Full-scale burning tests were carried out in a hangar of area 45.7 x 73.8 m, with a curved roof ranging from 12.2 m at the bottom of arch to 22.3 m directly over the center of the hangar. Draft curtains subdivide the ceiling into five equal bays approximately 14.8 m in width and 45.7 m in length. The draft curtains extend down to approximately 13.4 m above the floor [21] for experimental studies.

The whole hangar bay was taken as computational domain. Since the hangar was quite large, the non-uniform body-fitted system with local refinement technique was applied in the CFD model. The top and plan view of the grid system was shown in Fig. 7.

The result of test 21, which was the largest fire size in the tests, was employed here to verify the CFD model. The thermocouples were installed below the ceiling at different distance from the centreline of the fire. The pool fire was burnt in a pan of diameter 4.6m x 4.6 m using fuel JP-5. The heat release rate at steady state stage was 33 MW. The ignition time was 80 s. The ambient temperature was kept at 12 °C.



(a) Plan view



(b) Top View

Fig. 7: Grid system for the high bay hangar

With the view of figures published [22], the heat release rate of JP-5 fire can be treated as t^2-fire initially. After 90 s, it was kept at a constant value of its maximum output. The radiation effect was not modelled in the present study, as recommended in SFPE [17], 35 % of total heat release rate was radiated from the fire flame, and the remaining 65 % was used as convective heat release rate for driving the surrounding airflow.

Because of the short contact time between the gas and ceiling surface, solid boundaries such as roof, wall and floor were assumed as adiabatic, given that there was no significant difference of the predicted temperature between the conducting and non-conducting wall boundary conditions [23]. The simulation time was 220 s with time interval 0.5 s.

The volumetric heat source method was applied in the fire modelling, where the area of fuel in the simulation was taken the same as the pan (4.6 x 4.6 m) used in the experiment, which was nearly the same as the one (4.2 m x 4.2 m) calculated with equation (2). The plume height was calculated from equation (3).

Fig. 8 demonstrated the numerical and experimental results of the temperature variation with time. The temperature was measured by the thermocouples located below the ceiling deck, 9.1 m and 12.2 m east from the centreline of fire source. It can be clearly illustrated that the predicted and measured temperature agreed quite well with each other.

The maximum temperature at the different locations below the ceiling was shown in Table 2. The predicted results showed good agreement with the measured data at central and north, south direction. The temperature spikes measured on W1 were recorded in the experiment due to intermittent flames impinging on the thermocouple.

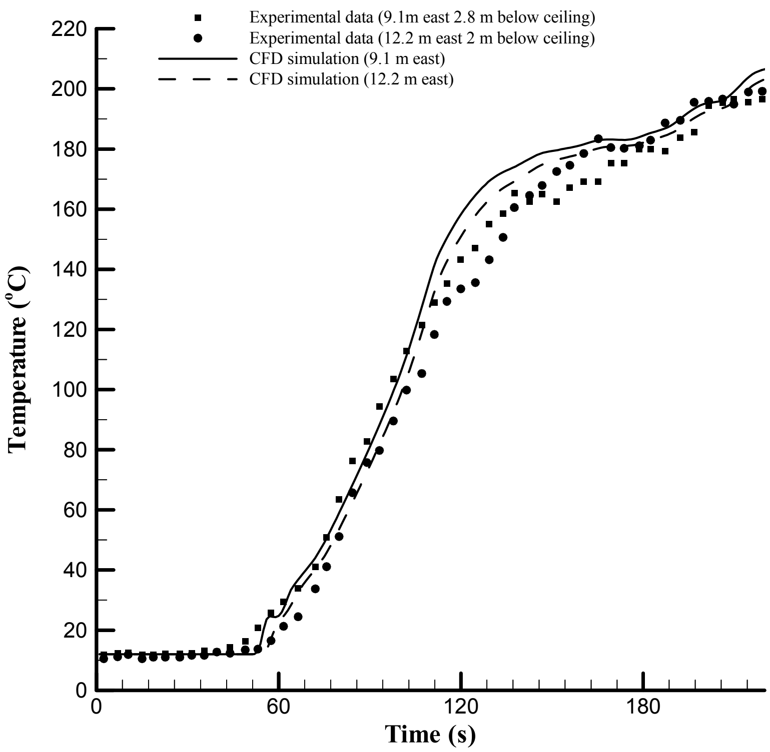


Fig. 8: Temperature variation with time

Table 2: Measured and predicted smoke temperature

	Temperature (°C)					
Location	C	N1	S1	W1	E1	Max. Value
Measured	277	278	249	464	210	277
Predicted	283	277	266	264	280	283

Note that C, N1, S1, W1, E1 represent the location at the centreline of fire source and 3.1 m away from centreline at North, South, West and East direction, 0.2 m below the ceiling.

6. CONCLUSION

Two different methods could be used to characterize fire source in the fire field model, volumetric heat source and combustion model. Both methods could provide good results that agree quite well with the experimental data.

For the CFD model with combustion submodel, it could provide good prediction of temperature distribution in the fire flame and would be great helpful for the forensic fire engineering to recur the fire accident procedure. However, for the application to practical fire safety design, combustion model is quite difficult to handle.

On the other hands, when utilized correctly, the CFD model with proper volumetric heat source could also provide good results in both room fire scenarios and atrium fire scenarios. It could be applied to the studies as following:

- Smoke and toxic-gas movement in buildings
- Smoke filling processing of rooms
- Effective of smoke extraction system
- Effective of smoke detection system
- Effective of smoke protection system, such as smoke barrier or smoke curtain
- All other studies that mainly concerned on smoke in large space

The methods on proper determination of fuel area and volume with given heat release rate were also investigated in this paper. It would help CFD engineers to build up a simple and accurate CFD model for further practical fire safety design.

ACKNOWLEDGEMENT

The study was funded by ARUP Design & Technical Fund (DTF) 77762/13 (Development of Fire Field Modelling). Special thanks to Dr. MingChun Luo for his help on experimental data in this paper.

REFERENCES

1. G. Cox, R. Chitty and S. Kumar, "Fire modeling and the King's Cross fire investigation", Fire Safety Journal, Vol. 15, pp. 103-106 (1989).
2. R. Yin and W.K. Chow, "Studies on thermal response of sprinkler heads in atrium buildings with fire field models", Fire and Materials, Vol. 25, pp. 13-19 (2001).
3. V. Cheng, R. Yau, S. Lee, M. Luo and L. Zhao, "Validation of CFD models for room fire and tunnel fires" Interflam 2001, 17-19 September, Vol. 1, pp. 807-810 (2001).
4. W.K. Chow and R. Yin, "Discussion on two plume formulae with Computational Fluid Dynamics", Journal of Fire Science, Vol. 20, pp. 179-201 (2002).
5. S. Nam and R. G. Bill Jr, "Numerical simulation of thermal plumes", Fire Safety Journal, Vol. 21, No. 3, pp. 231-256 (1993).
6. ASTM E1355-97, Standard guide for evaluating the predictive capability of deterministic fire models.
7. W.K. Chow, "Preliminary views on implementing engineering performance-based fire codes in Hong Kong: What should be done?" International Journal on Engineering Performance-Based Fire Codes, Vol. 4, No. 1, pp. 1-9 (2002).
8. S. Kumar and G. Cox, "Some guidance on 'correct' use of CFD models for fire applications with examples", Interflam 2001, 17-19 September 2001, Vol. 1, pp. 823-834 (2001).
9. W.K. Chow, "Simulation of fire environment for linear atria in Hong Kong" ASCE Journal of Architectural Engineering, Vol. 3, Part 2, pp. 80-88 (1997).
10. G. Cox, Combustion fundamentals of fire, Chapter 1 - Basic consideration, pp. 4-5 (1995).
11. D. Bradley, "How fast can we burn?", The 24th International Symposium on Combustion, The Combustion Institute, Pittsburg, Pennsylvania, pp.247-262 (1992).
12. X.C. Zhou and J.P. Gore, "A study of entrainment and flow patterns in pool fires using particle imaging velocimetry", NISTIR-GCR-97-706, March (1996).
13. B.J. McCaffrey, "Momentum implications for buoyant diffusion flames", Combustion and Flame, Vol. 52, pp.149-167 (1983).
14. STAR-CD manual, CD adapco Group (2001).
15. R. Yin and W.K. Chow, "Verification of plume expressions with fire field models: Part I", Journal of Applied Fire Science, Vol. 10, No. 1, pp. 45-58 (2000-2001).
16. M.C. Luo and V. Beck, "The fire environment in a multi-room building – comparison of predicted and experimental results", Fire Safety Journal, Vol. 23, pp. 413-438 (1994).
17. The SFPE Handbook of Fire Protection Engineering, 2nd edition, Society of Fire Protection Engineers and National Fire Protection Association, April (1995).
18. F.C. Lockwood and N.G. Shah, "A new radiation solution method for incorporation in general combustion procedures", Proceedings of 18th International Symposium on Combustion (1981).
19. 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).

20. K.T. Yang, "Numerical modeling of natural convection-radiation interactions in enclosures", Proceedings of 8th International Heat Transfer Conference, Vol. 1, pp.131-140 (1986).

21. J.E. Gott, D.L. Lowe, K. A. Notarianni and W. Davis, "Analysis of high bay hangar facilities for fire detector sensitivity and placement", NISTIR Technical note 1423 (1997).

22. K.A. Notarianni, W. Davis, D.L. Lowe, S. Larammee and J.E. Gott, Interflam'96, 7th International Fire Conference, Cambridge, UK 26-28 March, pp. 487-496 (1996).

23. W.D. Davis, G.P. Forney, and R.W. Bukowski, "Developing detector siting rules from computational experiments in spaces with complex geometries", Fire Safety Journal, Vol. 29, pp. 129-139 (1997).

24. B.F. Magnussen and B.W. Hjerager, "On the structure of turbulence and a generalised eddy dissipation concept for chemical reaction in turbulent flow", 19th AIAA Aerospace Meeting (1981).