International Journal on Engineering Performance-Based Fire Codes, Volume 6, Number 4, p.215-222, 2004

APPLICATION OF CFD AND NUMERICAL STUDIES ON TUNNEL FIRE SIMULATION

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

ABSTRACT

A better understanding of tunnel fire behaviour in a vehicular tunnel is proposed with the application of Computational Fluid Dynamics (CFD). Air flow induced by an accidental vehicular fire in a tunnel is simulated using CFD. Further, sensitivity analyses on the grid size, and number of iterations on the required computing time and accuracy of the results are carried out.

1. INTRODUCTION

Vehicular fires in tunnels can give disastrous consequences to life and property. The travel speed of smoke would decrease gradually due to cooling and loss of buoyancy. Smoke would be trapped when mixing with the counterflowing air. A better understanding of tunnel fire behaviour would help the tunnel authority to provide better fire safety management. Scale model experiments were commonly used in the past [e.g. 1], but the scaling parameters concerned should be preserved. Now, Computational Fluid Dynamics (CFD) [2] is a practical tool in fire engineering [e.g. 1]. The fire environment including air velocity, temperature and pressure in the tunnel can be predicted [e.g. 3].

2. NUMERICAL EXPERIEMENT OF VEHICLE FIRE IN A TUNNEL

This paper is a report on studying the fire environment in a tunnel [e.g. 4] using CFD. Air flow induced by a car fire in a section of a tunnel was simulated by PHOENICS v.3.2 [5]. A tunnel section of cross-sectional area similar to one of the oldest local tunnels [6] was considered. It was simplified as a rectangular tube. As the tunnel is considered to be very long, only a section of length 100 m as shown in Fig. 1a was considered. A private car located 50 m away from one end was assumed to be on fire. Typical dimensions of the car are taken to be 1.7 m in width, 4.6 m in length and 1.5 m in height as specified in the local transport design manual [7] on the maximum vehicle lengths permitted under the Road Traffic Regulations.

A Cartesian co-ordinate grid system was assigned along the x, y (vertical) and z directions as shown in Figs. 1b and c. The tunnel was divided into 24, 16 and 333 cells along the three directions, with a total of 127,872 cells.

The ceiling jet induced by burning a car located at one of the lanes in the centre of the carriageway as shown in Fig. 1 is studied. The fire was taken as a heat source of length 4.6 m, width 1.7 m and height 1.5 m, having a thermal power of 0.5 MW.

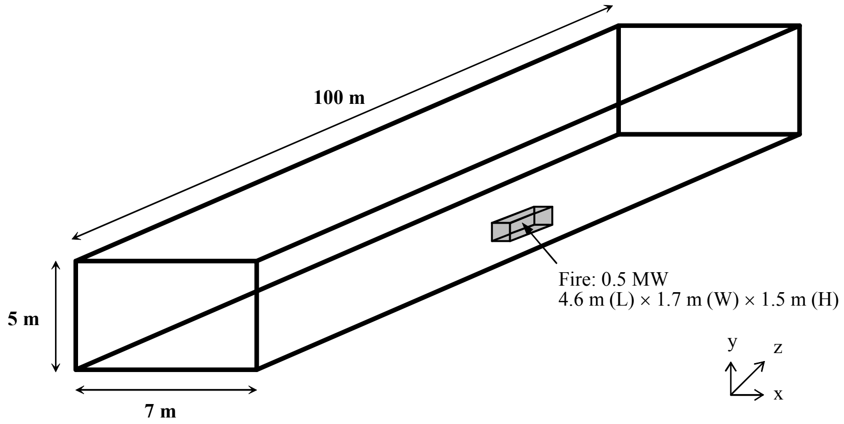
The transient CFD simulations were taken up to 120 s with time steps of 1 s, which was assumed to cover the critical situation at the early stage of fire development and smoke spread. The smoke extraction outlets and fresh air make-up inlets were set to be activated at 30 s. Tunnel ventilation fans shall achieve full speed operation from standstill in no more than 60 s as stated in the NFPA standard [8]. This criterion is satisfied for the simulations.

Simulations were in processed in 50 iterations per time step. The simulation started at time 0 s when the car started to burn. Steady burning with constant heat release rate of 0.5 MW was assumed. Typical temperature contours in the cross-sectional plane along the tunnel are shown in Figs. 2 and 3. The following are observed:

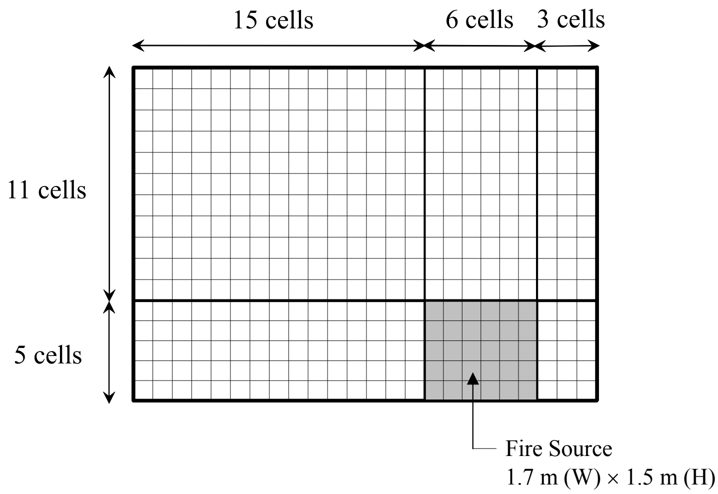
- The smoke temperature contours within the tunnel section in the first two minutes after the fire had started are shown in Fig. 2. Smoke was produced and propagated longitudinally along both directions of the tunnel. It can be seen from the velocity vectors in Fig. 3 that a strong buoyant plume is induced by the fire.

- Smoke rising in plume from the fire as in Fig. 3 would turn radially outward upon reaching the ceiling. Motion was blocked by walls, changing from a radial flow to one-dimensional long channel flow as described [9]. In this case, critical gravity flow conditions were established [10]. The smoke layer temperatures in the tunnel were roughly constant at distances along the fire channel greater than half of the tunnel width. Since the smoke production rate is actually taken as the air entrainment and there was no air entrainment in the developed channel flow as indicated in Figs. 3b and c. The smoke layer in the channel flow was also roughly constant.

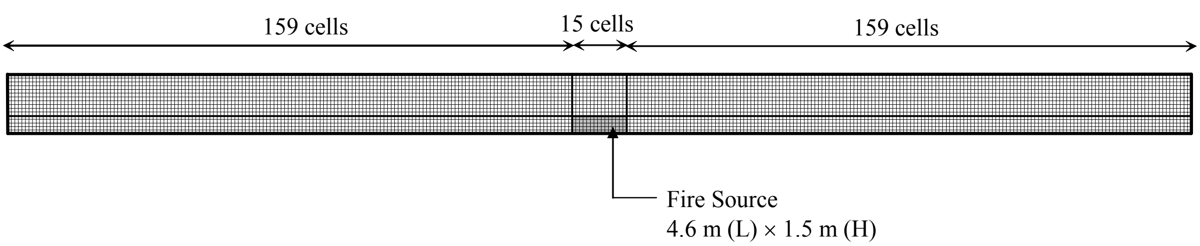
The results showed that smoke can be kept at a high elevation within the tunnel. High visibility can be obtained at heights below 3 m under low smoke concentration, which allows a high level of safety for passengers to escape.



(a) Geometry

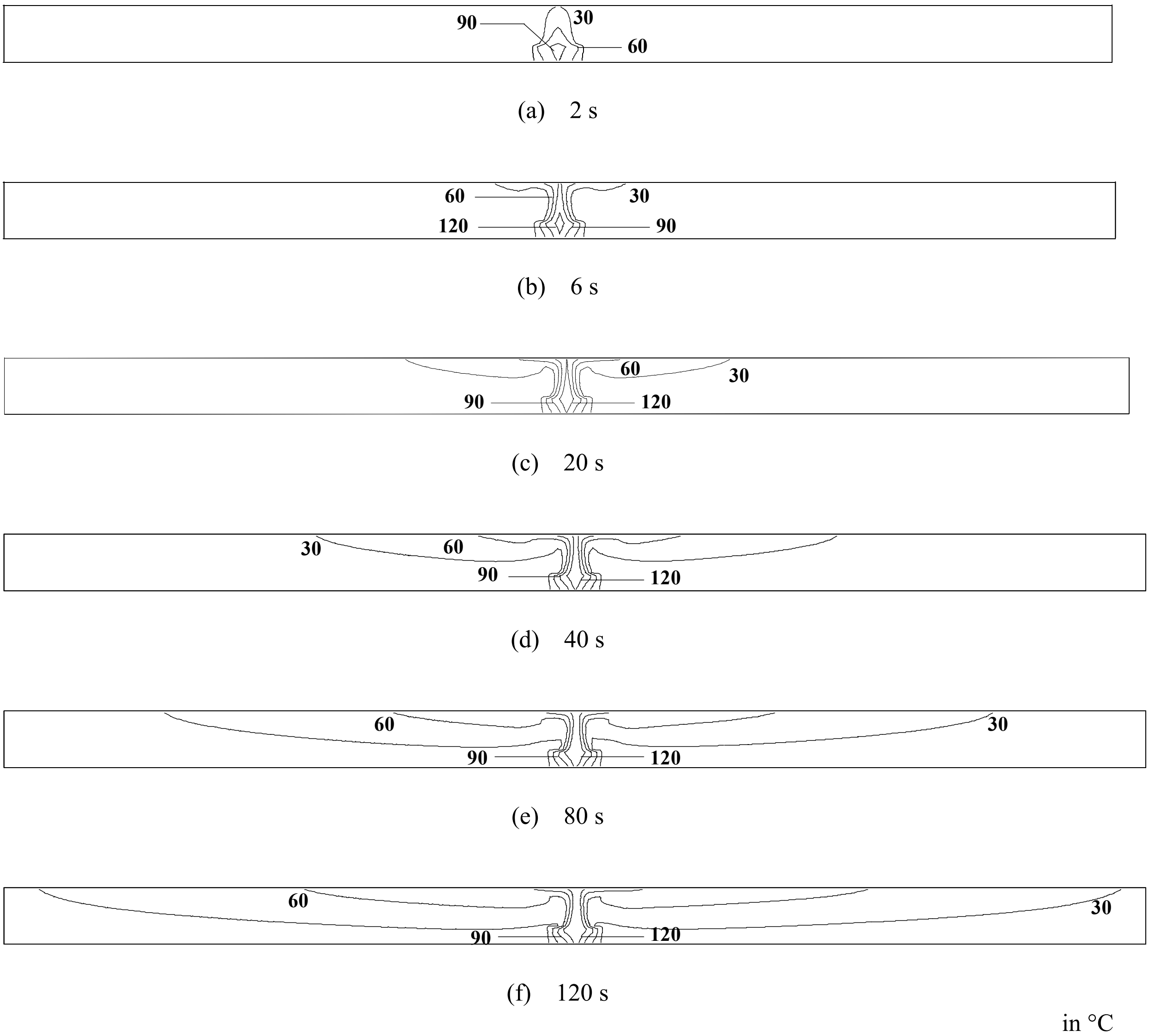


(b) Central x-y plane



(c) Central y-z plane

Fig. 1: Tunnel



(a) 2 s

(b) 6 s

(c) 20 s

(d) 40 s

(e) 80 s

(f) 120 s

Fig. 2: Temperature contours of the tunnel with a 0.5 MW fire

3. SENSITIVITY ANALYSIS

In applying CFD for fire engineering design in tunnels [11], very few examples have been reported on verifying the predicted results. This is also observed for building projects with fire safety designed though performance-based fire codes. It is difficult to judge whether the predicted results on smoke movement are good enough. Although CFD technique is generally accepted by the tunnel community as a tool to simulate fires in railway and subway tunnels and stations, engineers concerned must fully understand the physics behind so that intelligent judgement can be made on accuracy and validity of the results. In lieu of carrying out systematic experimental validation work, the following parameters are tested using steady-state conditions:

- Number of grids

In applying CFD, the domain of interest is required to be divided into computational grids, so that answers can be given by numerical solutions at discrete points. Variability in grid generation is very broad, for example, the spacing of the grid points in x, y and z directions are set to be uniform in this paper such that the programming of the solution can be greatly simplified, storage space can be saved, and usually more accurate results can be obtained [11]. On the basis of uniform grid spacing, simulations were performed with different grid finenesses. The tunnel air temperature simulated in steady-state using a different number of iterations and different grid size are compared in Figs. 4 and 5.

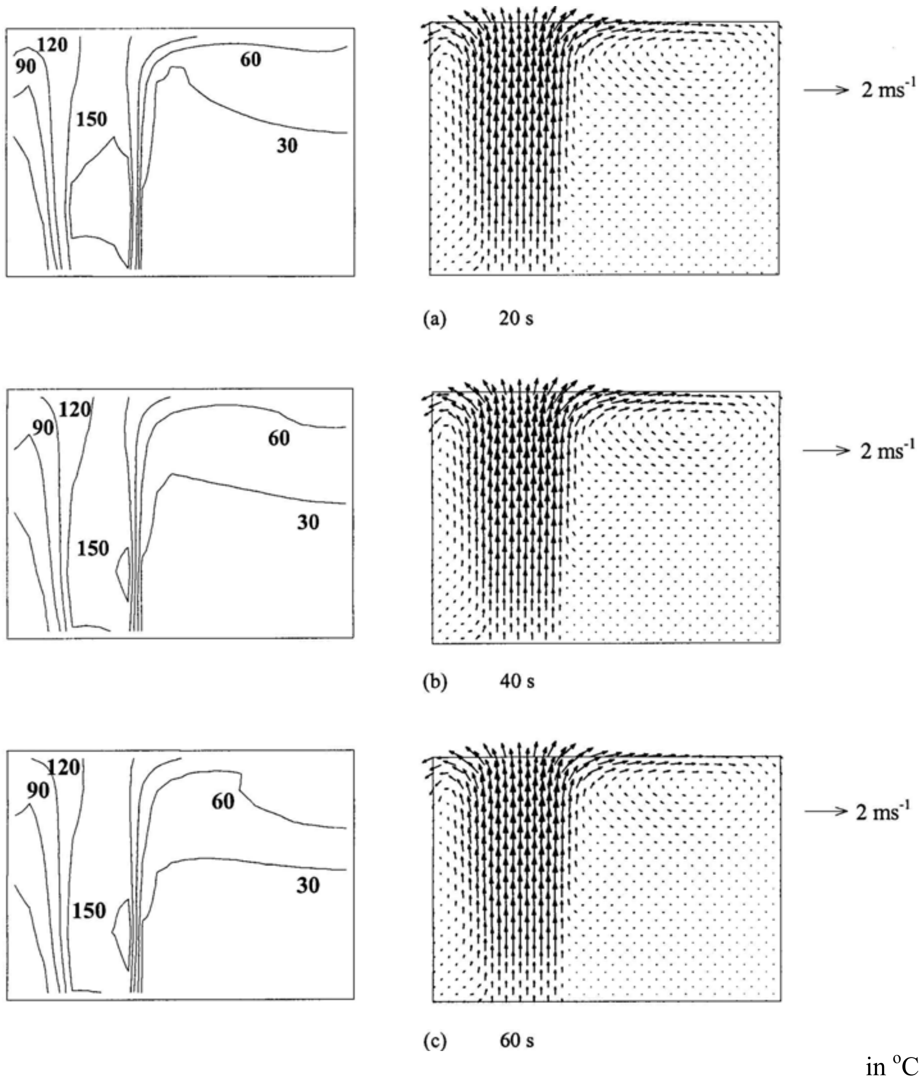


Fig. 3: Temperature and velocity profiles in sectional view

- Number of iterations

  By using one iteration a time step, five different number of iterations were 10, 25, 50, 75, 100. For simulations with coarse grids as shown in Figs. 4 and 5, the results from a small number of iterations differ markedly from the others. By refining the grids, the deviations between the results from different numbers of iterations were reduced as demonstrated in the figures. However, the results from the smallest number of iterations predicted by the finest grids, as shown in the figures, still deviated quite largely from the others. Although grid resolution is an important consideration with respect to capturing the simulation details, iteration numbers should be considered carefully.

- Computing time

  While fine grids and a large number of iterations can help to achieve high numerical accuracy, computing time is another issue that is important. A comparison of the computing time taken to run the simulations for the model at different numbers of iterations and control volumes is shown in Fig. 6. It is shown that about 17 hours of computing time is required to simulate up to 120 s. The computing time might be much longer for more complicated models. CFD results could not be claimed to be grid-independent everywhere at all time steps, however, the results presented indicate that the effects of grid size become small at 10 iterations and that there would not be a significant improvement by refining the grid further. Therefore, the definition of the engineering problem to be simulated and the skill and knowledge of the CFD user can play a vital part in obtaining accurate results.

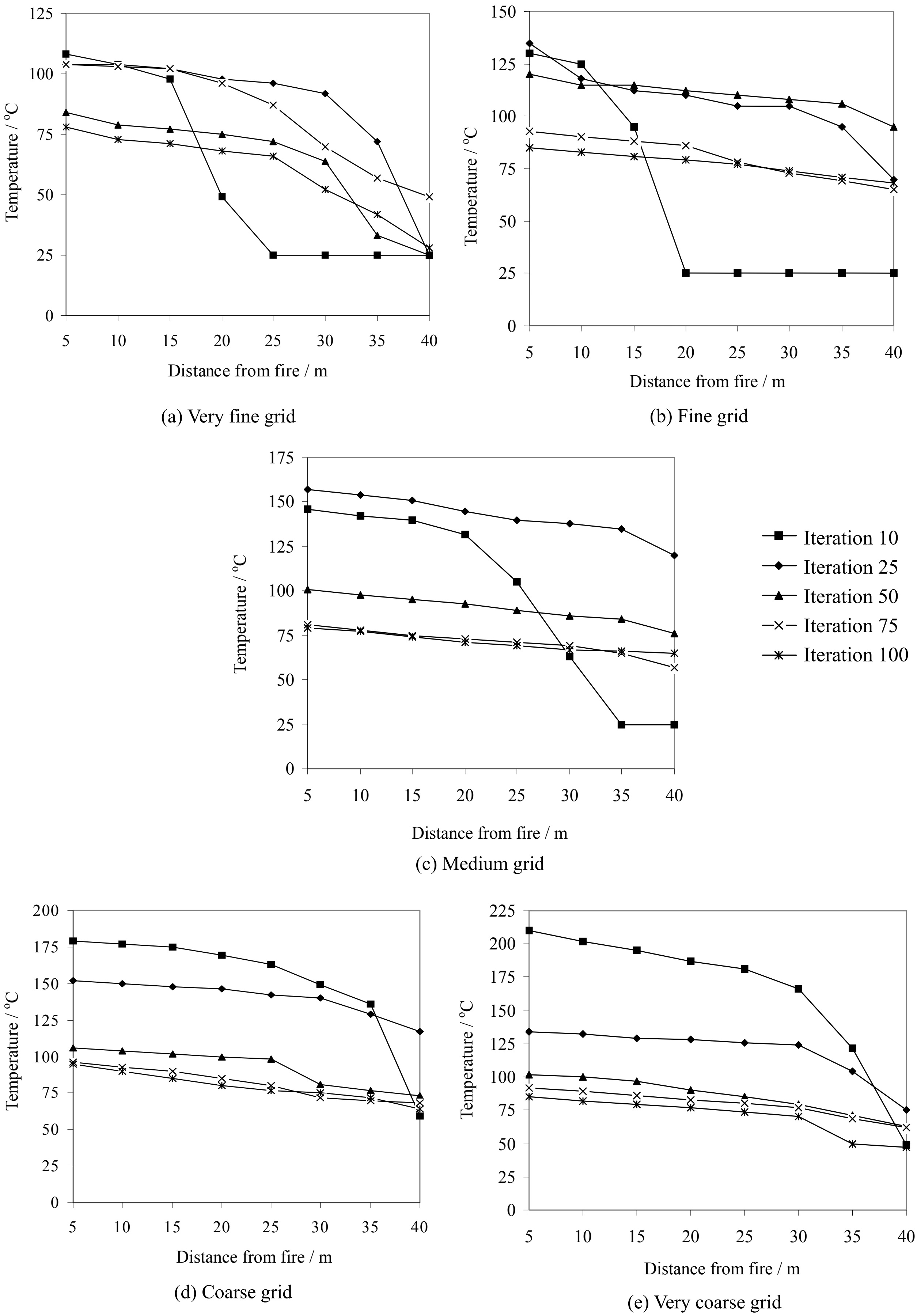
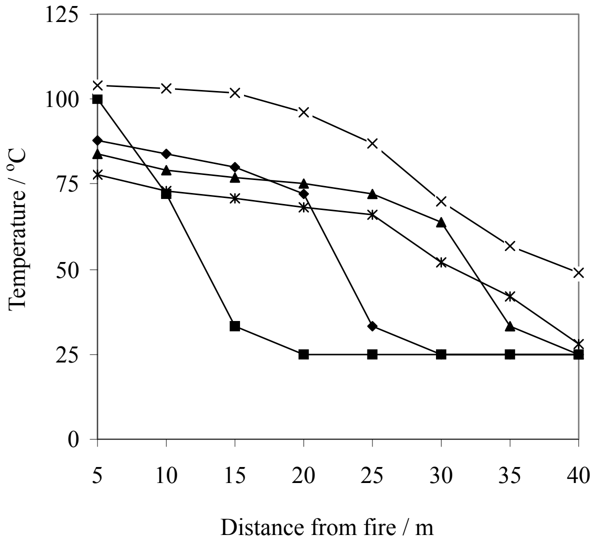
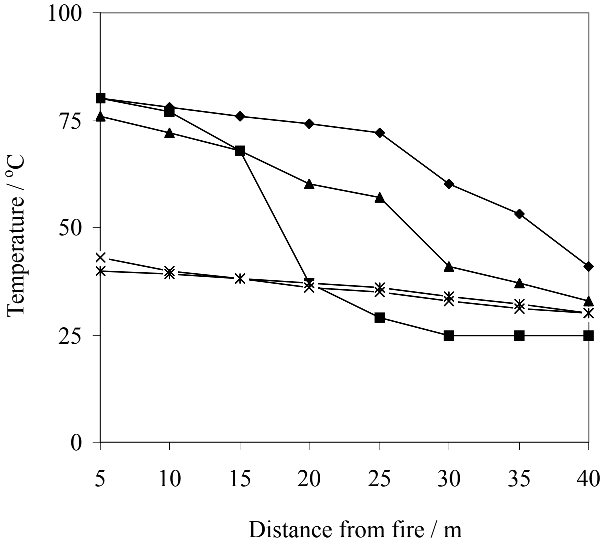


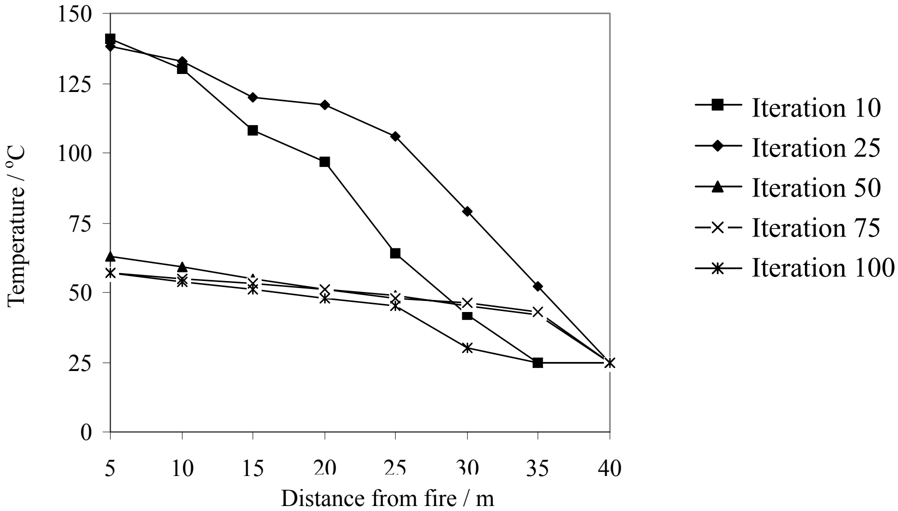
Fig. 4: Grid variation on air temperature at the ceiling



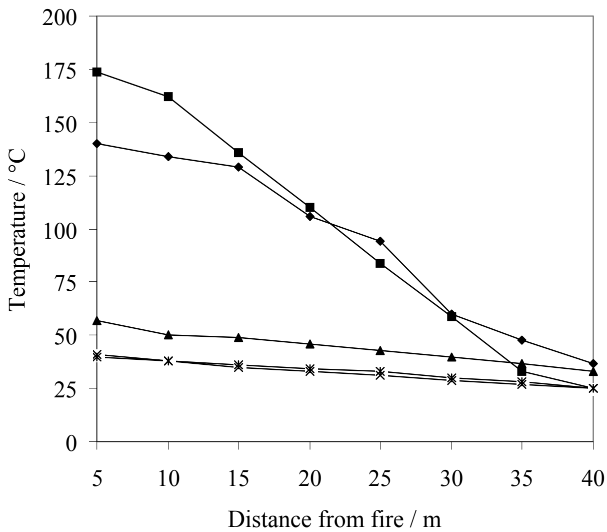
(a) Very fine grid



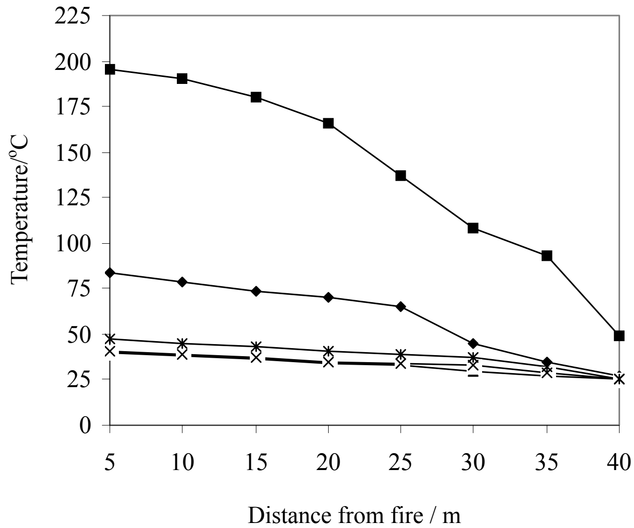
(b) Fine grid



(c) Medium grid



(d) Coarse grid



(e) Very coarse grid

Fig. 5: Grid variation on air temperature in the middle part of the tunnel

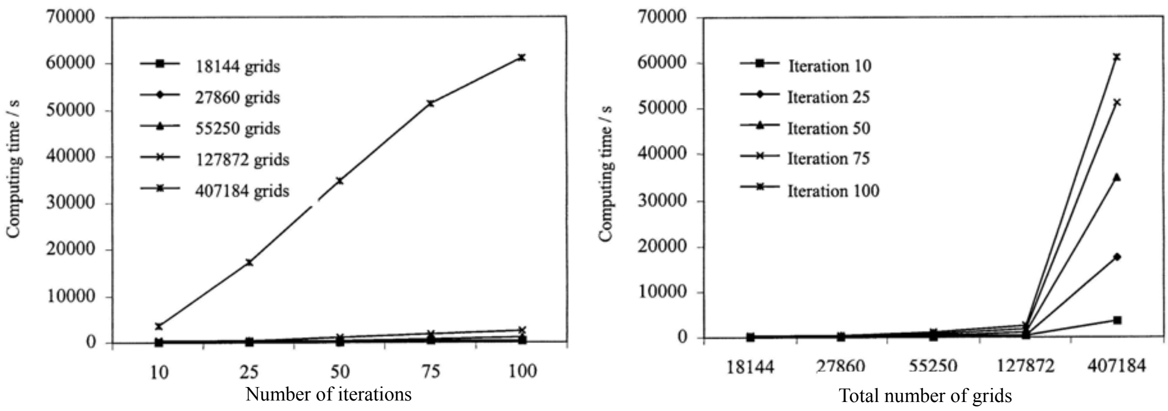


Fig. 6: Computing time of CFD in relation to the number of iterations and grids

4. CONCLUSION

Tunnel ventilation system design can be developed in depth from the predictions of various parameters, such as vehicle emission dispersion, visibility, air velocity, etc. Different tunnel configurations can be analyzed by CFD without the need to perform the actual tests. Operation parameters of the safety system can be changed regularly to cope with the demand. Grid resolution and iteration number are two of the important considerations with respect to capturing the details of the smoke and heat distribution in the tunnel. The computing time required for CFD simulations is still a concern for the engineering professionals. Further, most likely, a CFD expert has to be appointed for carrying out the works. More education and training on this subject area should be provided by higher education institutions.

ACKNOWLEDGEMENT

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

REFERENCES

1. O. Megret, O. Vauquelin, P. Chasse and E. Casale, "A reduced scale model for the study of fire-induced smoke control", Proceedings of Safety in Road and Rail Tunnels, pp. 713-723 (1998).
2. G. Cox, Combustion fundamentals of fires, Academic Press, UK (1995).
3. J.S.M. Li and W.K. Chow, "Vehicular tunnel ventilation design and application of CFD", Air Distribution in Rooms, Ventilation for Health and Sustainable Environment, ROOMVENT, Vol. 2, pp. 1171-1176 (2000).
4. W.K. Chow and J.S.M. Li, "Case study: Vehicle fire in a Cross Harbour Tunnel in Hong Kong", Tunnelling and Underground Space Technology, Vol. 16, No. 1, pp. 23-30 (2001).
5. PHOENICS Version 3.2, Concentration, heat and mass, CHAM Co., London, UK (1999).
6. South China Morning Post, 21 October (1972).
7. Transport planning and design manual, v.2, Highway design characteristics, Transport Department, Hong Kong (1984).
8. NFPA 502 Standard for road tunnels, bridges, and other limited access highways, National Fire Protection Association (1998).
9. SFPE Handbook of Fire Protection Engineering, 2nd ed., Section 2, Chapter 4, "Ceiling jet flows", Society of Fire Protection Engineers and National Fire Protection Association (1995).
10. M.A. Delichatsios, "The flow of fire gases under a beamed ceiling", Combustion and Flame, Vol. 43, pp. 1-10 (1981).
11. J.D. Anderson, Computational fluid dynamics: The basics with applications, McGraw-Hill (1995).

Q & A

Q1: With reference to some papers on CFD simulations, people assign more grids near the fire source. Why you used uniform grid spacing in your simulation?

Li: I used uniform grid spacing to save storage space of the computer and the computational time. Different grid systems will give different results but it might not be satisfactory in my simulation. So, I only used uniform grid system first. Variation of grid system is very board and complex. We can try different grid size assignment. The next stage will be comparison of computational and experimental results for validation and to find out different characteristics of different grid systems.

Q2: As indicated in your paper, temperature is dependent on the iteration number. What is the reason?

Li: The ceiling temperature or the temperature at the middle part of the tunnel varies with the number of iterations. According to my results, lower number of iterations may give results greatly different from those using other iteration numbers. I have not carried out comparison with experimental data yet and it is hard to draw a conclusion on what is correct at the moment.