

APPLYING LARGE EDDY SIMULATION IN THE STUDY OF FIRE AND SMOKE SPREAD AT UNDERGROUND CAR PARK

T.T. Chow, Zhang Lin and C.F. Tsang

Building Energy & Environmental Technology Research Unit
Division of Building Science & Technology, City University of Hong Kong
Tat Chee Avenue, Hong Kong, China

(Received 30 March 2006; Accepted 27 December 2006)

ABSTRACT

In this study, the large eddy simulation technique has been applied to predict the spread of smoke and fire in a public underground car park. The simulation platform has been the fire dynamics simulator, which was developed by the US National Institute of Standards and Technology (NIST). The results show that in such a confined space the smoke from a severe fire can spread quickly and dangerously. With the use of the conventional balanced ventilation system as the means of smoke extraction, a higher ventilation rate tends to accelerate the spread of fire, smoke, carbon dioxide and carbon monoxide. At the same time, this leads to a higher temperature environment which can endanger the human life. The probability of flashover was also found to increase with higher ventilation rates. The findings suggest that the current fire safety design recommendations are insufficient. Several suggestions on improving the public car park safety are raised, such as to avoid the use of excessive ventilation rate, to increase the number of emergency exits, and to shorten the vehicle exit routes.

1. INTRODUCTION

The underground car park has been the subject of study in recent years due to the increasing prevalence of this type of structure in modern cities. The nature of the car park as a relatively confined space and the high risk of vehicle fire make them potentially dangerous. In these days, the fire risk potential can be assessed through fire modeling. Fire modeling is generally divided into three categories. The first is the empirical method, which involves costly experimentation and also physically an element of risk. The second method is the zonal method which assumes certain characteristics of the fire, such as its stratification. The final method is through the use of computational fluid dynamics (CFD) analysis, which can be based on finite difference, finite element or finite volume methods.

The poor performance of conventional ventilation systems in clearing out smoke was demonstrated through a series of fire tests by Colt International [1]. A new method involving impulse and induction was outlined. Safety systems in relation to underground car park were investigated by Chow [2]. The fire environment was studied by a fire zone model (CFAST), and based on this the recommendations for future car park design were given. The activation time of the sprinkler was found critical and suggested to be longer than the escape time of occupants. According to the fire regulation of Hong Kong, automatic sprinkler, fire hydrant and hose reel systems are to be provided at underground car parks. In addition, smoke

extraction systems are to be provided for those greater than 7,000 m³ in volume. The mechanical means of smoke extraction gives a better guarantee of the flow rate since it affects little by the stack and wind interaction. It is also effective to prevent smoke from spreading out into the escape routes and to other functional areas of the building. A simple design is to make direct use of the ventilation system already installed for public health. As a general design practice, the ventilation rate is usually 6 to 8 air changes per hour (ACH).

2. LITERATURE REVIEW

Numerous types of CFD software are available in the market and able to simulate combustion and fire effects. The accuracy of various turbulence models such as k-epsilon, renormalized k-epsilon and other combinations was assessed by Chow [3]. It was also found that similar results could be obtained from these various models. Kandola and Morris analyzed the results of simulation studies that were carried out using AEA CFD-FLOW3D [4]. The hazards of smoke were highlighted, since the smoke from distant fires could affect people and fire safety equipment. The attempts of AEA Technology in validating CFD codes were discussed by Sinai et al. [5]. Practical examples were also presented, such as the case of an explosion on an oil rig platform. The PHOENICS CFD software was used by Mawhinney et al. to simulate domestic fires [6]. It was found that the simulated results agreed well with observed trends.

A review of CFD fire modeling and the important aspects of CFD applications such as mesh size was given by Wang et al [7]. Steckler et al. used experimental fire data to validate several CFD models [8]. Novozhilov reviewed the current trends in compartment fire modeling [9]. His paper discussed the importance of validation and experimental studies. Large eddy simulations were discussed in detail as well. Also given was a comparison between zone and field models, and the continuing trend towards field models.

Fires in tunnels where vehicle explosions are frequent have been studied extensively through both CFD and full scale tests. The vehicle tunnel is similar to a basement car park in that both are confined structure where smoke and fire can trap people. den Boer et al. carried out full scale fire tests on a new tunnel built in the Netherlands [10]. The CFD PHOENICS software was used to simulate several of the tests. It was found that the accuracy of the software was reasonable, considered that onsite conditions could not be simulated completely and accurately. The smoke was found over predicted in some cases. CFD fire simulation is economical in use and inherently safer than full scale fire tests. Tabarra et al. used scale model tests to validate CFD models applying to a tunnel. The aerodynamic interaction of supply vents and fire was studied, for the purpose of generating technical data for ventilation system design [11]. The use of CFD modeling of train fires in underground stations was studied by Deng et al. [12]. The paper discussed grid selection, physical modeling, transient versus steady state flows. The problems associated with fire modeling such as setting up of appropriate boundary conditions were also addressed in detail.

In the current study, the large eddy simulation method was applied to study smoke and flame propagation at the type of public underground car park commonly found in Hong Kong. This was performed through the use of a public-domain software – the Fire Dynamics Simulator.

3. LARGE EDDY SIMULATION

The Fire Dynamics Simulator (FDS) software was developed based on the solution of the Navier Stokes equations for momentum and energy conservation in fluid flow [13]. The incompressible Navier Stokes equations, written in tensorial notations, are:

$$\frac{\partial u_i}{\partial x_i} = 0 \quad (1)$$

and

$$\frac{\partial u_i}{\partial t} + \frac{\partial u_i u_j}{\partial x_j} = \frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j \partial x_j} \quad (2)$$

where i and j stand for 1, 2 or 3 and correspond to x , y and z co-ordinates. p is the static pressure, ρ is the density and ν is the kinematic viscosity of the fluid. The problem with turbulence modeling is largely due to the wide variation of length scales. In order to describe the physical processes adequately a high grid resolution is needed. This requires large amounts of computing power and memory. In addition, the large range of time scales further complicates the problem. To overcome these problems, various solutions have been proposed including the RANS (Reynolds Averaged Navier Stokes) solution which gives the time averaged values. These solutions are based on the well known eddy viscosity models. Large eddy simulation is another type of approximation where the large scales are simulated directly, and the smaller scales are computed based on simple turbulence model.

The FDS has two options: large eddy simulation (LES) and direct numerical simulation (DNS). The solution for the small grade scales are based on the Smagorinsky model [14]. Further, there are two types of combustion models, of which the choice depends on whether the LES or DNS simulation has been chosen. In LES simulation, a mixture fraction combustion is utilized. In this case the large scale convective and radiative heat transfer processes are calculated directly while the small scales are approximated. The soot yields are taken from the FDS database which came from the SFPE Handbook of Fire Protection Engineering [15]. Numerous texts give illustrations and explanations about the differences when comparing LES, DNS, and typical eddy viscosity models such as k -epsilon. The FDS program uses finite differences for solution of the above equations (1) and (2) while finite volume is used to solve the thermal radiation transport equation. Lagrangian particles simulate the smoke and air movement. The use of the finite difference approach obviously limits the program to rectilinear meshes and basic geometries, but increases the computation speed.

4. VALIDATION EXAMPLES USING FDS

Hadjisophocleous and McCartney presented the guidelines on the use of CFD to model fire [16]. In particular the NIST fire dynamics simulator FDS was covered in detail. Several issues were examined, including grid resolution, combustion modeling, smoke modeling and radiative fraction. Ryder et al. carried out a validation test of the FDS simulator. Four case examples were used and

compared with experimental data; good agreement was found [17]. Devaud and Weckman tested the ability of the FDS simulator in modeling combustion, fire and soot, in the case of a fuel spill near an aircraft fuselage [18]. Reasonable agreement between experiments and computations was found. The NIST FDS was used by Lautenberger et al. to predict soot formation for hydrocarbon flames [19]. A large outdoor fire was simulated by Vidmar and Petelin [20] using FDS. The results were compared with fire safety analysis reports. More discussions and examples of the validation efforts can be found in the NIST FDS manual [21].

5. PHYSICAL MODEL

The physical model of a basement car park, including the ventilation supply inlet and exhaust outlet positions, and the mesh grid, is outlined in Figs. 1 to 3. This $60\text{ m} \times 30\text{ m} \times 5\text{ m}$ underground car park is of size commonly found in basements of commercial shopping malls in Hong Kong. Two vehicle exits are at the west side and the east side

of the enclosing walls; both are 10 m wide. A number of emergency exits are also at the boundary walls. Exhaust outlets and supply inlets are uniformly distributed at the ceiling, typically for mixing ventilation systems. In many cases, the sources of ignition energy in vehicles are the same as those associated with structure fires. In our study, the modeled fire was started on top of a vehicle. The fire was increased linearly up to 5 MW from $t = 0$ to 100 s , as recommended by Morgan and Gardener [22]. Two case studies were performed using different ventilating rates of supply air at ambient temperature, i.e. 7 ACH for Case 1 and 14 ACH for Case 2. The supply velocities were respectively 3 ms^{-1} and 6 ms^{-1} .

The following three meshes were investigated to determine the most time effective solution: $60 \times 30 \times 10$, $100 \times 50 \times 20$ and $140 \times 80 \times 30$. The second mesh was finally chosen based on the satisfactory accuracy and speed of computation. The program files were then run on a Pentium 5 1.8 GHz PC for 12 hours. The program simulated three minutes real time, starting from the fire outbreak.

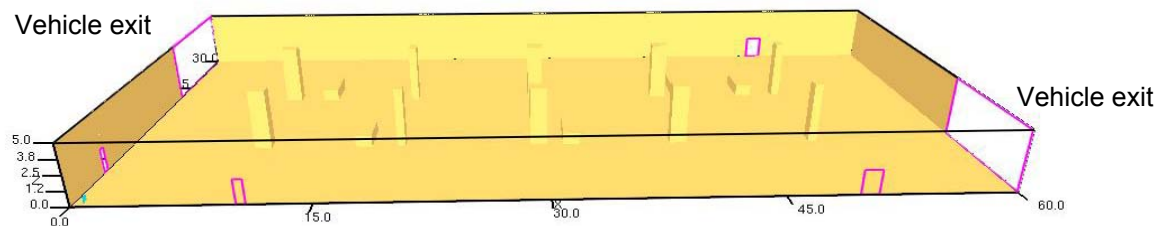


Fig. 1: Isometric view of the physical model of the underground car park

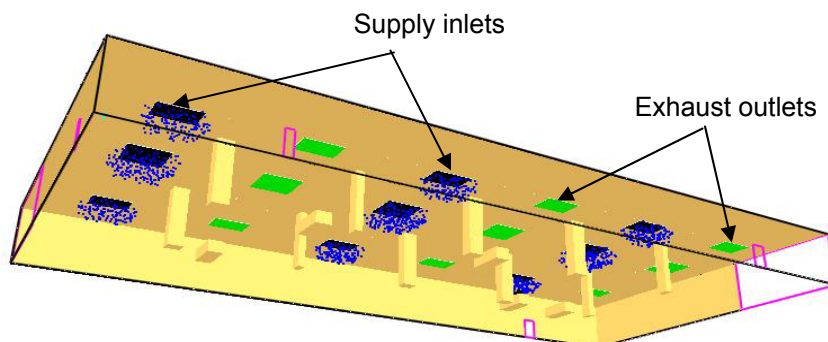


Fig. 2: Locations of supply inlets and exhaust outlets at ceiling

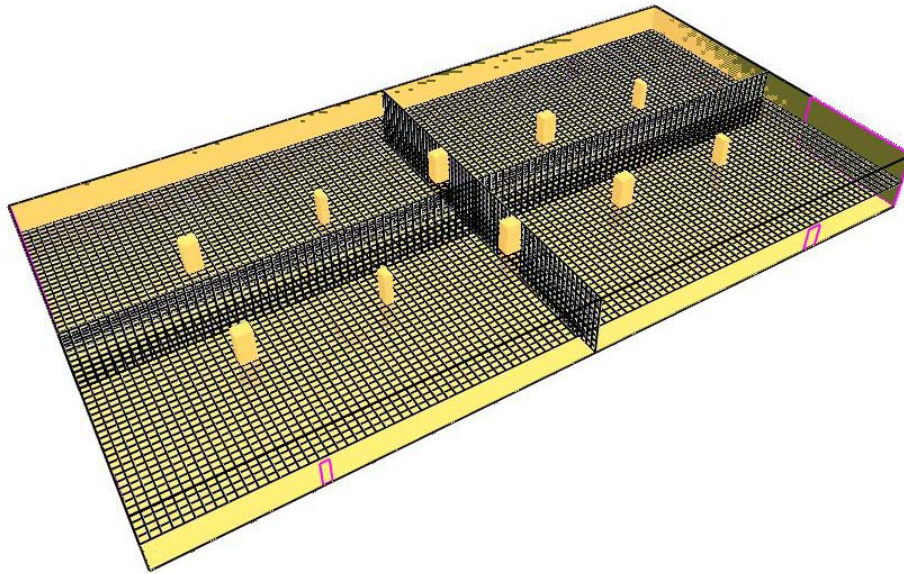


Fig. 3: Selected mesh grid of the underground car park model

6. RESULTS AND DISCUSSION

The case studies indicate the sensitivity of the fire and smoke spread to the ventilation system performance. Comparison between Cases 1 and 2, as in Figs. 4 and 5, shows that the smoke and fire is significantly increased by the change in ventilation rate. Initially during the first 20 s, as shown in Figs. 4 and 5, the fire (light color) and smoke (dark color) propagations can be seen very similar. As time progresses, the fire in Case 2 is more dispersed and intense than in Case 1. The smoke produced is also slightly less for Case 2, the one with more fire present, although this increases later on as the simulation progresses. Obviously the increased ventilation rate supplies more oxygen to the fire and accordingly, increased the intensity of the fire. The bulk of the fire in Case 2 also seems to drift towards the left hand side of the car park as evidenced when $t = 100$ s.

The profile of the smoke in Fig. 5 illustrates further the smoke spread pattern. In both cases a mushroom cloud rises up to the ceiling and then spreads to the rest of the car park. The time for the smoke to reach almost all sections of the underground car park is about 20 s. The speedy spread within this short time span gives alarm to the danger of these indoor spaces.

The temperature profiles, given in Fig. 6, show the instantaneous temperature levels at 1.5 m above the ground level. The average temperatures for Case 2 are clearly higher than Case 1 at all time instants. The maximum fire temperature occurs at around $t = 40$ s, reaching 965°C for Case 1, and 1000°C for

Case 2. The temperature then drops gradually due to the decrease in combustion efficiency, which results from the build up of smoke and consequential reduction in oxygen supply. Comparing the temperatures at $t = 40$ s and at $t = 100$ s, it is found that for both cases, the areas of high temperature become smaller as the time elapses. Generally, the temperature can become unbearable after 30s. The doubling of the ventilation rate increases the overall temperature in the occupied zone significantly.

Fig. 7 shows the levels of carbon dioxide at the 1.5 m level. The average level of carbon dioxide is slightly higher for Case 2 with the higher air change per hour. This is particularly evident in the first 20 s of the fire starting. At around $t = 60$ s and $t = 100$ s similar levels of CO₂ are found, though in Case 2 the levels are more evenly dispersed throughout the car park. In Case 1, a more polarized distribution with higher levels at one end of the car park is observed. The carbon monoxide, shown in Fig. 8, follows a similar pattern and distribution for both cases studied.

Fig. 9 shows the visibility in metres for the 1.5 m level of the car park. It can be seen clearly that around $t = 20$ s the visibility decreases significantly. The visibility reduces to less than 1 m afterwards. In Case 2 the visibility is worse than Case 1 for most parts of the underground car park especially within the first 20 s. The initial stage of the fire is particularly crucial for evacuation; it is when the occupants can assess the situation and give responses.

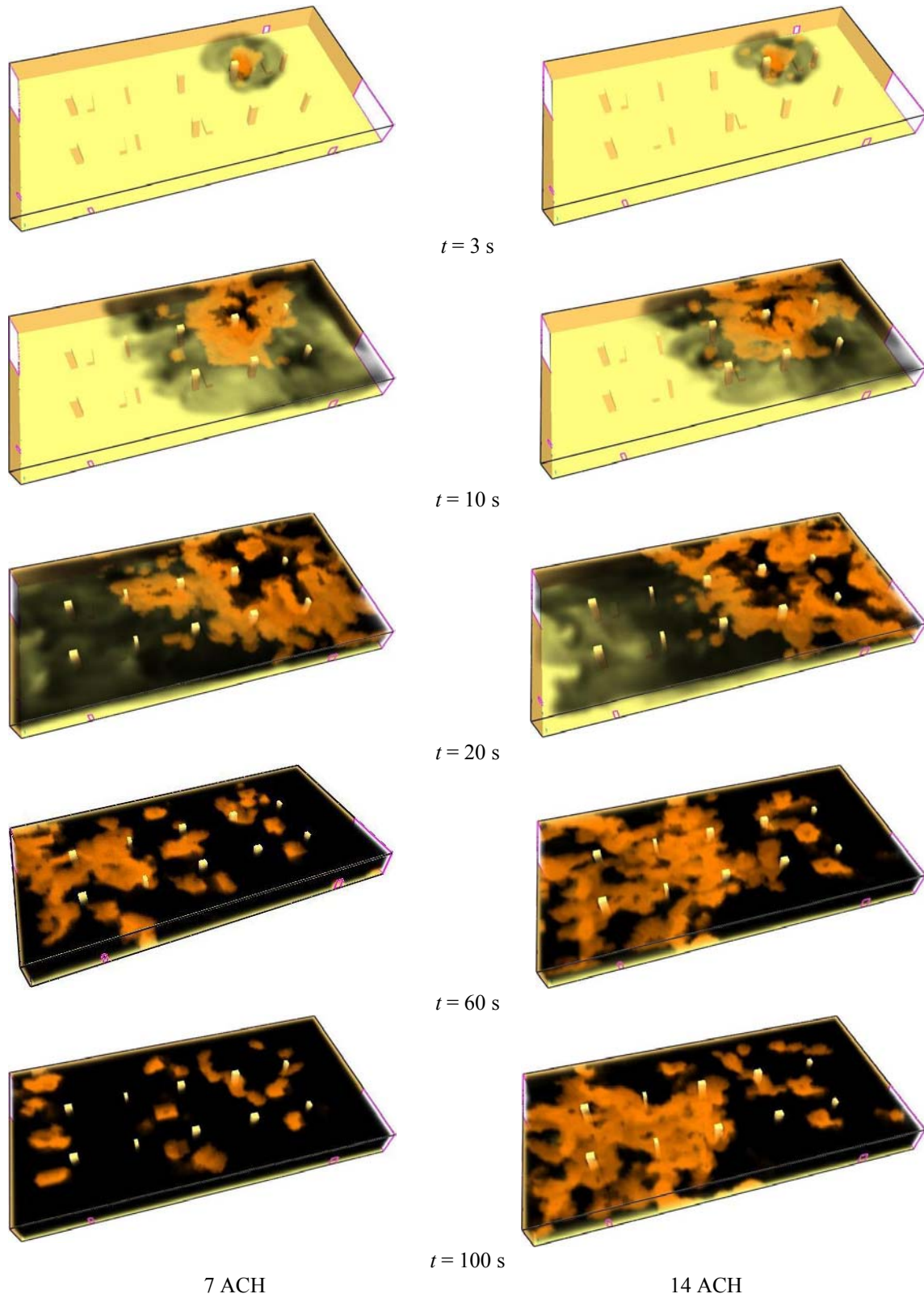


Fig. 4: Isometric view of smoke and flame propagation with time after the fire out break

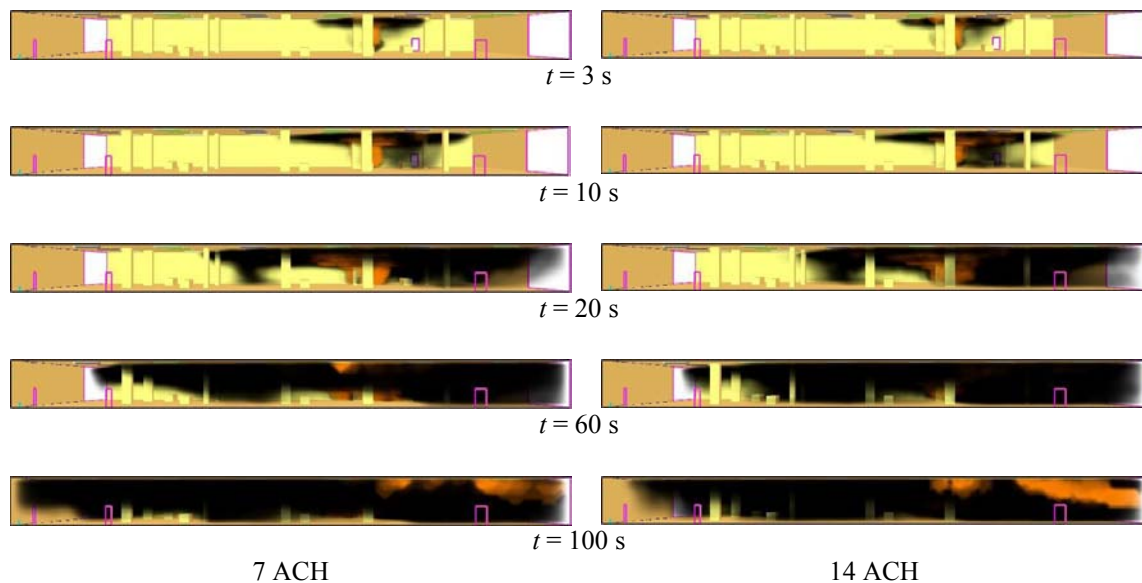


Fig. 5: Elevation view of smoke and flame propagation with time

Fig. 10 shows the heat released rate per unit volume (HRRPUV) from $t = 3$ to 100 s. The comparison between the two cases shows that the increased ventilation rate increases the combustibility of the surrounding air. The fire initially starts at the location of the vehicle. As the temperature increases, the surrounding gaseous mixture ignites as the fuel gasifying. The increased ventilation rate distributes the gasified fuel more effectively hence the higher level of heat released rate. The flashover point can be observed at around $t = 40$ s when the entire car park is undergoing complete combustion in Case 2, and the combustion in Case 1 is nearly (if not yet) complete. However, because of the lower ventilation rate in Case 1 the car park is never in full combustion, probably due to inadequate distribution of fuel vapor. Hence from the above observations, increasing ventilation rate increases the speed and likelihood of the fire reaching the flashover point. Also it can be seen that in the case of lower ventilation rate the fire subsides slightly after $t = 40$ s. This could be because of the increased proportion of smoke which reduces the combustibility of the air in the underground car park. And the lower ventilation rate results in more soot produced owing to inefficient combustion.

Taking the evacuation speed of the occupants (who are not familiar with the environment) as 0.8 ms^{-1} and the diagonal length (67 m) of the car park as the longest traveling distance, the maximum required escape time is therefore 84 s. For those occupants inside the vehicles, the escape time by means of the vehicles could be around 60 s, taking a vehicle traveling speed of 1 ms^{-1} . On the other hand, the sprinkler heads in a car park are commonly started to operate at 68°C . By cooling

the fire gases, and reducing the fire size (and hence the smoke), the sprinklers are viewed as an effective means of smoke control. For a fire size of 5 MW, its actuation time can be from 31 s to 66 s, depending on its response time index [2]. Since an early discharge of water may lead to a quick production of hot steam, which subsequently may hurt the occupants, a suitable response time of the sprinklers could be approaching 60 s. This is triple the 20 s for the smoke and fire to fill up the entire car park space. Another issue is that the smoke will further lower down the visibility and slow down the escape speed. For people who are not familiar with the building, a visibility of 15 m to 20 m is required for safe evacuation. If this is the case, the car park design should facilitate the occupants and the vehicles to evacuate within 20 s or so. Hence there are needs to increase the number of emergency exits for the occupants, and to shorten the vehicle exit routes.

7. CONCLUSIONS

The FDS simulator has been made use of to predict the dynamics of a reasonably large fire at 5 MW in an underground car park. The CFD technique allows an effective control of the boundary conditions and so an examination of the effect of various parameters. Our results show that in such a confined space the smoke and the flame can spread quickly and may fill up the entire volume space in less than one minute. This can be before the actuation of the automatic sprinkler protection system, and shorter than the evacuation time of the occupants. In this sense, the threat is high. For this type of car park, it will be desirable to increase the

number of exits, to shorten the vehicle exit routes, and to make the exit signs clear to the occupants.

A higher ventilation rate tends to accelerate the fire spread. This also leads to a higher temperature environment which can endanger the human life. In terms of safety it appears questionable whether the use of a higher ventilating rate (for the sake of indoor air quality) is a good design solution. In addition, increasing ventilation tends to disperse

the smoke more and to reduce visibility, especially during the initial phase of the fire outbreak. The possibility of flashover was also found higher for increased air ventilation rates. Our findings suggest that the current fire safety design recommendations can be insufficient to help the practicing engineers. More work to explore the potential hazards and the effective means of protecting human life should be proceeded in this area.

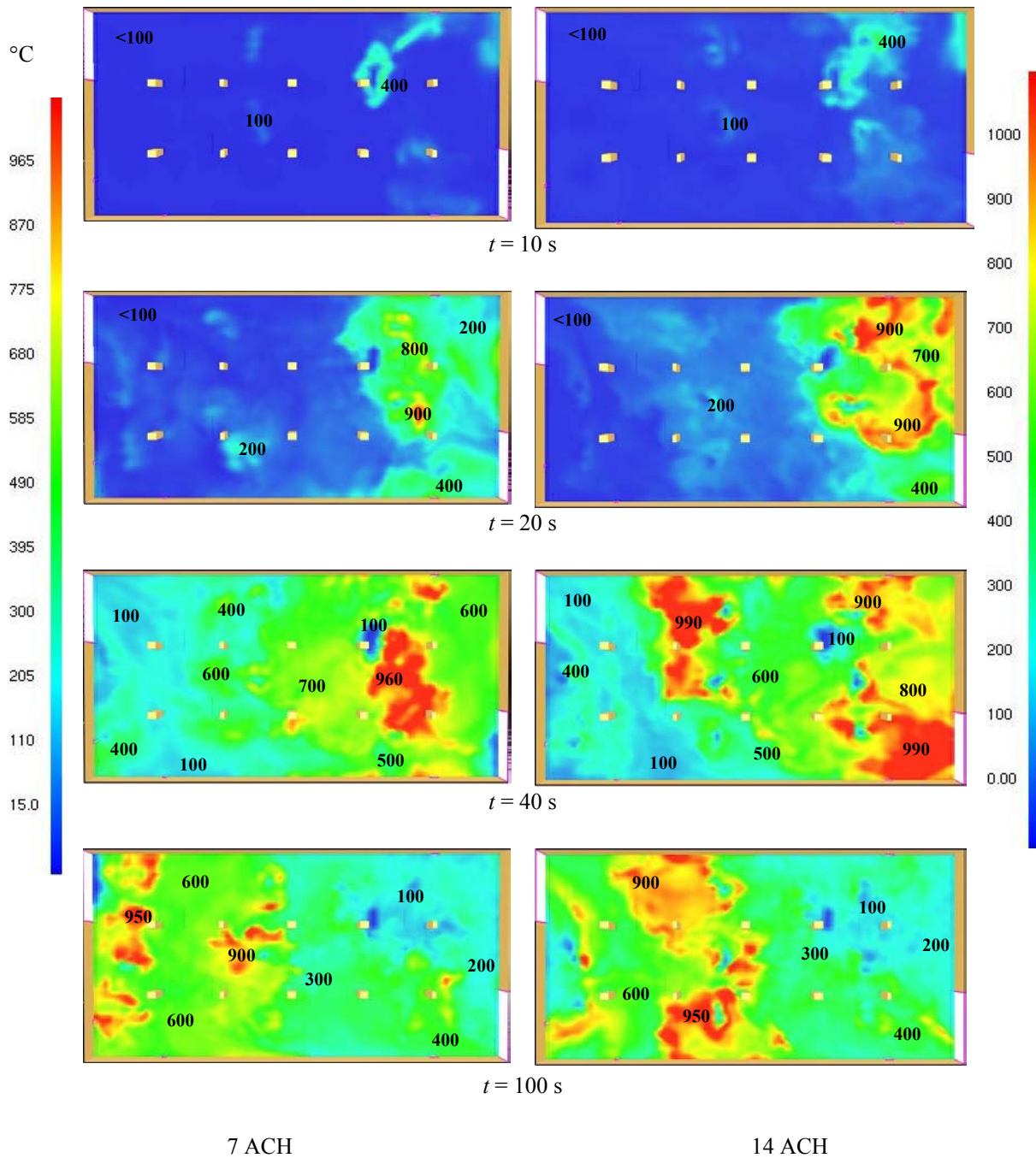


Fig. 6: Plan view of temperature distribution at 1.5 m level (°C)

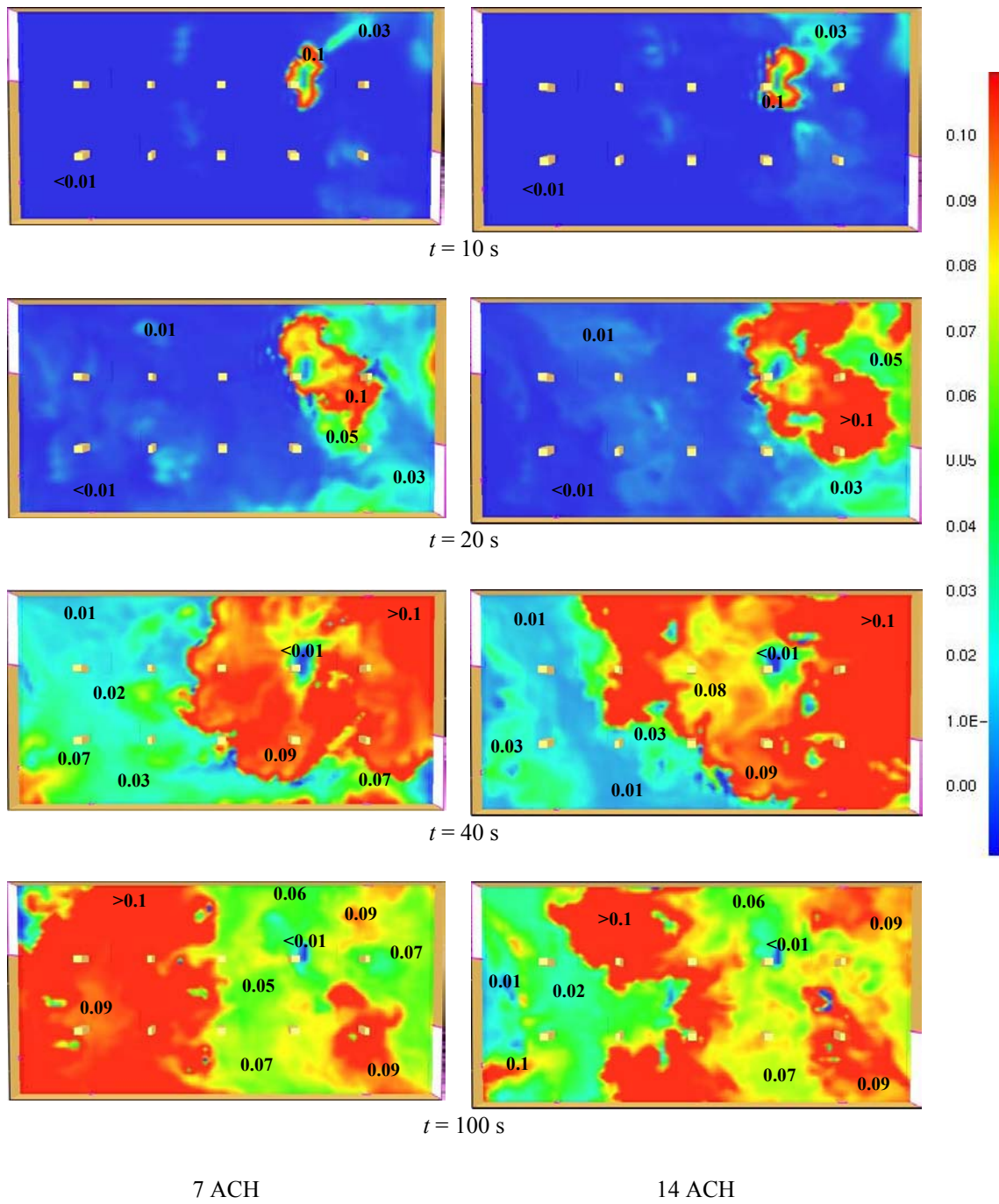


Fig. 7: Plan level of carbon dioxide at 1.5 m (mol/mol)

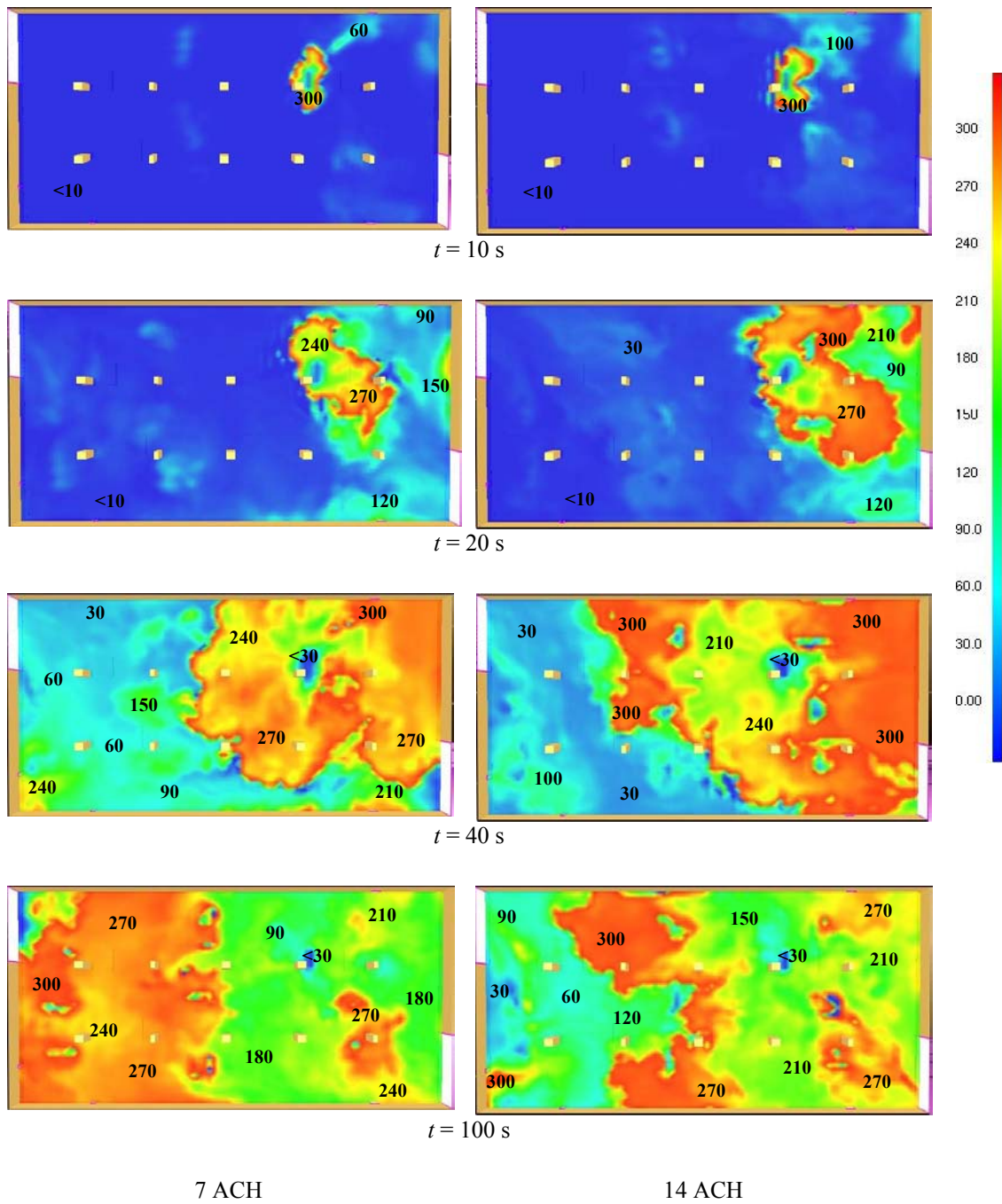
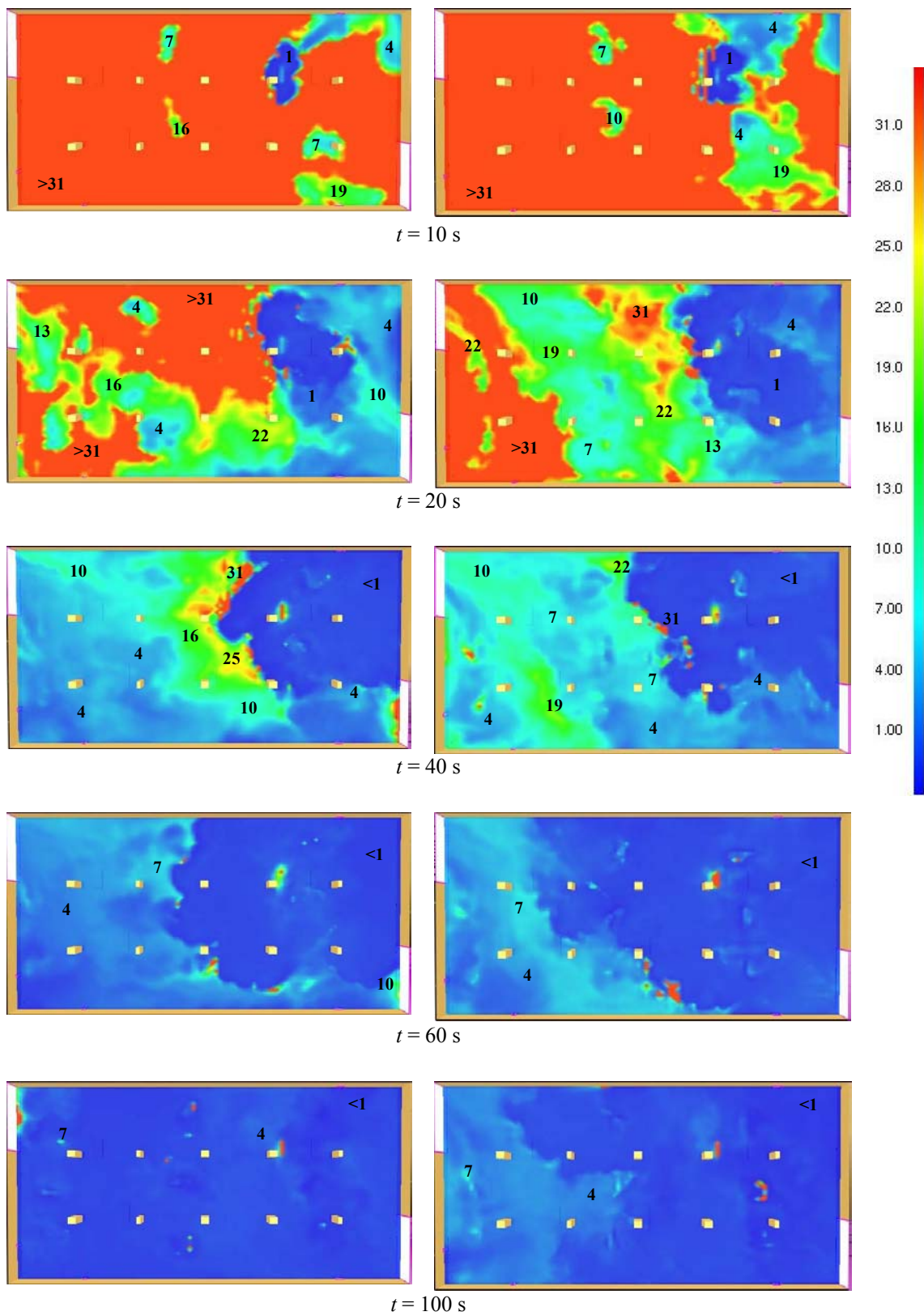


Fig. 8: Plan level of carbon monoxide at 1.5 m (ppm)



7 ACH

14 ACH

Fig. 9: Plan view of level of visibility at 1.5 m (m)

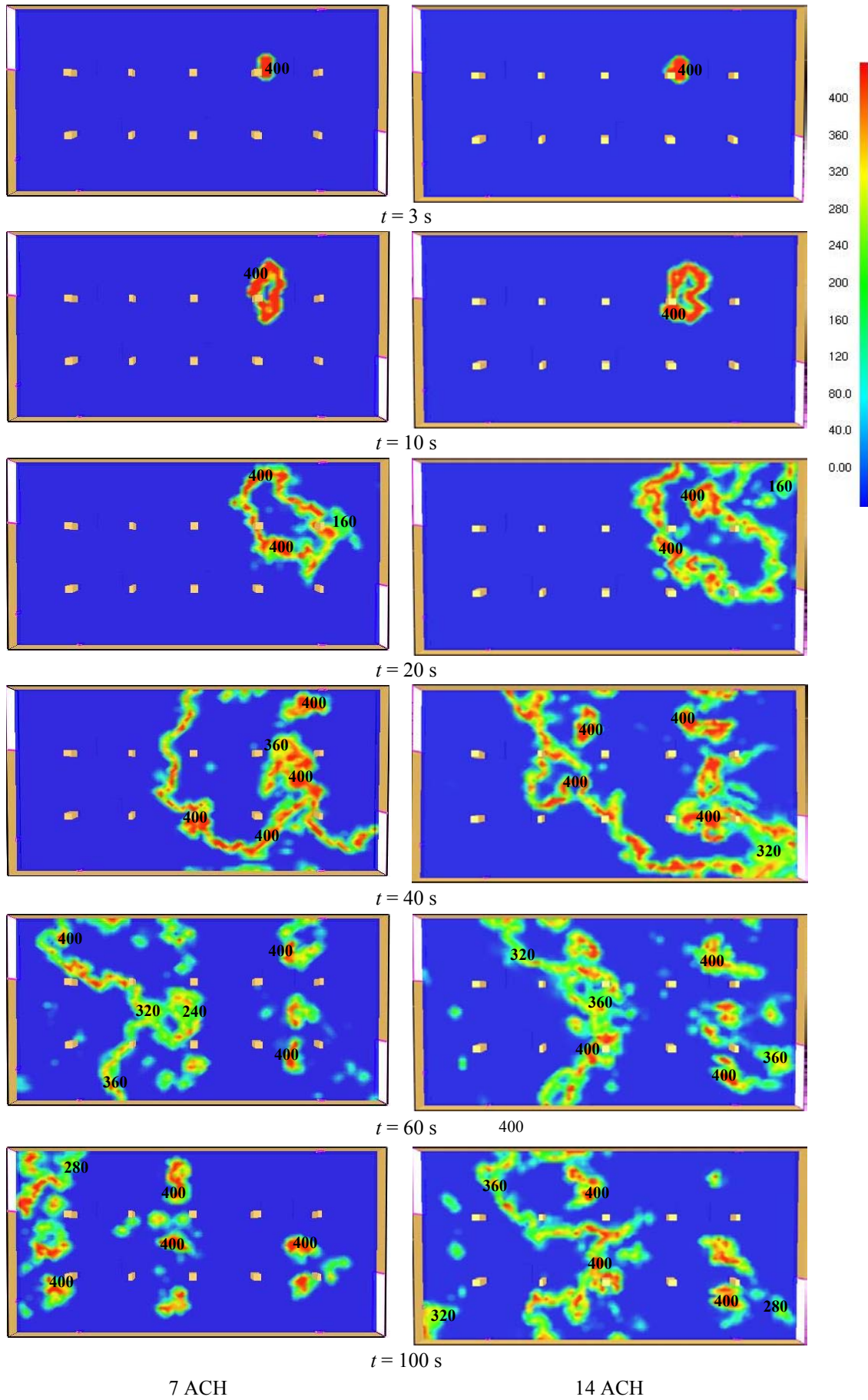


Fig. 10: HRRPUV (heat released rate per unit volume, kWm^{-3}) at 1.5 m level

REFERENCES

1. Colt International, "Smoke ventilation for underground car parks", *Building Engineer*, Vol. 80, No. 1, pp. 41-42 (2005).
2. W.K. Chow, "On safety systems for underground car parks", *Tunnelling and Underground Space Technology*, Vol. 13, No. 3, pp. 281-287 (1998).
3. W.K. Chow, "Numerical studies of airflows induced by mechanical ventilation and air-conditioning (MVAC) systems", *Applied Energy*, Vol. 68, No. 2, pp. 135-159 (2001).
4. B.S. Kandola and M. Morris, "Smoke hazard assessment using computational fluid dynamics (CFD) modeling", *Institution of Chemical Engineers Symposium Series*, No. 134, pp. 171-194 (1994).
5. Y.L. Sinai, M.P. Owens and P. Smith, "Advances in CFD assessment of fire and smoke movement", *Proceedings of the International Conference on Offshore Mechanics and Arctic Engineering - OMAE*, Vol. 2, pp. 389-395 (1995).
6. R.N. Mawhinney, E.R. Galea, N. Hoffmann and M.K. Patel, "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).
7. J. Wang, J.S. Hua, K. Kumar and S. Kurichi, "Modeling and validation of fire induced airflow in a room", *Proceedings of the 2004 International Symposium on Safety Science and Technology, PART A, Progress in Safety Science and Technology*, Vol. 4, pp. 1164-1170 (2004).
8. K.D. Steckler, J.G. Quintiere and W.J. Rinkinen, "Flow induced by fire in a compartment", *Symposium (International) on Combustion*, pp. 913-920 (1982).
9. V. Novozhilov, "Computational fluid dynamics modeling of compartment fires", *Progress in Energy and Combustion Science*, Vol. 27, No. 6, pp. 611-666 (2001).
10. D.J. den Boer, J.J. Mertens, N.J. van Oerle and J.F. Wijnia, "Validation of CFD calculations of full scale medium sized fires in a two lane road tunnel", *11th International Symposium on Aerodynamics and Ventilation of Vehicle Tunnels*, Vol. 2, pp. 657-667 (2003).
11. M. Tabarra, B. Kenrick and R.D. Matthews, "CFD validation of natural smoke movement in a model tunnel", *American Society of Mechanical Engineers, Fluids Engineering Division (Publication) FED*, Vol. 238, No. 3, pp. 543-546 (1996).
12. M.P. Deng, P.C. Miclea and D. McKinney, "CFD modeling considerations for train fires in underground subway stations", *American Society of Mechanical Engineers, Fluids Engineering Division (Publication) FED*, Vol. 238, No. 3, pp. 547-555 (1996).
13. K.B. McGrattan, H.R. Baum, R.G. Rehm, A. Hamins, G.P. Forney, J.E. Floyd and S. Hostikka, *Fire Dynamics Simulator - Technical Reference Guide*, NISTIR 1081, National Institute of Standards and Technology, Gaithersburg, MD, USA (2004).
14. J. Smagorinsky, "General circulation experiments with the primitive equations. Part I: The basic experiment", *Monthly Weather Review*, Vol. 91, pp. 99-152 (1963).
15. Society of Fire Protection Engineers, *SFPE Handbook of Fire Protection engineering*, National Fire Protection Association, Quincy, Boston, Massachusetts, USA (1988).
16. G.V. Hadjisophocleous and C.J. McCartney, "Guidelines for the use of CFD simulations for fire and smoke modeling", *ASHRAE Transactions*, Vol. 111, Part 2, pp. 583-594 (2005).
17. N.L. Ryder, J.A. Sutula, C.F. Schemel, A.J. Hamer and V. van Brunt, "Consequence modeling using the fire dynamics simulator", *Journal of Hazardous Materials*, Vol. 115, No. 1-3, pp. 149-154 (2004).
18. C.B. Devaud and E.J. Weckman, "Numerical simulation of a pool fire and large object in a cross-wind", *Proceedings of the ASME Heat Transfer/Fluids Engineering Summer Conference 2004*, Vol. 4, pp. 217-220 (2004).
19. C.W. Lautenberger, J.L. de Ris, N.A. Dembsey, J.R. Barnett and H.R. Baum, "A simplified model for soot formation and oxidation in CFD simulation of non-premixed hydrocarbon flames", *Fire Safety Journal*, Vol. 40, No. 2, pp. 141-176 (2005).
20. P. Vidmar and S. Petelin, "Large outdoor fire model analysis", *Proceedings of the 2003 International Joint Power Generation Conference*, pp. 817-822 (2003).
21. K.B. McGrattan, H.R. Baum, R.G. Rehm, A. Hamins and G.P. Forney, *Fire Dynamics Simulator - Technical Reference Manual*, NISTIR 6467, National Institute of Standards and Technology, Gaithersburg, MD, USA (2004).
22. H.P. Morgan and J.P. Gardner, "Design principles for smoke ventilation in enclosed shopping centres", *Building Research Establishment Report*, Fire Research Station, Borehamwood, UK (1990).