LARGE EDDY SIMULATION OF SMOKE MOVEMENT IN A SHAFT

Lianyu Cao and Yincheng Guo
Department of Engineering Mechanics, Tsinghua University, Beijing 100084, China

ABSTRACT

It is important to understand the behavior of smoke movement for designing effective fire control systems in buildings. On the basis of large eddy simulation, smoke movement in the case of a fire in a shaft is investigated here using Fire Dynamics Simulator (FDS) code. The predicted results show the behavior of smoke movement under different heat release rates and geometrical sizes of a shaft. Simulations show smoke vortex formation, evolution and abruption near the inlet wall of the shaft at the beginning of fire, and then the strong turbulent mixing process will occur in the shaft. It is found that heat release rate and geometrical size have remarkable effect on the formation of so called "stack effect".

Keywords: smoke movement, stack effect, large eddy simulation, shaft

1. INTRODUCTION

With the development of economy in recent years, large high buildings are constructed in many major cities. As there are many stairwells and elevator shafts in such high buildings, it will cause the so called "stack effect" when fire occurs. Flame and smoke will spread vertically rather than horizontally to those parts of the building, and then fire-generated smoke will overspread to building locations remote from the fire space, threatening life and damaging property. It is recognized that smoke is the major killer in fire situations. So, the investigation of smoke movement in buildings is of considerable practical importance for architects and engineers when they design the fire protection systems such as sprinklers and smoke control systems, etc.

Peppes et al. [1] had investigated both experimentally and numerically the buoyancy-driven air flow through a stairwell that connects two individual floors of a residential building. Their research focused on the mass and heat transfer between the two floors. The analysis of results provided the relations which can predict the mass and heat flow rate as a function of the inter-zone average temperature difference. Reynolds [2] and Zohrabian et al. [3] had conducted experiments in a reduced scale model of a typical stairwell. They investigated the air flow through an inclined channel connecting the two compartments of the model. The flow was driven by energy input from an electric panel heater located in the lower zone of the model. They found that there is a through-flow caused naturally by providing openings in both compartments in their experiments. Kazansky et al. [4] had analyzed chimney-enhanced natural convection from a vertical plate by means of experiments and numerical simulations. Their study deals with natural-convection heat transfer from a vertical electrically heated plate, which is symmetrically placed in a chimney of variable height. Experimental and numerical results indicated that the overall mass flow rate through the chimney increased with the height of the chimney. For the particular size and configuration of the system, an enhancement of the air flow rate up to 10 times was achieved in their experiments. Zukoski [5] had made a comprehensive review of smoke movement in shafts. It is found that two mechanisms are primarily responsible for vertical motion of buoyant smoke within a building. One mechanism is called as stack effect produced by pressure differences caused by density differences between the smoke inside the shaft and ambient atmosphere outside. Another mechanism is called as turbulent mixing process produced by instability between the upper layer cold gas and lower layer smoke.

The problems addressed in most of the above studies focus on the flow patterns in different stairwells and shaft models under certain heating conditions. Limited work has been performed on the flow in open vertical enclosures due to a fire. This study was undertaken to obtain detailed information on the flow of smoke in a shaft under different conditions with changing heat release rate of fire.

2. MATHEMATICAL MODEL

When investigating a fire process, it is difficult to obtain detailed information about the hydrodynamics of fire spread and smoke movement experimentally. Therefore, mathematical modeling and numerical simulation have the great advantage in evaluating and predicting the fire process. In numerical approach, mathematical models can be classified as three kinds of models [6] according to the level of describing information of a fire. They are field model, zone model and network model. Field model is based on computational fluid dynamics, computational heat transfer and combustion technologies, which can give much detailed information of the fire, and it is useful to recognize the characteristics of fire spread and smoke movement. On the basis of field model, large eddy simulation has been carried out in the present study to predict smoke movement in the case of a fire in a shaft. Hydrodynamics of smoke movement is investigated here using Fire Dynamics Simulator (FDS) code.

As fire usually happens in large regions and Reynolds number of smoke movement is also high and the flow is turbulent, it is necessary to consider the effect of turbulence for simulating fire-induced flows of smoke. For modeling turbulent flows, there are three kinds of micro-scale numerical simulation methods, named as direct numerical simulation (DNS), discrete vortex method and large eddy simulation (LES). DNS is a method in which all of the scales of motion of a turbulent flow are computed, DNS requires very fine grids in order to describe Kolmogorov micro-scale eddy and it is a valuable tool for investigating the physics of turbulence. For any realistic fire conditions, DNS requires a large number of grid points and is very costly. Discrete vortex method is mainly used in two-dimensional mixing layer flows or planar jet flows, and it is difficult to deal with complex geometrical boundaries in most fire conditions. The basic idea of LES is to simulate the larger scales motions of the turbulence while approximating the smaller ones, the larger eddies contain most of the energy and have evident effect on the mean flow, the smaller eddies are more universal and would be easier to model. Whereas, there is no evident limit between large scales and small eddies. If the computational grid size is small enough, the sub-grid scale model would be more universal. In recent years, the LES approach has come to be an effective method not only for studying turbulent flows but also for investigating fire and smoke movement in buildings [7].

In Cartesian coordinates, with the box-filter applied, the filtered governing equations of continuity, momentum, energy and mixture fraction have the following forms,

Continuity equation:

$$\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial \bar{\rho}\tilde{u}_j}{\partial x_j} = 0 \tag{1}$$

Momentum equation:

$$\frac{\partial \bar{\rho}\tilde{u}_i}{\partial t} + \frac{\partial \bar{\rho}\tilde{u}_j\tilde{u}_i}{\partial x_j}$$

$$= -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left\{\mu\left(\frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i} - \frac{2}{3}\frac{\partial \tilde{u}_k}{\partial x_k}\delta_{i,j}\right)\right\} + \bar{\rho}g_i + \frac{\partial \tau_{i,j}}{\partial x_j} \tag{2}$$

Energy equation:

$$\frac{\partial \bar{\rho}C_P\tilde{T}}{\partial t} + \frac{\partial \bar{\rho}C_P\tilde{u}_j\tilde{T}}{\partial x_j} = \frac{\partial}{\partial x_j}\left(k\frac{\partial \tilde{T}}{\partial x_j}\right) + \frac{\partial h_j}{\partial x_j} + \dot{q}_c + q_r \tag{3}$$

Mixture fraction equation:

$$\frac{\partial \bar{\rho}\tilde{Z}}{\partial t} + \frac{\partial \bar{\rho}\tilde{u}_j\tilde{Z}}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\rho D\frac{\partial \tilde{Z}}{\partial x_j}\right) + \frac{\partial M_j}{\partial x_j} \tag{4}$$

The ideal gas state equation:

$$\bar{p}_0 = \bar{\rho}R\tilde{T}\sum(\tilde{Y}_S / M_S) \tag{5}$$

where

$$\tau_{i,j} = -\bar{\rho}(\overline{u_i u_j} - \tilde{u}_i\tilde{u}_j)$$

$$h_j = -\bar{\rho}C_P(\overline{u_j T} - \tilde{u}_j\tilde{T})$$

$$M_j = -\bar{\rho}(\overline{u_j Z} - \tilde{u}_j\tilde{Z})$$

The sign "—" denotes normal filtering and "~" denotes the Favre filtering. The above expressions of $\tau_{i,j}$, h_j and M_j can be modeled as:

$$\tau_{i,j} = \mu_T\left(2\tilde{S}_{i,j} - \frac{2}{3}\tilde{S}_{k,k}\delta_{i,j}\right)$$

$$h_j = \frac{\mu_T C_P}{\mathrm{Pr}_T}\frac{\partial \tilde{T}}{\partial x_j}$$

$$M_j = \frac{\mu_T}{Sc_T} \cdot \frac{\partial \tilde{Z}}{\partial x_j}$$

where μ_T is the sub-grid scale turbulent viscosity determined by Smagorinsky model:

$$\mu_T = \bar{\rho}(C_S \Delta)^2 \left\{ 2(\tilde{S}_{i,j})^2 - \frac{2}{3}(\tilde{S}_{k,k}\delta_{i,j})^2 \right\}^{1/2}$$

where C_s is an empirical constant, Δ stands for grid scale, and

$$\tilde{S}_{i,j} = \frac{1}{2}\left(\frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i}\right)$$

Sub-grid scale turbulent thermal conductivity and material diffusivity are calculated by sub-grid scale turbulent viscosity as:

$$k_T = \frac{\mu_T C_p}{\Pr_T}, \quad \rho D_T = \frac{\mu_T}{Sc_T}.$$

The actual chemical reaction processes that control the combustion energy release are often complex. To solve those multi-step and finite-rate chemical reactions demands huge calculations. The spatial and temporal resolution limits imposed by both present and foreseeable computer resources make it impossible to get a detailed description of combustion processes. Thus, the mixture fraction model is adopted in FDS. The mixture fraction is a conserved quantity representing the fraction of material at a given point that originated as fuel. The relations between the mass fraction of each species and the mixture fraction are known as "state relations".

The local heat release rate $\dot{q}_c$ is computed from the local oxygen consumption rate at the flame surface with adopting the expression given by Huggett [8],

$$\dot{q}_c = \Delta H_O \dot{m}_O'''$$

$$-\dot{m}''' = \nabla \cdot (\rho D \frac{dY_O}{dZ} \nabla Z) - \frac{dY_O}{dZ} \nabla \cdot \rho D \nabla Z$$

where ΔH_O is the heat release rate per unit mass of oxygen consumed. The thermal radiation source term q_r in energy equation is determined by finite volume method (FVM) [9]. All spatial derivatives are approximated by second-order central differences and flow variables are updated in time using an explicit second-order Runge-Kutta scheme. The Poisson equation is solved by a direct FFT-based solver.

3. PREDICTED RESULTS AND DISCUSSIONS

3.1 Simulation of Buoyant Flow in a Shaft

In order to verify the ability of present FDS model for simulating smoke movement in a shaft, experimental data by Mercie and Jaluria [10] were used to compare with predictions. Fire-induced flow of smoke and hot gases in open vertical enclosures was investigated in their experiments. Wide ranges of the physical variables of the inlet temperature and flow rate of the hot gases were considered.

Experimental results indicate that the location and shape of outlet and smoke inlet velocity at the bottom of the shaft have remarkable effect on smoke flow pattern and temperature distribution in the shaft. Fig. 1 shows the simulated effect of inlet temperature difference on time taken by injected smoke to move from the inlet to the top opening together with the experimental data of Mercie and Jaluria [10]. In general, the comparison between the experimental data and simulation results predicted by FDS is good. With increasing temperature at the inlet, the buoyancy effect is larger, resulting in higher velocities and shorter time to reach the top.

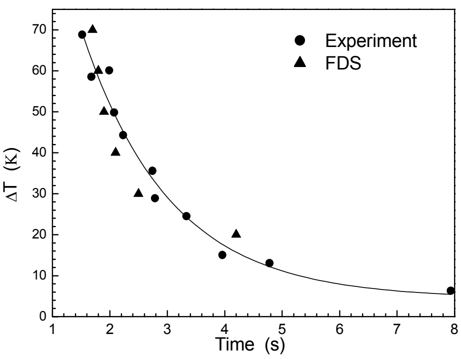


Fig. 1: Effect of inlet temperature difference on time taken by injected smoke to move from the inlet to the top opening

3.2 Predicted Results of Smoke Movement in a Shaft under Fire Scenario

A sketch of building shaft and fire scenario is shown in Fig. 2, assuming that the fire takes place in one side of corridor and the corridor is 8 m long, 1.2 m wide and 3 m high. The height of the shaft is taken as 1.2 m to 15 m with different widths of 0.6 m and 0.8 m. Leaving both sides of the corridor open, so the air outside can flow into the corridor freely. There are two partitions with the height of 1 m at the top of corridor near the inlet ports in order to prevent smoke escaping from the corridors.

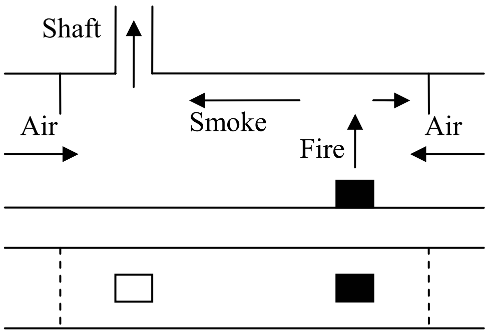


Fig. 2: Sketch of the building shaft and fire scenario

3.2.1 Simulations of the beginning of stack effect

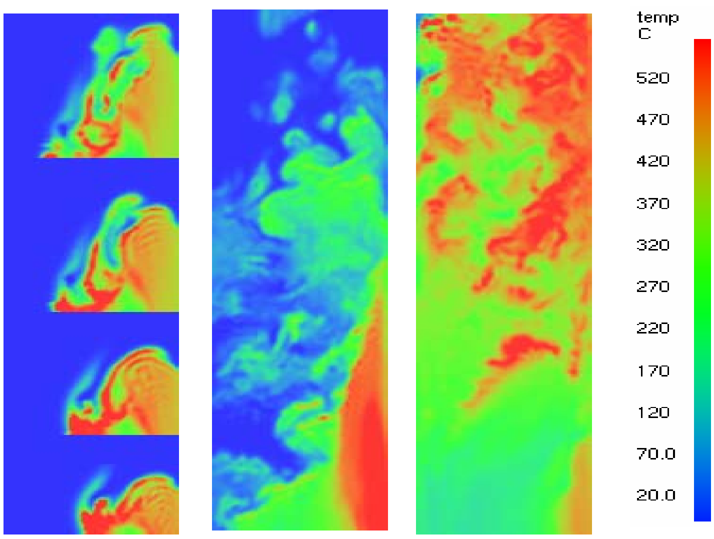
Compared with conventional methods such as Reynolds averaging approaches, large eddy simulations can predict the transient behavior of the smoke motion, more useful information about mixing process between smoke and air at the beginning of a fire can be obtained. It is beneficial to analyze the phenomena of so called stack effect when the smoke movement is in steady state.

In order to give detailed information about flow pattern of smoke motion in the shaft, fine grid size of 10 mm is adopted in the present simulation. In the case of simulating the beginning of stack effect, the shaft has an aspect (height/width) ratio of 4, in which the height of the shaft is taken as 2.4 m, and its width is 0.6 m.

Fig. 3 shows a sequence of temperature distributions at different time in the shaft at the section of Y = 0.6 m, during the initial period, simulation results show smoke vortex formation, evolution and abruption near the inlet wall of the shaft clearly. At t = 3.05 s, the hot smoke flows into the shaft. After t = 3.08 s, the smoke motion accelerates abruptly when the front eddy of smoke plume breaks up and results in a wall plume. With time increasing, the wall plume flows downstream of the entrance and goes farther into the shaft, resulting in extensive mixing (Fig. 3b). At t = 4.01 s, the smoke motion is in a nearly steady developing phase, as is shown in Fig. 3c.

3.2.2 Simulations of the steady state of stack effect

For the purpose of investigating the influence of different factors for stack effect formation, several simulation cases have been carried out considering the heat release rate variation and different geometrical sizes of the shaft. At first, with a given width of 0.6 m, two aspect (height/width) ratios of 4 and 10 are selected in the present calculations. The heat release rates change from 0.5 MW to 3.0 MW. Fig. 4 shows the predicted time-averaged temperatures and velocities affected by heat release rates at the inlet of the shaft. It can be seen that the inlet time-averaged temperature is a linear function of heat release rate, similar trend can be found in Fig. 4b, with increasing of time-averaged temperature at the inlet, the time-averaged velocities increase linearly. Another phenomenon is that the larger the aspect (height/width) ratio, the lower the time-averaged temperature at the inlet of the shaft, because the larger aspect (height/width) ratio of the shaft would cause stronger stack effect, which will result in faster motion of smoke and enhance the ambient air flowing into the corridor and the shaft, then the temperature would be lower.



(a) t=3.05s~3.08s (b) t=3.46s (c) t=4.01s

Fig. 3: Temperature distributions at different time in the shaft (at the section of Y = 0.6 m)

With a given heat release rate of 2.0 MW and a given width of 0.6 m, simulations of smoke movement are carried out when changing the aspect (height/width) ratio from 2 to 20. In order to obtain the characteristic of smoke motion affected by the height of the shaft, defining a coefficient as:

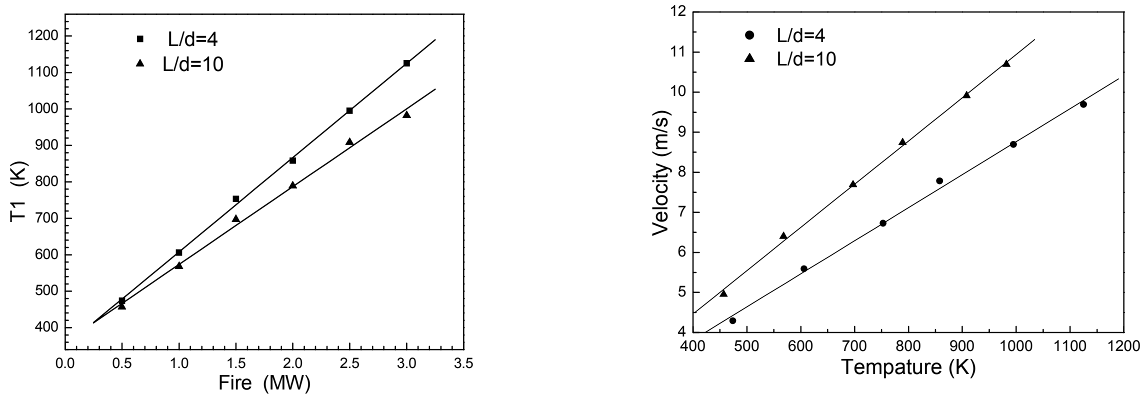
$$\theta = \frac{V_1(L)}{V_1(L=0)}$$

where $V_1(L)$ is the mean-velocity of smoke flow at the inlet of the shaft when the shaft has a height of L, which stands for the smoke volume flow rate, and $V_1(L=0)$ is also the mean-velocity of smoke flow at the inlet of the shaft but the height of the shaft is zero, that is to say there is no shaft but only a outlet at the top of corridor.

Fig. 5 shows the predicted coefficient θ when changing the height of shaft. When the aspect (height/width) ratio of the shaft is smaller than 4.0, the coefficient θ increases fast with the increasing of the aspect (height/width) ratio, whereas, when the aspect (height/width) ratio of the shaft is larger than 4.0, with the aspect (height/width) ratio increasing, the increasing trend of coefficient θ turns to slow slightly.

Fig. 6 shows effect of the aspect (height/width) ratio on time-averaged temperature under a given heat release rate of 2.0 MW and a given width of 0.6 m, with the height of shaft increasing, the mean temperature at the inlet of the shaft decreases gradually due to more ambient air flowing into the fire region in the corridor, resulting in lower temperature of whole region.

Table 1 gives predicted mean temperatures and velocities at the inlet under different geometrical sizes of the shaft. Under the same aspect (height/width) ratio, the mean temperatures and mean velocities at the inlet decrease when increasing the width of the shaft, this indicates that more ambient air flowing into the corridor due to strong stack effect.



(a) time-averaged temperatures vs. heat release rate

(b) time-averaged velocities vs. temperatures

Fig. 4: Predicted time-averaged temperatures and velocities at the inlet of shaft

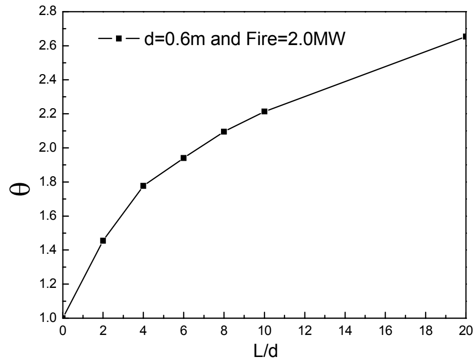


Fig. 5: Coefficient θ vs. the height of shaft

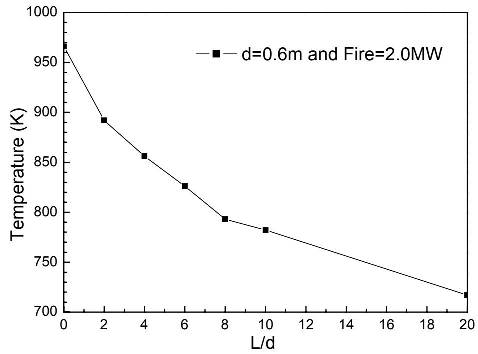


Fig. 6: Effect of aspect ratio on time-averaged temperature

Table 1: Effect of geometrical sizes of shaft on time-averaged temperatures and velocities at the inlet

parameters	L/d = 4		L/d = 10	
	d = 0.6 m	d = 0.8 m	d = 0.6 m	d = 0.8 m
T_1 (K)	858	656	789	559
V_1 (ms^{-1})	7.783	5.986	8.739	6.611

4. CONCLUSIONS

The method of large eddy simulation has been adopted to predict smoke movement in a shaft under fire scenario. Predicted results show the transient behavior of smoke vortex formation, evolution and abruption near the inlet wall of the shaft at the beginning of fire, and later turbulent mixing process. It is found that the geometrical size of the shaft has remarkable effect on the smoke motion. With increasing the aspect (height/width) ratio, smoke motion becomes faster due to stronger stack effect. Besides, with heat release rate increasing, the time-averaged temperature increases linearly. Similar trend can also be found in correlation of time-averaged velocities and temperatures under the same aspect (height/width) ratio.

ACKNOWLEDGEMENTS

Authors acknowledge the financial support provided by Ministry of Science & Technology (MOST) of China under project of National Key Basic Research Special Funds with Grant No. 2001CB409600, and Beijing Municipal Commission for Science & Technology, under Grant No. H023220050120 and No. H023620020120.

REFERENCES

1. A.A. Peppes, M. Santamouris and D.N. Asimakopoulos, "Buoyancy-driven flow through a stairwell", Building and Environment, Vol. 36, pp. 167-180 (2001).

2. A.J. Reynolds, "The scaling of flows of energy and mass through stairwells", Building and Environment, Vol. 21, No. 314, pp. 149-153 (1986).

3. A.S. Zohrabian, M.R. Mokhtarzadeh, A.J. Reynolds and B.S.T. Marriott, "An experimental study of buoyancy-driven flow in a half scale stairwell model", Building and Environment, Vol. 24, No. 2, pp. 141-148 (1989).

4. S. Kazansky, V. Dubovsky, G. Ziskind and R. Letan, "Chimney-enhanced natural convection from a vertical plate: experiments and numerical simulations", International Journal of Heat and Mass Transfer, Vol. 46, No. 3, pp. 497-512 (2003).

5. E.E. Zukoski, A review of flows driven by natural convection in adiabatic shafts, NIST-GCR-95-679 (1995).

6. J. Yao, W. Fan, S. Kohyu and K. Daisuke, "Verification and application of field-zone-network model in building fire", Fire Safety Journal, Vol. 33, No. 1, pp. 35-44 (1999).

7. W. Zhang, A. Hamer, M. Klassen, D. Carpenter and R. Roby, "Turbulence statistics in a fire room model by large eddy simulation", Fire Safety Journal, Vol. 37, No. 8, pp. 721-752 (2002).

8. C. Huggett, "Estimation of the rate of heat release by means of oxygen consumption measurements," Fire and Materials, Vol. 4, pp. 61-65 (1980).

9. K.B. McGrattan, H.R. Baum, R.G. Rehm, A. Hamins and G.P. Forney, Fire Dynamics Simulator - Technical reference guide, NIST 6467, January (2000).

10. G.P. Mercier and Y. Jaluria, "Fire-induced flow of smoke and hot gases in open vertical enclosures", Experimental Thermal and Fluid Science, Vol. 19, No. 2, pp. 77-84 (1999).