Vertical Flow Mechanism of Fire Smoke in High-rise Buildings with Internal Corridor

Xiaotao Zhang [1], Guihong Pei[1], Lei Zhu[1], Yue Ren[1]

[1] School of Architectural Economics and Engineering Management, Hubei Business College Wuhan 430079, People's Republic of China;

Corresponding author: Xiaotao Zhang, Email: smiletao@163.com

Abstract. Analyzing numerous computational fluid dynamics (CFD) simulations of a high-rise building with internal corridor, the airflow velocity, height of neutral plane and carbon monoxide concentration under different heat release rate (HRR) of fire were investigated. The result shows that with the HRR growing, the airflow velocity increases significantly, while the flow structure has no obvious change. The dependence of neutral plane height on HRR is weak. When the HRR increases from 2MW to 10MW, the height of neutral plane just increases 4.2%. The area affected by smoke containing CO is not significantly increase with the HRR growing. However, the CO concentration increases remarkably, which indicates that rapid evacuation is crucial for the people on the higher floor.

1. Introduction

With the rapid process of urbanization, huge numbers of people rushed into metropolitan areas, which causes an ever-increasing demand for residential space in high-rise buildings. Therefore, numerous high-rise buildings have been constructed in urban areas, especially in China. According to the Council on Tall Buildings and Urban Habitat (CTBUH) [1], the 144 buildings completed in 2017 beat every previous year on record. By 2015, the number of constructed super high-rise building in China has been leading the world for 8 years. Although the growing number of high-rise building could meet the needs of urban development and ease the pressure of housing resources, it would increase the difficulty level in fire control and evacuation. In the past few years, building fires have occurred frequently (according to fire statistics, about 60% of accidental fires occur in buildings) and caused tremendous loss of life and property [2-6]. When a fire is occurred in a typical high-rise building, smoke released during the fire can migrate along the roof and then buoyancy forces can drive it into the upper floor through shaft structure and it will decrease the visibility, which is also very bad in an emergency. Statistical data shows that 85% of the people killed in building fires are killed by toxic smoke [7]. Therefore, vertical flow mechanism of fire smoke in high-rise buildings has received increasing attentions.

Many studies have been conducted to investigate fire smoke movement in high-rise building. Miller and Beasley studied the stack effect and pressurization systems for stairwells designed to prevent smoke from spreading into the stairwell [8]. Shi et.al studied the mechanisms of smoke movement in emergency staircase through a set of experiments in a scaled building model with 12 floors [9]. Li et.al investigated the characteristics of fire smoke induced buoyant plume movement in a 12-storey stairwell with three vents though a set of experiment which is conducted in a 1/3 scale stairwell [10].

In general, the high-rise building, especially office shape, is relatively commonly used the

rectangular shape, the layout is often the internal corridor (hereafter the internal corridor layout is named IC layout) in order to make full use of construction space as much as possible, but the IC layout is adverse to ventilation and day lighting, which causes greater potential fire hazard exists in this kind of buildings. However, little attention has been focused on the smoke vertical flow mechanism in the kind of high-rise buildings. To investigate smoke movement in high-rise building with internal corridor layout, a set of CFD simulation was conducted in selected building which is a 9-floors-staircase configuration with internal corridor. Smoke velocity, range of movement and temperature, were analyzed.

2.   CFD simulations

In this study, the smoke movement analysis was based on a series of CFD simulations by the FDS software, which was released by the National Institute of Standards and Technology (NIST). In these simulations, different values of HRRs are considered. The Navier–Stokes equations for fire-driven fluid flow are solved by LES, which is second-order accurate with respect to space and time differences. The governing equations for smoke flowing in a fire are the conservation laws of mass, momentum and energy. Simplified equations and other detailed information regarding to FDS simulator, the user's guide in [11] can be referred.

Our research purpose is to study the vertical flow characteristic in high-rise building with IC layout.In general, when fire break out in building, the room doors connect with internal corridor should be close in order to prevent the toxic smoke flow into the room. So there are no needs to consider the effect of the rooms. Figure 1 presents the actual building with IC layout and the model configuration used in our simulation. The dimension of the 9-storey modeled building section is 18.2m×3.6m×27.2m (L×W×H), which is consisted of two main sections, the stairwell and the internal corridor. Each floor is 3.0m high. The window at each floor has a surface area of 1.0 m (height) × 1.0 m (width). Each floor has one door with the surface area of 2.4 m (height) × 2.0 m (width) to connect stairwell. In the stairwell, there are two windows located in the first and the ninth floor conectet with the outside world.

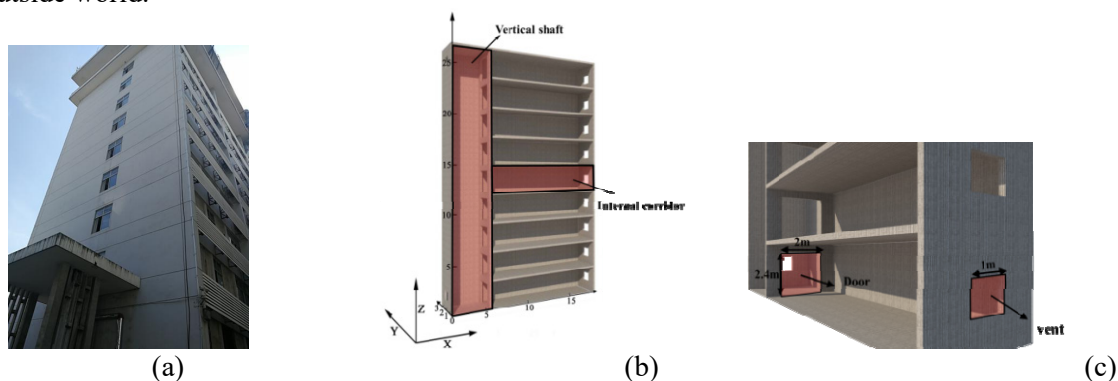

| (a) | (b) | (c) |

Figure 1. Model configuration: (a) High-rise buildings with internal corridor; (b) Geometry of simulation model;(c) internal corridor model.

Environmental temperature is 25 °C. Square pool fires were used as fire sources, which was located in the first floor and the distance to window is 6.5m. The length was 1.0m and the heptane was used as fuel. According to the study by Shi et.al, smoke temperature in stairwell is mainly determined by HRR of fire source [9]. In order to analyze the influence of HRR on smoke movement, three different HRR, which is 2MW, 5MW and 10MW respectively, were used with fire power

In general, getting accurate data is at cost of computation time. To get accurate simulated data with high efficiency, grid sensitivity should be analyzed before simulating. Three grid systems were used in our FDS simulation study:

- Coarse grid: $Nx \times Ny \times Nz = 0.3\text{m} \times 0.3\text{m} \times 0.3\text{m}$
- Medium grid: $Nx \times Ny \times Nz = 0.2\text{m} \times 0.2\text{m} \times 0.2\text{m}$
- Fine grid: $Nx \times Ny \times Nz = 0.1\text{m} \times 0.1 \text{ m} \times 0.1\text{m}$

The time-dependent temperatures at the middle of the ninth floor were recorded in the FDS results. Figure 2 presents the comparison of FDS predictions with three different grid resolutions.

As indicated by figure 2, the differences between the coarse grid and fine grid are significant. While refining from medium grid to fine grid, the predicted values changed a little. As the simulation time of using fine grid is six times that of medium grid, the medium grid was used in this study.
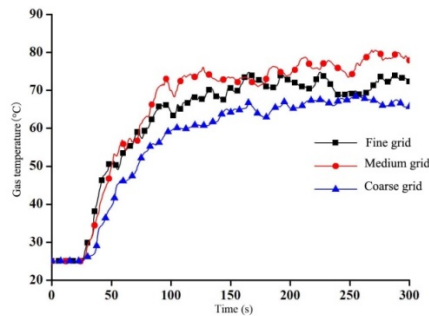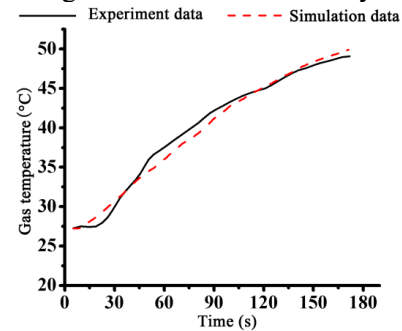

Figure 2. Mesh sensitivity analyses

Figure 3. Comparison of smoke temperature

Pr and Sc, together with C_S, are the most important parameters in the simulation of fire-induced transportation of smoke, especially in the prediction of smoke temperature. According to the study by Zhang et.al [13], in our simulation, the values of Pr , Sc and C_S are set to be 0.5, 0.5 and 0.18 respectively.

To test and verify whether all of the settings in the FDS were correct, the simulation results were compared to the experimental data in Luo et al study [2], in which the corridor sectional dimension of experimental model model is equal to the simulated model based on the similarity principle. The results, shown in figure 3, shows that there was good agreement between the simulation and the experiment. The conclusion was that all the settings in the simulations were appropriate. So the same FDS model was used in the following study.

3. Results and Discussions

3.1. Velocity distribution in the building model

The airflow structure in the high-rise building model under different HRR is illustrated by the mean velocity contour at the mid-plane (see figure 4) and the airflow velocity curves measured at each floor vent are displayed in figure 5.
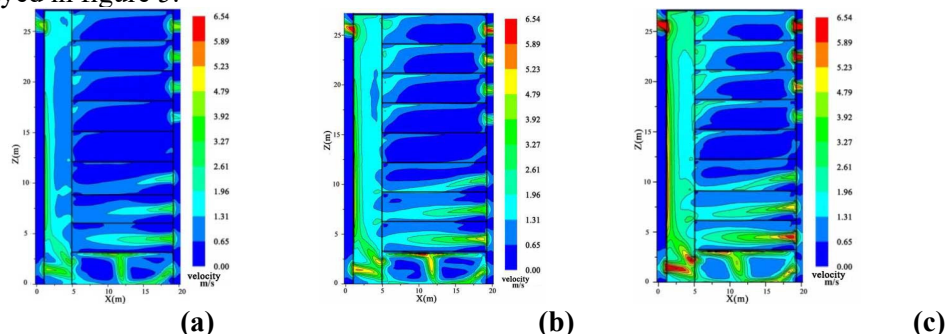

(a)   **(b)**   **(c)**

Figure 4. Mean velocity contour of central plane(Y=1.8m):(a) HRR=2MW;(b) HRR=5MW;(c) HRR=10MW.

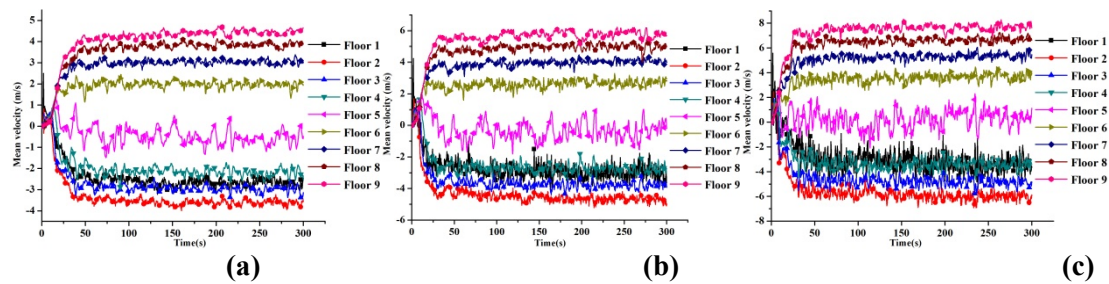**(a)**   **(b)**   **(c)**

Figure 5. Velocity of airflow into vertical shaft: (a) HRR=2MW;(b) HRR=5MW;(c) HRR=10MW.

As shown in figure4a, in the case when HRR is 2MW, a distinct interface exists in the fifth floor. The air will flow into the building through the vents below the fourth floor and smoke will flow out from windows above to the sixth floor. On the fifth floor, the mean air velocity is close to zero (showed in figure 5a). With heat release rate of fire source increasing, the airflow velocity at each floor is significantly increased except the fifth floor. On the fifth floor, the mean air velocity remains close to zero. These results indicate that the HRR of fire source can dramatically impact on the air and the smoke velocity. However, the influence of HRR on the flow structure is small, thus the smoke spreading area is roughly keep same with increasing of fire HRR.

3.2. Height of the neutral plane
In general, when smoke flowing into the stairwell, the temperature inside will be higher than outside, and the normal stack effect will form. Inside the building, a horizontal plane exists where the pressure inside equals to the one outside, which is called the neutral plane. Above the neutral plane, the air pressure inside is higher than that outside and the hot smoke will flow out from the vent above the neutral plane, which seriously threaten the people on the upper floor [10].Therefore, the height of the neutral plane can represent the range of smoke spreading. In order to accurately analyze the range of smoke spreading under different HRR, the height of the neutral plane will be determined by analyzing the height of zero pressure plane. Figure 6 shows the pressure contour at middle plane when HRR is 2MW and the zero pressure planes can be determined by collecting the pressure data which stored in 'slice file'. Similarly, the height of zero pressure planes under other HRR can be determined. Figure 7 shows the height of neutral planes development under different HRR.
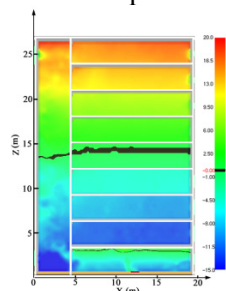

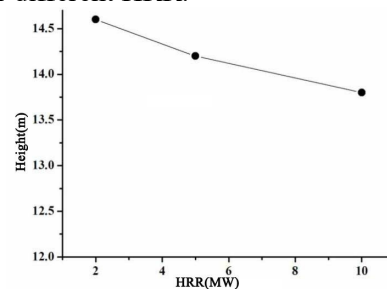Figure 6. Mean pressure contour at middle plane (Y=1.8m) when HRR is 2MW.

Figure 7. The height of neutral plane for various heat release rates.

It could be seen that the height of zero pressure plane increase insignificantly with the HRR growing. Compared to the height of that under 2MW, when the HRR increased by 5 times, which is 10 MW, the height of that is just decrease by 4.2 %. This result indicated that the dependence of neutral plane height is weak. The range of smoke spreading is increase with the HRR growing, but the increasing of that is not remarkable. Therefore, the threats to people on the lower floor increase slightly compared to the one on upper floor.

3.3. Concentration of smoke and CO
A comparison of the range affected by smoke in the model is shown in figure 8. As seen in figure 8, the affected area is not significantly increase with the HRR growing, while the smoke concentration
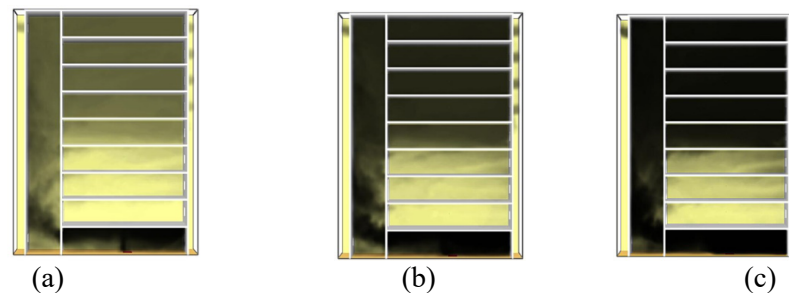
increases obviously.

(a)   (b)   (c)

Figure 8. The area affected by smoke for various heat release rates: (a) HRR=2MW; (b) HRR=5MW; (c) HRR=10MW

(a)   (b)   (c)
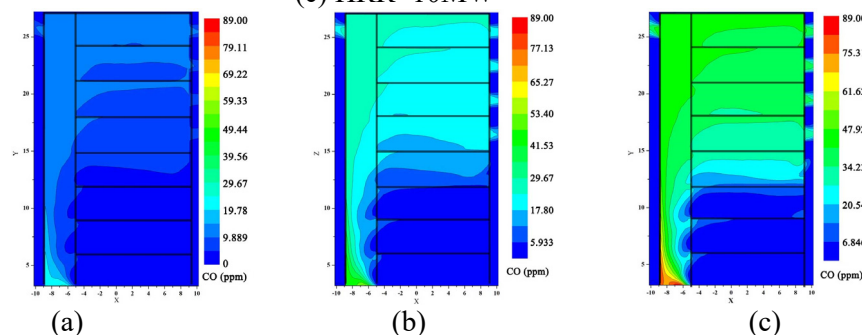
Figure 9. Mean CO concentration contour at central plane(Y=1.8m) for various HRR: (a) HRR=2MW; (b) HRR=5MW; (c) HRR=10MW

As we all know, CO is one of the major causes of death in a building fire. To further analyze the influence of smoke under different HRR, the CO concentration and the range of movement are discussed. Figure 9 describes the field distribution of the CO concentrations from the floor 2 to 9 under three different HRR. It is can be seen from figure 9, the area affected by CO just increase slightly. However, the CO concentration increases significantly in the affected area. Based on the result, it was concluded that with the HRR of fire increasing, the fire threats to the people on lower floors are would not significantly increase, while for higher floors, the threats would increase significantly. Therefore, rapid evacuation is crucial for the ones on higher floor.

4.   Conclusion

Bench numerous CFD simulations using FDS were carried out to study the vertical flow mechanism of fire smoke in high-rise buildings with internal corridor. HRR of fire source can dramatically impact on the airflow velocity. With the HRR growing, the airflow velocity increase significantly. However, the HRR has no prominent influence on the flow structure. The range of smoke spreading is increase not remarkable with HRR growing. Neutral plane can represent the range of smoke spreading. The height of neutral plane just increases 4.2% when the HRR increase from 2MW to 10MW, which indicate the dependence of height of neutral plane on HRR is weak. The area affected by smoke containing CO is not significantly increase with the HRR growing. However, in the affected area, the CO concentration increases significantly. Therefore, rapid evacuation is crucial for the people on higher floor.

References

[1]   The Skyscraper Centre, The Global Tall Building Database of the CTBUH. [Online], Available: http://skyscrapercenter.com, Last accessed on June 14th, 2018.

[2]   Luo, N., et al., An experiment and simulation of smoke confinement and exhaust efficiency utilizing a modified Opposite Double-Jet Air Curtain. Safety Science, 2013. 55: p. 17-25.

[3]   Zhang, B., et al., Buoyancy-driven flow through a ceiling aperture in a corridor: A study on smoke propagation and prevention. Building Simulation, 2015. 8(6): p. 701-709.

[4]   Niu, H., et al., Experimental study and numerical simulation of spread law for fire on tunnel.

           Journal of Central South University, 2015. 22(2): p. 701-706.

[5]   Yuen, A.C.Y., et al., Fire scene investigation of an arson fire incident using computational fluid dynamics based fire simulation. Building Simulation, 2014. 7(5): p. 477-487.

[6]   Luo, N., et al., Smoke confinement utilizing the USME ventilation mode for subway station fire. Safety Science, 2014. 70: p. 202-210.

[7]   Hu, L.H., et al., Confinement of fire-induced smoke and carbon monoxide transportation by air curtain in channels. Journal of Hazardous Materials, 2008. 156(1-3): p. 327-334.

[8]   Miller, Richard S., Beasley, Don. On stairwell and elevator shaft pressurization for smoke control in tall building. Building and Environment 44, 2009. p.1306–1317.

[9]   Shi, W.X., et al., Influence of fire power and window position on smoke movement mechanisms and temperature distribution in an emergency staircase. Energy and Buildings, 2014. 79: p. 132-142.

[10] Li, L.J., et al., Experimental investigation on the characteristics of buoyant plume movement in a stairwell with multiple openings. Energy and Buildings, 2014. 68: p. 108-120.

[11] K. McGrattan, M. Randall, Fire dynamics simulator (version5) user's guide NIST Special Publication 1019-5, 2010.

[12] Luo, N., et al., An experiment and simulation of smoke confinement utilizing an air curtain. Safety Science, 2013. 59: p. 10-18.

[13] Zhang, W., Hamer, A., Klassen, M., Carpenter, D., Roby, R.,. Turbulence statistics in a fire room model by large eddy simulation. Fire Safety Journal 2002.37: p. 10-18.