Contents lists available at ScienceDirect



International Journal of Heat and Mass Transfer

journal homepage: www.elsevier.com/locate/ijhmt

Numerical investigation of smoke dynamics in unconfined and confined environments



HEAT and M

Chan-Sol Ahn^{a,b,1}, Boo-Hyoung Bang^{a,c,1}, Min-Woo Kim^a, Tae-Gun Kim^a, Scott C. James^d, Alexander L. Yarin^{e,*}, Sam S. Yoon^{a,*}

^a School of Mechanical Engineering, Korea University, Seoul 136-713, Republic of Korea

^b Fire Research Center, Korea Institute of Civil Engineering and Building Technology, 283, Goyang-daero, Ilsanseo-gu, 10223 Goyang-si, Republic of Korea

^c Technology Development Team, Daewoo Inst. of Construction Tech., 60, Songjuk-dong, Jangan-gu, 440-210 Suwon, Republic of Korea

^d Depts. of Geosciences and Mech. Eng., Baylor University, Waco, TX 76798-2534, USA

e Department of Mechanical and Industrial Engineering, University of Illinois at Chicago, 842 W. Taylor St., Chicago, IL 60607-7022, USA

ARTICLE INFO

Article history: Received 26 March 2018 Received in revised form 11 July 2018 Accepted 11 July 2018

Keywords: Fire Dynamics Simulator Buoyant smoke dynamics High-rise buildings Wall effects

1. Introduction

ABSTRACT

Because of their implications to safety, the study of plume dynamics in high-rise buildings is a research area of interest to building engineers. In this study, the temperature, velocity, and pressure of smoke rising in buildings of various sizes were considered as functions of fire size, and were simulated using the Fire Dynamics Simulator software. Numerical results were validated against the analytical solutions for confined (building enclosure) and unconfined (open-air) systems. As the building area decreased and the fire size increased, buoyancy-driven flow accelerated and the overall building temperature increased. Additionally, the low pressure at the bottom of the building, which resulted from buoyant smoke, increased the vertical pressure gradient throughout the building. These parametric investigations can be used by building engineers concerned with smoke dynamics to develop design-safety guidelines.

© 2018 Elsevier Ltd. All rights reserved.

When building fires occur, buoyancy forces rapidly lift the smoke to higher floors or cause it to spread across the ceilings, thus creating a life-threatening situation for occupants. The increased buoyancy also increases the rate of fresh air inflow through the lower building floors, and this fresh air fans the flames as illustrated in Fig. 1. Therefore, building safety engineers are interested in how much and how fast smoke rises. The intensity of the fire can be assessed if the rate of fresh air inflow is known. Risks to building occupants can be investigated as a function of the fire size and building size. The volume, concentration, and thermo-physical properties of the smoke can be modeled using plume theory based on analyses within the framework of self-similar solutions. In other words, the properties of smoke can be modeled as remote asymptotics, which are formally valid at distances much larger than the fire source size. Therefore, the focus in this study was placed on the dynamics of a plume produced by a small that fire spreads throughout a large-scale building.

* Corresponding authors.

The theory of self-similar laminar plumes was first developed by Zeldovich [1] (cf. the well-known monographs of Jaluria [2] and Yarin [3] and the review of Turner [4]). Self-similar solutions for turbulent plumes based on Prandtl mixing length theory have also been reported [3]. Apart from self-similar solutions, plume rise due to buoyancy has been extensively investigated. Morton et al. [5] analyzed convection from point sources for variable-density stratified fluids. Conner et al. [6] presented an empirical evaluation of the optical properties of plumes over a broad range of illuminating and viewing conditions. McCaffrey [7] used thermocouples and an impact probe to measure smoke temperature and velocity around a purely buoyant diffusion flame issued from a porous refractory burner. Baum et al. [8] performed a theoretical investigation of the velocity and temperature distributions of fire plumes using large eddy simulations. Papanicolaou and List [9] measured the axial and radial velocities of a buoyant, turbulent plume using laser-Doppler anemometry, while also measuring the smoke concentration using laser-induced fluorescence. Chen [10] theoretically and experimentally investigated the flow in buoyant, turbulent plumes. Shabbir and George [11] reported comprehensive hot-wire measurements of an axisymmetric buoyant plume injected vertically into a quiescent environment. Sangras and Faeth [12] reported theoretical and experimental work on symmetric, non-buoyant jets and puffs, and evaluated the buoyant and thermal properties of

E-mail addresses: ayarin@uic.edu (A.L. Yarin), skyoon@korea.ac.kr (S.S. Yoon). ¹ Equal contributors.



Fig. 1. Schematic of hot air rising because of fire-induced buoyancy.

plumes under quiescent, non-stratified conditions. Kaminski et al. [13] investigated entrainment into buoyant, turbulent jets, and presented the buoyancy profiles as a function of the axial distance from the point source. Carazzo [14] scrutinized the inherent assumption of the classical self-similar approach for plumes and jets and demonstrated that self-similarity occurs earlier in plumes than in nonbuoyant jets. Sun et al. [15] evaluated the properties of wildfire smoke using the Fire Dynamics Simulator software. Tanaka [16] experimentally investigated the properties of smoke from an unconfined fire. Hu et al. [17] experimentally and numerically investigated the properties of smoke from unconfined, full-scale fires. Ji et al. [18] evaluated an experimental stairwell fire and described the corresponding smoke dynamics. Laser induced fluorescence (LIF) and particle image velocimetry (PIV) data for a transitional buoyant plume above a horizontal cylinder were obtained by Grafsronningen and Jensen [19]. Theoretical investigations of the interaction between the rising plume and the counter current of cold air under open-air conditions were reported by Wang et al. [20].

The abovementioned studies were mostly concerned with smoke-plume dynamics under open-air (unconfined) conditions. Even though there are numerous studies of fires within confined spaces, plume and fire properties cannot be directly compared because of differences with regard to length-scale and thermophysical properties [21]. The study of confined plumes has been limited to a real-scale prototype experiment and simulation with a limited fire size [18,22]. The effect of a high-rise shaft's temperature distribution on a rising plume was investigated by Qi and Wang [20,23]. Plume rise in a shaft has been numerically modeled by Xue et al. [24]; however, such studies have been limited to the fixed size of the shaft and fire. There is a lack of knowledge with regard to how the sizes of the building and fire influence the smoke-plume characteristics. To the best of our knowledge, this aspect has not been investigated to date. In this study, we characterized the vertical smoke and plume flows according to variations in building and the fire size, and by using three-dimensional numerical simulations. The numerical results obtained by the Fire Dynamics Simulator software were first verified by comparison to the dynamics predicted by the self-similar plume theory. Then, they were validated against experimental data.

2. Theoretical and numerical models

2.1. Plume jet theory

According to Yarin [3], for the axisymmetric turbulent plumes that are of primary interest here, the most important parameter governing the flow is the released power, $Q_Z(W)$, or its counterpart $Q_Y = Q_Z/(2\pi\rho c)$, where ρ and c are the density and specific heat at constant gas pressure, respectively. In an axisymmetric plume, the following relationship holds:

$$Q_Y = \int_0^\infty u(T - T_\infty) y \mathrm{d}y,\tag{1}$$

where *u* is the longitudinal velocity profile in the plume, *T* is the temperature distribution in the plume, T_{∞} is the temperature of the surrounding gas, and *y* is the radial coordinate in any plume cross-section measured from its axis (the vertical axis *z*). Note that Q_Y (and hence Q_Z) is always an input parameter.

The maximum longitudinal cross-section velocity along the rising plume is u_{max} . In a turbulent axially symmetric plume, u_{max} is estimated as [3]:

$$u_{\max}(z) = \left(\frac{\beta g Q_z}{\rho c z}\right)^{1/3},\tag{2}$$

where β is the gas thermal expansion coefficient and g is gravity.

Similarly, the axial temperature (i.e., at its cross-sectional maximum) in such a plume is [3]:

$$T_{\max}(z) - T_{\infty} = \left(\frac{Q_z}{\rho c}\right)^{2/3} \frac{1}{\left(\beta g\right)^{1/3}} \frac{1}{z^{5/3}}.$$
(3)

If a shaft provides air from intermediate floors to a rising plume, the volumetric flow rate in the shaft will increase similar to that of a free plume:

$$\dot{Q} = \int_0^\infty u y dy \tag{4}$$

Accordingly, the self-similar solution becomes:

$$\dot{Q} = \left(\frac{\beta g Q_Z}{\rho c}\right)^{1/3} z^{5/3} \times \text{constant}$$
(5)

2.2. Fire Dynamics Simulator

The Fire Dynamics Simulator (FDS) software (version 6.6.0) is used to solve the Navier-Stokes equations for low-speed flows (Mach number, Ma < 0.3) [25] and is also appropriate for using thermallydriven plumes. Turbulence was considered a large eddy simulation (LES). Combustion was modeled by mixture-fraction analyses, which assumed immediate reaction of fuel and oxygen. Additionally, radiation transport was based on a non-scattering gray gas and wide-band model. The governing equations included continuity, species-concentration balance, momentum and energy balances, and the ideal gas law [26]. A full description of the model was reported previously by the authors [25].

3. Results and discussion

3.1. Model validation

Prior to parametric investigations, the accuracy and applicability of the numerical simulations were validated by comparing to the experimental data of McCaffrey [7]. McCaffrey identified three zones for a smoke-producing fire: flame, intermediate, and plume zones, which were identified along the axial direction with the velocities and temperatures in each zone were scaled in relation to the heating power (Q) and axial location (z). The proposed scaling laws are:

$$\frac{u}{Q_Z^{1/5}} = k \left(\frac{z}{Q_Z^{2/5}}\right)^\eta,\tag{6}$$

$$\Delta T = \frac{T_0}{2g} \left(\frac{k}{C}\right)^2 \left(\frac{z}{Q_Z^{2/5}}\right)^{2\eta-1},\tag{7}$$

where *C* is a buoyancy constant, *k* is a coefficient for the centerline correlation of various dimensions, η is the exponent of the centerline correlation, and ΔT is the temperature increase. These constants and coefficients are summarized in Table 1.

The following heating powers were used for the parametric runs in the open-air system: $Q_z = 14.4$, 21.7, 33.1, 44.9, and 57.5 kW, in accordance with the experimental data presented by McCaffrey [7]. The fire had a diameter of 0.3 m. A computational domain of $2 \times 2 \times 2$ m³ with a computational mesh of $40 \times 40 \times 80$ (0.128 M nodes) was used. The smallest mesh size of $\Delta x = \Delta y = 0.0026$ m was used at the center, and the mesh was stretched to the boundary such that $\Delta x = \Delta y = 0.138$ m. A uniform mesh was used in the

Table 1

Summary of the centerline data [7].

	k	η	$\frac{z}{Q_z^{2/5}} \left[\frac{m}{kW^{2/5}} \right]$	С
Flame	6.8 $\left[\frac{m^{1/2}}{s}\right]$	1/2	<0.08	0.9
Intermediate	1.9 $\left[\frac{m}{s \cdot kW^{1/5}}\right]$	0	0.08-0.2	0.9
Plume	1.1 $\left[\frac{m^{4/3}}{s \cdot k W^{1/3}}\right]$	-1/3	>0.2	0.9

axial direction with $\Delta z = 0.025$ m. All numerical data were timeaveraged from t = 20 to 600 s to capture the steady-state conditions and for comparison to McCaffrey's empirical fit [7].

Fig. 2a and b shows the time-averaged centerline temperature and axial velocity, respectively, as functions of the height above the fire at z = 0 for various fire sizes with Q_z ranging from 14.4 to 57.5 kW. McCaffrey's scaling factors for z and u were $Q_z^{2/5}$ and $Q_z^{1/5}$, respectively. These factors were empirically determined based on log–log plots of u and $\Delta T = T - T_{\infty}$ with respect to z. The coefficient η was selected to collapse all data in each zone, i.e., the flame, intermediate, and plume zones.

The obtained numerical results appropriately matched the empirical fit for *T* and *u* in all zones. The flame zone shown in Fig. 2a was flat ($\eta = 0$) and retained a high flame temperature. A drop in temperature began in the intermediate zone ($\eta = -1$) and became more pronounced in the plume zone ($\eta = -3/5$). The increasing temperature drop was due to cooling associated with the mixing of smoke with newly entrained fresh air from the exterior along with an increase in the distance from the heat source. As shown in Fig. 2a, the simulation over-predicted *T* in the flame and intermediate zones. The Fire Dynamics Simulator used mixture-fraction and single-step chemical-reaction models that assumed a uniform mixture of fuel and oxidizer over the combustion area. Thus, the flame intensity exceeds that of real combustion wherein the mixing between the fuel and oxidizer was not perfect.

In Fig. 2b, the simulated u matched the empirical fit and revealed the three distinct zones. The smoke accelerated in the flame zone where the buoyancy was maximum (highest temperatures and lowest gas densities). Outside of the flame zone, the rising smoke lost momentum when it interacted with the surrounding air that was entrained into the expanding smoke volume. Eventually, significant momentum re-distribution was observed in the expanding plume zone as more air was entrained. The FDS model, based on the sub-grid-scale (SGS) model, assumes momentum exchange in all directions. However, in reality, momentum exchange in the axial direction is dominant. This discrepancy explains why the simulations under-predicted the results relative to the empirical fit. Overall simulated T and u values were in agreement with the experimental data, while only slightly overpredicted T and under-predicted u. Hence, the model was considered appropriate for use in parametric investigations.

3.2. Model verification

A grid-convergence study was carried out for a CH₄ fire with Q_z = 10 MW in a $12 \times 12 \times 40 \text{ m}^3$ open-air domain with grid



Fig. 2. Centerline (a) ΔT vs. z scaled by $Q_z^{2/5}$; (b) u scaled by $Q_z^{1/5}$ vs. z scaled by $Q_z^{2/5}$.

resolutions of $64 \times 64 \times 800$ (3.28 M nodes), $64 \times 64 \times 400$ (1.64 M nodes), $64 \times 64 \times 200$ (0.82 M nodes), and $64 \times 64 \times 100$ (0.41 M nodes). The fire source size was 2.1 m in diameter. Cartesian grids were used in the x and y directions with the smallest cells $\Delta x = \Delta y = 0.04$ m, at the center and expanding to $\Delta x = \Delta y =$ 0.78 m near the boundary. In the axial direction, the finest resolution was $\Delta z = 0.05$ m, and was increased to $\Delta z = 0.1, 0.2, \text{ and } 0.4$ m for the grid-convergence study. The results indicated that 1.64 M nodes were appropriate for refinement. All data were acquired at the centerline at an axial location of z = 20 m. All simulations were conducted for t = 100 s and the time-series variations of u. T, and the volumetric flow-rate \dot{O} , were compared for all grid resolutions as shown in Fig. 3. The bottom surface of the domain was specified as a no-flux, perfectly insulated wall with a no-slip condition, while all of the other boundaries were open for air to enter or exit through all of the lateral walls. The initial air temperature and pressure were fixed at T_{∞} = 20 °C and P_{∞} = 1 bar, respectively.

Fig. 3 shows a comparison of the computational results for *u*, *T*, and \dot{Q} , for grid resolutions of $\Delta z = 0.05$, 0.1, 0.2, and 0.4 m. Both *u* and *T* were under-predicted by the coarsest grid ($\Delta z = 0.4$ m) because insufficient resolution smoothed the detailed information, which in turn reduced the overall *u* and *T* values. Both *u* and *T* converged for $\Delta z = 0.05$ and 0.1 m. So a grid resolution of $\Delta z = 0.1$ m was selected. With regard to the volumetric flowrate, the coarsest grid of $\Delta z = 0.4$ m yielded the largest fluctuation and magnitude, and convergence at $\Delta z = 0.05$ and 0.1 m is shown in the figure. In Fig. 3d, pressure is not strongly sensitive to grid resolution.

3.3. Effects of building cross-sectional area

There are four forces that drive plume flow in a high-rise building [27]. First, the heat from the fire increases buoyancy. Second, the temperature difference between the inside and outside of the building yields a pressure gradient between the top and bottom of the building. Additionally, the external winds and the internal heat, ventilation, and air cooling (HVAC) system can affect the smoke dynamics. In this study, we considered only fire-driven buoyancy. The smoke dynamics were influenced by the size of the fire and the cross-sectional area (size) of the building, crosssectional areas of the inlet and outlet, heat transfer through walls, wall roughness, and intermediate flow inlets and outlets such as windows and hallways. To consider all of these features would require hundreds of simulations, which is impractical. Therefore, this study only considered variations in fire and building size.

According to the National Fire Protection Association (NFPA) guideline, a building is considered high-rise if its height is exceeds 100 m. Thus, building height was 100 m in all cases. The fire size varied from 2 to 20 MW based on an NFPA guideline. A typical fire load for a small-scale office fire is $20-120 \text{ kg/m}^2$. For a room size of 10 m², 200–1200 kg of combustible wood was assumed to be adequate. The heat from the combustion of wood was approximately 20 MJ/kg and the fire released energy in the range of 4000–24,000 MJ. Assuming a duration of 4 h, the heating power was 0.28–1.7 MW, and ten times as much for a large-scale fire.

Fig. 4 shows the 3D computational domains used in the parametric investigations. The buildings were rectangular and their crosssections varied from 25 to 1600 m². An unconfined (open) system was also simulated. The computational domain was much larger than that described in the previous section. The smallest grid-cell size was $\Delta x = \Delta y = 0.04$ m along the centerline and stretched to $\Delta x = \Delta y = 1$ m near the outer boundaries. A uniform mesh was used in the axial direction with $\Delta z = 0.2$ m. The maximum number of grid cells was $100 \times 100 \times 500$ (5 M nodes). For the open-air system, free-stream Neumann boundary conditions were imposed on all



Fig. 3. Grid-refinement results for (a) u, (b) T, (c) \dot{Q} , and (d) P at z = 10 m. The methane flame delivered 10 MW at z = 20 m and the surrounding temperature was T_{∞} = 20 °C.



Fig. 4. 3D computational domain for open-air system and buildings with cross-sectional areas ranging from 25 to 1600 m². Building height was 100 m in all cases and heating power was $Q_z = 10$ MW.



Fig. 5. Centerlines (a) *T*, (b) *u*, (c) \dot{Q} , and (d) *P* as functions of building size.

sides, while a no-slip boundary condition was imposed at the bottom. For the buildings, walls were assigned no-flux and no-slip conditions. Air at 20 °C was allowed to enter the building vertically through the open bottom boundary and exit vertically through the open top boundary. The initial air temperature and pressure were $T_{\infty} = 20$ °C and $P_{\infty} = 1$ bar, respectively. The fire source was CH₄ with a heating power of $Q_z = 10$ MW. The radiative heat source (instead of CH₄) provided 10 MW from a pan with a diameter of 2.1 m, which was located at the center of the building's base. Building crosssectional areas were 1600, 400, 100, and 25 m². Steady-state results averaged over 580 s from 20 to 600 s are presented. The solid curves in Fig. 5a, b, and c are the plume characteristic predictions obtained by Eqs. (3), (2), and (1) for *T*, *u*, and \dot{Q} , respectively. As shown in Fig. 5a, the simulated open-air data approximated the solution obtained by the self-similar plume theory, which assumed open-air conditions. As the building area decreased, the numerical solutions deviated from self-similar plume theory. Because of fresh-air entrainment, smoke temperatures decreased with distance from the heat source at *z* = 0. However, as the cross-sectional area of the building shrank, the temperature drop became less severe. For a cross-sectional area of 25 m², plume temperature remained approximately 80 °C for

z > 60 m. Fig. 5b shows u_{max} different cross-sectional areas. The FDS simulation was in good agreement with the theoretical predictions of the open-air system obtained by Eq. (2). As the crosssectional area of the building narrowed, u_{max} behaved similarly to T as the momentum dissipated with increasing distance from the heat source. However, the decrease was least for the smallest building cross-sectional area. The small building acted like a pipe wherein flow was distributed rather uniformly across the lateral direction. Fig. 5c compares \dot{O} values for the various systems. The flow-rate increased with z as a significant volume of air was entrained into the buoyant plume. However, building walls constrained the air volume that could be entrained (unlike the open case). As the cross-sectional area increased, the total flow-rate increased; however, its variation with z was minimal (unlike the open case). As shown in Fig. 5b, *u* increased as the building area decreased, even when Q_z was constant. Because $\dot{Q} \propto \textit{uA}$, the increase in u and decrease in A competed to yield the resulting flow-rate as the building size decreased. In Fig. 5c, building size (A) dominated and flow-rate increased regardless of the decrease in u.

Fig. 5d shows how *P* varied with *z* for the different building sizes. For both the open-air system and the largest building, the pressure drop was approximately zero. However, as the building size decreased, a low-pressure region formed near the heat source and the air inflow increased as a result. For the smallest building size, P = -62 Pa at the bottom of the building (atmospheric pressure was 101 kPa). Thus, this low pressure at the building bottom was small, but indicated that the velocity was maximum at the low-pressure area, according to the Bernoulli principle.

The u(y) profiles shown in Fig. 6a and b (for the open and 1600 m² cases) exhibited a decrease from the maxima at the centerline (y = 0) to zero at the no-slip walls. Along the axial direction, u(y) transformed from a near step function just over the heat source with a diameter of 0.67 m to a parabolic distribution because of thermal and eddy diffusivity. The velocity profiles in the open-air system were approximately equal to those of the 1600 m² building (Fig. 6a and b). As the building size decreased to 400 and 100 m² (Fig. 6c and d), the plume velocity profile increasingly resembled the velocity profile of turbulent flow in a pipe. For all building sizes the maximum velocities ranged from 12 to 13 m·s⁻¹ near the heat source. However, the building size clearly altered the u(y) distributions of the plumes.

3.4. Effect of fire size

Fig. 7 shows the effect of fire size in buildings of various crosssectional areas (1600, 400, 100, and 25 m²) on the plume temperature as compared to McCaffrey's empirical fit [7]. To change the heating power of the fire, its physical diameter was changed from 0.96 to 1.52, 2.14, and 3.03 m to yield 2, 5, 10, and 20 MW heat sources, respectively. Recall that the model results were timeaveraged over 580 s along the centerline of the computational domain. In the figures, the horizontal axis extends to $z/Q_z^{2/5} \approx 4$, which corresponds to a maximum axial location of $z_{max} = 100$ m.

Fig. 7a compares the theoretical solution obtained with Eq. (3) to the empirical fit. Significant deviation was observed in the flame zone because self-similar theory is a remote asymptotic only applicable in the intermediate and developed plume zones; its use is not



Fig. 6. Lateral profiles *u*(*y*) for (a) an open-air system and buildings with cross-sectional areas of (b) 1600, (c) 400, and (d) 100 m².



Fig. 7. Effect of fire size in buildings with different cross-sectional areas (1600, 400, 100, and 25 m²) on plume T. The analytical solution and numerical simulations are compared to McCaffrey's empirical fit [7].

appropriate in the near field. The self-similar theory predicts an infinite temperature at z = 0, which is typically resolved by introducing the polar distance in the same way as in the theory of turbulent jets [28]. However, an excellent prediction was made in the plume zone where the accuracy of the theory increased with increasing distance from the point heat source. This excellent prediction was made for all fire sizes of $Q_z = 2$, 5, 10, and 20 MW

Fig. 7b–f compare the numerical solutions obtained by the FDS with the empirical fit for all heating values. Fig. 7b shows the numerical solutions for the open-air system. In the numerical simulations, the fire size increased with the heating power. Thus, the applicability of the far-field, self-similar theory decreased. Conse-

quently, the numerical solution performed well in the flame zone (Fig. 7b) while the self-similar theory over-predicted the temperatures in this zone (Fig. 7a). In the intermediate and plume zones, there was excellent agreement between the theoretical and numerical solutions.

Fig. 7c compares simulated temperatures in the largest building (1600 m^2) to those of the empirical fit. Because the building cross-sectional area was sufficiently large, the plume behavior was similar to that of the open-air system. Thus, Fig. 7b and c are quite similer.

When the building size was reduced to 400 m² (Fig. 7d), the plume temperature remained fairly constant. This indicated that

the building did not constrain the smoke temperature when *z* was scaled by $Q_z^{2/5}$. However, it did not indicate that there was no influence from the building. In fact, plume dynamics and *T* changed with building size. However, the scaling applied to *z* with $Q_z^{2/5}$ collapsed the temperature distributions over a wide range of Q_z . Therefore, the temperature was not strongly affected by the building size. When the building size was further reduced to 100 and 25 m² (Fig. 7e and f), clear deviations from the empirical fit were observed. As the building size shrank, the plume rose faster because of the continuity equation (mass balance). As shown in Fig. 7e and 7f, velocity increases enhanced plume cooling and reduced ΔT .

Fig. 8 compares axial velocities (*u*) for various fire and building sizes. According to McCaffrey [7], *u* scaled by $Q_z^{1/5}$ and *z* scaled by $Q_z^{2/5}$ collapsed all the data that corresponded to the various heating powers. With the exception of small buildings (100 and 25 m²), this scaling was appropriate.

As shown in Fig. 8a, the smoke velocities were accurately predicted in the plume zone by theoretical Eq. (2). However, they were over-predicted in the intermediate and flame zones. Again, this occurred because self-similar plume theory is a remote asymptotic and valid only in the plume zone. Because a fire has finite size, it was expected that the simulations would deviate from the selfsimilar theory in the flame and intermediate zones. Note that the



Fig. 8. Effect of fire size in buildings with different sizes (1600, 400, 100, and 25 m²) on plume axial velocities. The analytical solution and numerical simulations are compared to McCaffrey's empirical fit [7].



Fig. 9. Temperature fields for each building.

vertical velocities were lower near the heat source ($z \approx 0$) while they increased toward the intermediate zone. The velocities decreased again in the plume zone as the entrained air sapped momentum.

Fig. 8b compares of the FDS numerical data to those of the empirical fit under open-air conditions. There is good agreement between the data in all three zones. For the 1600 m^2 building shown in Fig. 8c, the simulations were similar to those of the

open-air system, with the exception of a slight increase in the velocity caused by the faster flow in a confined space. As shown in Fig. 8d, the simulations departed from the empirical fit, particularly for the high- Q_Z cases (5, 10, and 20 MW). The 2 MW case had the lowest axial velocity because of the low buoyancy force. Despite the deviations, the velocity patterns (increasing in the flame zone and decreasing in the plume zone) in the simulation and empirical fit were consistent. These results indicated that the



Fig. 10. Velocity fields for each building.

constraints imposed by building size do not dominate the smoke dynamics, although their effects were more obvious with larger heat sources. the velocities for all heat sources in the two smallest buildings were fairly uniform in the lateral direction.

3.5. Temperature, velocity, and pressure distributions

For the smaller buildings (Fig. 8e and f), building size dominated smoke dynamics such that the flame zone was strongly influenced by a faster flow through the confined space (100 and 25 m²). For the 100 m² building, the smoke achieved a velocity of $u/Q_z^{1/5} \approx$ 1.6 at $z/Q_z^{2/5} \approx 0.01$ in the 2 MW case. As expected in pipe flow,

Fig. 9 qualitatively compares the temperature distributions for the different fire and building sizes (the maximum contour was T = 100 °C). The high-temperature region extended to higher



Fig. 11. Pressure fields for each building.

elevations as the fire and building size decreased. For the open-air system and the 1600 m² building with smaller fire sizes ($Q \le 5$ MW), the smoke properties were approximately the same. However, for the 20 MW fire, the open-air system had a wider smoke plume, whereas in the 1600 m² building, the smoke plume was narrowed by the walls constraining the flow. For the 2 MW heat source, the maximum temperatures (not shown) were approxi-

mately the same for all building sizes and temperature distributions where z > 50 m was safe against thermal damage because T < 35 °C. With the 20 MW heat source, the thermal-damage region ($T \ge 100$ °C) extended beyond 40 m in the open-air system. For the smallest building size, even z = 100 m was subject to thermal damage. Fig. 10 shows the axial smoke velocities for the different fire and building sizes. Smaller buildings resulted in faster velocities, while the velocities increased with the size of the heat source. The velocity profile was approximately uniform across the smallest building with the largest fire. Fig. 11 shows the pressure distributions with the different fire and building sizes. Similar to the *T* and *u* distributions, the pressure distributions of the open-air system and the 1600 m² building were approximately the same. When the building area decreased, the pressure decreased at the inlet and this increased the air inflow. For the 25 m² building, the pressure was lower than the atmospheric pressure for *z* < 75 m, even with the smallest heat source.

4. Conclusion

The smoke dynamics in building fires were numerically investigated using the Fire Dynamics Simulator software. The temperatures, velocities, and pressures of the buoyant plumes were simulated and compared to those obtained by the analytical selfsimilar solution for unconfined (open-air) plumes. For fires in buildings, the numerical simulations were compared to an empirical fit obtained by an experiment conducted under unconfined conditions. To visualize the effect of the building cross-sectional area, deviations of the numerical simulations from the empirical data were quantified. The fire and building sizes were varied to investigate their effect on the smoke dynamics. As the building size decreased and the fire size increased, the smoke was increasingly buoyant and dispersed more uniformly throughout the building. When the building was sufficiently large, the smoke behaved similarly to an unconfined smoke plume. These quantitative and qualitative parametric investigations elucidated the effect of building size on the dynamics of building fires. Additionally, they can provide useful information to building engineers involved in firesafety design. Notably, because obstacles in the buildings were not considered, the complexities arising from the stairs and windows in actual buildings were not represented. Although the simulations presented in this paper may not be directly applicable to an actual building fire, they can be used as guidelines to address the effects of building and fire sizes on smoke dispersion, which had not been investigated numerically or experimentally prior to this study.

Conflict of Interest

The authors declared that there is no conflict of interest.

Acknowledgement

This research was supported by ICT & Future Planning (NRF-2016M1A2A2936760, NRF-2017R1A2B4005639, NRF-2013R1A5A 1073861) and the Technology Development Program to Solve Climate Changes of the National Research Foundation (NRF) funded by the Ministry of Science of South Korea. This work was also supported by the National Research Council of Science & Technology (NST) grant by South Korea government (MSIP) (No. CRC-16-02-KICT).

References

- Y.B. Zeldovich, Limiting laws of freely rising convection currents, Zhurnal Eksperimentalnoii Teoreticheskoy Fizika 7 (1937) 1463–1465.
- [2] Y. Jaluria, Natural Convection: Heat and Mass Transfer, Pergamon Press, Oxford, 1980.
- [3] L.P. Yarin, The Pi-Theorem: Applications to Fluid Mechanics and Heat and Mass Transfer, Springer-Verlag, Berlin Heidelberg, 2012.
- [4] J. Turner, Buoyant plumes and thermals, Annu. Rev. Fluid Mech. 1 (1) (1969) 29-44.
- [5] B. Morton, G. Taylor, J. Turner, Turbulent gravitational convection from maintained and instantaneous sources, in: Proceedings of the Royal Society of London A: Mathematical, Physical and Engineering Sciences, The Royal Society, 1956, pp. 1–23.
- [6] W.D. Conner, J.R. Hodkinson, Optical Properties and Visual Effects of Smoke-Stack Plumes, NAPCA Publication, DHEW, 1967.
- [7] B. McCaffrey, Purely buoyant diffusion flames: Some experimental results, in: Final Report, Chemical and Physical Processes in Combustion, The National Institute of Standards and Technology (NIST), Miami Beach, 1979, p. 49.
- [8] H.R. Baum, K.B. McGrattan, R.G. Rehm, Three dimensional simulations of fire plume dynamics, Fire Safety Sci. 5 (1997) 511–522.
- [9] P.N. Papanicolaou, E.J. List, Investigations of round vertical turbulent buoyant jets, J. Fluid Mech. 195 (1988) 341–391.
- [10] H.-B. Chen, Turbulent buoyant Jets and Plumes in Flowing Ambient Environments, Department of Civil Engineering, University of Aalborg, 1991.
- [11] A. Shabbir, W.K. George, Experiments on a round turbulent buoyant plume, J. Fluid Mech. 275 (1994) 1–32.
- [12] R. Sangras, G. Faeth, Buoyant turbulent jets and plumes: III. Round turbulent nonbuoyant starting jets and puffs and buoyant starting plumes and thermals, National Institute Standard and Technology Report No. GDL/GMF-99-03, 1999.
- [13] E. Kaminski, S. Tait, G. Carazzo, Turbulent entrainment in jets with arbitrary buoyancy, J. Fluid Mech. 526 (2005) 361–376.
- [14] G. Carazzo, E. Kaminski, S. Tait, The route to self-similarity in turbulent jets and plumes, J. Fluid Mech. 547 (2006) 137–148.
- [15] R. Sun, M.A. Jenkins, S.K. Krueger, W. Mell, J.J. Charney, An evaluation of fireplume properties simulated with the Fire Dynamics Simulator (FDS) and the Clark coupled wildfire model, Can. J. For. Res. 36 (11) (2006) 2894–2908.
- [16] T. Tanaka, T. Fujita, J. Yamaguchi, Investigation into rise time of buoyant fire plume fronts, Int. J. Eng. Perform.-Based Fire Codes 2 (1) (2000) 14–25.
- [17] L. Hu, W. Chow, R. Huo, Y. Li, H. Wang, Comparison of rise time of free smoke plume front predicted by a field model with full-scale experiments, Fire Saf. Sci. 6 (2004), 2b-1–1.
- [18] J. Ji, L. Li, W. Shi, C. Fan, J. Sun, Experimental investigation on the rising characteristics of the fire-induced buoyant plume in stairwells, Int. J. Heat Mass Transf. 64 (2013) 193–201.
- [19] S. Grafsrønningen, A. Jensen, B.A.P. Reif, PIV investigation of buoyant plume from natural convection heat transfer above a horizontal heated cylinder, Int. J. Heat Mass Transf. 54 (23–24) (2011) 4975–4987.
- [20] X. Wang, S. Shen, G. Du, Y. Dai, Interaction mechanism model for a nonisothermal air jet and a buoyant plume, Adv. Mech. Eng. 9 (6) (2017), 1687814017702805.
- [21] R. Wasson, M.N. Nahid, B.Y. Lattimer, T.E. Diller, Influence of a ceiling on fire plume velocity and temperature, Fire Technol. 52 (6) (2016) 1863–1886.
- [22] M. Li, Z. Gao, J. Ji, K. Li, Modeling of positive pressure ventilation to prevent smoke spreading in sprinklered high-rise buildings, Fire Saf. J. 95 (2018) 87– 100.
- [23] D. Qi, L. Wang, R. Zmeureanu, The effects of non-uniform temperature distribution on neutral plane level in non-adiabatic high-rise shafts during fires, Fire Technol. 53 (1) (2017) 153–172.
- [24] L. Xue, S. Yuan, Q. He, A comparative study on the influence of ventilation on weather-and fire-induced stack effect in the elevator shafts of a high-rise building, Fire Technol. 54 (1) (2018) 163–186.
- [25] K.B. McGrattan, H.R. Baum, R.G. Rehm, A. Hamins, G.P. Forney, J.E. Floyd, S. Hostikka, K. Prasad, Fire Dynamics Simulator–Technical Reference Guide, National Institute of Standards and Technology, Building and Fire Research Laboratory, 2000.
- [26] R.G. Rehm, H.R. Baum, The equations of motion for thermally driven, buoyant flows, J. Res. Nat. Bar. Stand. 83 (1978) 297–308.
- [27] D. Drysdale, An Introduction to Fire Dynamics, John Wiley & Sons, 2011.
- [28] G. Abramovich, The Theory of Turbulent Jets, MIT Press, 1963, p. 75.