The 9th Asia-Oceania Symposium on Fire Science and Technology

Numerical investigation on the effects of human movements on smoke propagation in building fire

Zhuyang Han*, Wenguo Weng, Quanyi Huang

Institute of Public Safety Research, Department of Engineering Physics, Tsinghua University, Beijing 100084, China

Abstract

This work aims at analyzing the dynamics mechanism of airflow and fire spread, especially for the effects of human movements on smoke propagation. A numerical investigation method composed of fire parameters and a smoke propagation analysis is proposed in this paper. The diversifications of flow field when human moves are the main part of the smoke propagation analysis, which is based on the dynamic mesh method of computational fluid dynamics. The numerical results show that the movements of human may cause great changes to the velocity and temperature fields in the building. The high temperature and toxic gas follows the human’s movement and lead to a great change to the risk distribution in the risk region by promoting the heat transfer effects of convection between the gases with different temperatures. These movements also lead to a farther spread of the cool air that getting through the inlet window, which decreases the average temperature in the room. These effects should be taken into the consideration for the formulation of evacuation routes.

© 2013 International Association for Fire Safety Science. Published by Elsevier Ltd. All Rights Reserved
Selection and peer-review under responsibility of the Asian-Oceania Association of Fire Science and Technology

Keywords: Human move; Building fire; Risk assessment; Smoke propagation

1. Introduction

With fire development and smoke propagation, exposure to toxic smoke is one of the hazards confronting people in fires [1]. High-temperature, smoke, carbon monoxide and carbon dioxide can affect the safety of the occupants by reducing the capability of judging, acting and memory [2-4]. These effects will finally change the evacuation behaviours and rescue mission. Recently, more and more authorities are aware of the security problems in high-rise buildings. For years, many significant works have been made on this issue, especially on the dynamic mechanism of fire spread and smoke propagation, and some life safety and risk evaluation models were also developed [2, 5-8]. Some investigations have studied the connection between fire and human behaviour evacuation model [9-11], but few are about the dynamic mechanism of the occupants’ performance and its effects to the smoke propagation [7]. In fact, when the trapped people try to escape from a building which is on fire, the movements may cause significant changes to the flow field of fire smoke. So the effects of human movement are also extraordinary for the evaluation of the flow field in the building, and the diversification of velocity and temperature fields may cause the evaluation result of evacuation route extremely different. So that is the reason why it is very important to analyse the dynamics mechanism and effects of human’s movements in buildings. The evaluation result will be a great help for the formulation of evacuation route and safety management.

This work focuses on the effects of human movements to smoke propagation in building fires. A numerical investigation method is proposed in this paper to quantitatively assess flow field distribution and its diversification when human moves. The results of this method include velocity and temperature fields that can be used in risk evaluation. In the second section

* Corresponding author. Tel.: +86 138 1164 6262; fax: +86 10 6279 2863.
E-mail address: hanzhuyang@gmail.com.
of this paper, the method is presented. In the third section, the application of this method is demonstrated by using a sample building for numerical investigation. The numerical results are also discussed in Section 3, followed by the conclusions.

2. Method

This section proposes a numerical method for the investigation of the dynamics effects of human movements on smoke propagation. This method is composed of fire parameters and a smoke propagation analysis. The fire parameters show the detailed information of the fire source. These parameters can be used as the boundary conditions of the numerical simulation of combustion and smoke propagation process. The smoke propagation analysis aims at analyzing the flow field of fire. The initial flow field can be analyzed in static state. The effects of human’s movements on fire dynamics can be obtained by using the unsteady-state analysis, and then the diversification of flow field can be analyzed by comparing the results of steady-state and unsteady-state analysis. And then analysis results can be used in the prediction of fire field trends. This prediction is necessary and useful for the formulation and prediction of evacuation route in safety management.

2.1. Fire parameters

The fire parameters include combustible material, air inlet condition, combustible matter content, oxygen content, and combustion mode. The combustible can be methane, gas, or furniture. The air inlet condition is always as windows or pipeline ventilation system. The combustible matter content is the volume fraction of combustible material when it is in gaseous state. The oxygen contents denote the volume fraction of oxygen in air inlet. The combustion modes include two types: steady-state and unsteady-state. The steady-state assumes that the combustion process lasts long time enough and is in static, which means the temperature field and the velocity field do not change with time. This parameter can be used for the initial flow field analysis before human moves. When human moves, the flow field will change with time and the unsteady-state assumption is used to analyze the diversification of flow field.

2.2. Smoke propagation analysis

The smoke propagation and flow field can be analysed by using the computational fluid dynamics (CFD) method. For many years, CFD method has been widely used in flow field simulation and aerodynamics research. Nowadays, CFD method and software are also used in fire accident simulation and smoke propagation simulation [11-17]. The CFD method can quantitative calculated the airflow field, temperature field and chemical reaction such as combustion. The calculation accuracy can also be improved by increasing the iterations number until the accuracy of the results meets the requirements.

In this work, Ansys 12.1.4 is used for CFD calculation. Turbulence model, energy equation, combustion mode and corresponding models are used for the analysis of gas flow, energy transport and chemical reaction. The models chosen in the CFD method are shown in Table 1.

<table>
<thead>
<tr>
<th>Solver Formulation</th>
<th>Viscous Model</th>
<th>Species Model</th>
<th>Model</th>
<th>Model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure based</td>
<td>k-epsilon (2 eqn) model</td>
<td>model</td>
<td>Species transport</td>
<td></td>
</tr>
<tr>
<td>Implicit</td>
<td>Standard</td>
<td>Reactions</td>
<td>Volumetric</td>
<td></td>
</tr>
<tr>
<td>3D</td>
<td>Standard wall functions</td>
<td>Options</td>
<td>Inlet diffusion</td>
<td></td>
</tr>
<tr>
<td>Absolute</td>
<td>Turbulence-chemistry interaction</td>
<td>Eddy-dissipation</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Green-gauss cell based</td>
<td>Diffusion energy source</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Superficial velocity</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

For the initial flow field analysis before human moves, the steady-state assumption and steady-state solver are chosen. The unsteady-state assumption and unsteady-state solver are used when human moves. And the dynamic mesh method is
used for the dynamic analysis of human’s movements. In the dynamic mesh method, the layering model is used. When the human moves, the zone in front of the human is minifying and the zone behind the human is extending.

3. Applications and discussions

The sample building used here is a room in a building. The room is 6.0 m length, 3.5 m width and 3 m height. It contains eight openings including five air inlet openings and three outflow openings. Fig. 1(a) shows the structure of this room. A man is standing at one end of the room facing the opposite wall. The structure of the human is obtained by scanning the shape structure of a real respiratory thermal manikin in. There are five air inlet openings and three outflow openings in this room. All of the openings are 1.0 m width and 0.5 m height. Three of the air inlet openings are placed on the left wall of the human, 2.25 m height from the ground. The other two air inlet openings placed on the front and back wall in the room with a height of 1.25 m from the ground. All the outflow openings are on the right side of the human, 0.25 m above the ground. The fire source locates on the ground right under an air outflow opening, on the right side of the human. The fire source is assumed as a gas pipeline outlet with 0.1 m length and 0.1 m width.

![Diagram](image)

Fig. 1. The structure of the sample building. (a) Three-dimensional view of the building structure, (b) Top view (XY plane) of the sample building, (c) Side view (YZ plane) of the sample building, (d) Side view (XZ plane) of the sample building.

For the opening conditions, the gravity is taken into consideration. According to the needs of the calculation in Computational Fluid Dynamics, the room is meshed by grids. The profiles of the grids are shown in Fig. 1 (b)–(d). For the human body, the profile of the respiratory thermal manikin is very complicated especially in the face and figures, so the space around the human is meshed by unstructured grid of tetrahedron which can fit the profile of the respiratory thermal manikin better. The interval size in this space is 0.03 and the total number of elements is 231,069. Other parts of the sample room are meshed by structured grid of hexahedral. The interval size is 0.05 and the total number of elements is 492,000.

As for the boundary conditions, the velocity of the air inlet is 1 m/s and the volume fraction of oxygen in the air is 0.21. This is to simulate the ventilation facilities as an air supplier in a real room and providing enough air (oxygen) from outside. For the fire source, the combustible material is set to methane gas. The velocity is 5.0 m/s with a volume fraction of methane is 0.05. The temperature of both air inlets and methane gas inlet are 300 K. And for the outflow, all of the three
openings are general ventilation window, and the flow rate weighting is 1. The temperature of the human body remains 310 K. All of the boundary conditions are shown in Table 2.

Table 2. The boundary conditions of the sample building

<table>
<thead>
<tr>
<th>Zone</th>
<th>Area (m × m)</th>
<th>Velocity (m/s)</th>
<th>Volume fraction</th>
<th>Temperature (K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>air inlet</td>
<td>1.0 × 0.5</td>
<td>1</td>
<td>oxygen: 0.21</td>
<td>300</td>
</tr>
<tr>
<td>outflow</td>
<td>1.0 × 0.5</td>
<td></td>
<td>flow rate weighting: 1</td>
<td></td>
</tr>
<tr>
<td>methane gas inlet</td>
<td>0.1 × 0.1</td>
<td>5</td>
<td>methane gas: 0.05</td>
<td>300</td>
</tr>
<tr>
<td>Human</td>
<td></td>
<td></td>
<td></td>
<td>310</td>
</tr>
</tbody>
</table>

The operation conditions of the numerical simulation are as follows:
A. Since the ventilation facilities in the building works well, oxygen is enough in the room. So the combustion process is almost entirely complete combustion. Hardly any carbon monoxide may generate in the smoke and be transmitted by airborne in the building.
B. All the windows in the building are keeping open. The ventilation facilities in the rooms keep running.
C. Before the human moves in the room, the combustion process last long enough time that the flow field and temperature field is steady in the room.
D. When the human moves, the combustion process and flow field changes with the movements of human body. So does the temperature field.
E. The respiratory process of the move human is ignored.
F. The human physiological responses to high-temperature and toxic gas are ignored.
G. The origin of coordinate located at the left corner of the human on the ground.
H. All the boundary conditions remain unchanged when human moves.
I. The walls and the human body are combustible.

Fig. 2. The temperature and velocity fields distribution around the human body of the steady-state combustion (K, m/s). (a) Top view (XY plane, Z=1.1 m) of the sample building, (b) Side view (YZ plane, X=1.0 m) of the sample building, (c) Side view (XZ plane, Y=0.45 m) of the sample building.
The numerical simulation of the combustion process can be achieved by using the CFD method. The results of temperature distribution and velocity field distribution can be visual displayed to show the detailed status of the fire and flow fields, as shown in Fig. 2. Figs. 2(a1), (b1), (c1) show the temperature distribution around the human body, and Figs. 2 (a2), (b2), (c2) show the velocity field distribution around the human body. In Fig. 2(a2), the vectors stand for the velocity in X and Y direction of XY plane and the counters stand for the Z direction velocity. In Fig. 2(b2), the vectors stand for the velocity in Y and Z direction of YZ plane and the counters stand for the X velocity. In Fig. 2(c2), the vectors stand for the velocity in X and Z direction of XZ plane and the counters stand for the Y velocity. As shown in Figs. 2(a1), (b1), (c1), the combustion process in the room developed sufficiently, since oxygen and methane are fully mixed around the fire source. With the increasing in gas temperature, the density of mixed gas decreases then the gas mixture moves upward. Higher plane has a higher temperature because of the upward movements of high temperature flue gas in this room. The temperature field in this room is also affected by the cool air getting through the inlet openings, which promotes the propagation of smoke in the room as shown in Figs. 2 (a2), (b2), (c2). After the cool air entering the room, it sinks and moves forward close the ground. And then, the cold air is heated by the fire and moves upward across the room, which finally results in the air convection in this room, as shown in Figs. 2(c1) and (c2). When the flow field is in steady state, the temperature distribution and velocity distribution remain stable.

By using the dynamic mesh method, the human moves follow the positive direction of Y-axis with a velocity of 1m/s. After moving forward for 5 seconds, the human stops at the other side of the room. The numerical simulation results are shown in Figs. 3–6. Fig. 3 shows the velocity field distribution in XY plane when human moves (Z=1.1 m, m/s). The vectors stand for the velocity in X and Y direction of XY plane and the counters stand for the Z direction velocity. Fig. 4 shows the velocity field distribution in YZ plane when human moves (X=1.0 m, m/s). The vectors stand for the velocity in Y and Z direction of YZ plane and the counters stand for the X direction velocity.

![Velocity Field Distribution](image)

Fig. 3. The velocity field distribution in XY plane when human moves (Z=1.1 m, m/s): (a) t = 1.0 s, (b) t = 2.0 s, (c) t = 3.0 s, (d) t = 4.0 s, (e) t = 5.0 s, (f) t = 6.0 s.
By comparing Figs. 3–4 with Fig. 2(a2), it can be seen that the velocity field is changed significantly. The human’s movements make the flow field more unstable. In $XY$ plane, the airflow follows the human’s movements and cyclones occur right behind the human body, as shown in Figs. 3(a)–(e). These cyclones make the cool air that getting through the inlet opening spread further. In $YZ$ plane, the sinking airflow goes downwards and follows the human’s movements towards the moving direction. For the $Z$ direction velocity, the human’s movement lead to a uniform velocity field distribution. The sinking airflow in the middle part of the room is disturbed and divided into several parts. It is indicated that when the human moves, the human body are followed by the air that around the human. This movement and the following effect promote the gas propagation and change the flow field distribution in the room. When the human stops moving, the flow fields will not become stable immediately. The airflow keeps moving follow the same direction and promotes the smoke propagation in the room. So the effects of human’s movements on airflow field last much longer than the time human moves.

Figures 5 and 6 give the temperature distribution in $XY$ ($Z=1.1$ m, K) plane and $YZ$ plane ($X=1$ m, m/s) when human moves, respectively. In $XY$ plane, by comparing Figs. 5–6 with Fig. 2 (b2), it can be seen that the cool air that gets through the air inlet openings diffuses faster and further than before. So the cyclones behind the human body promote the air convection in the room. This effect makes the average temperature in the room decreases significantly. In $YZ$ plane, the high-temperature gas move follows the human’s movements and mixes with the cool air that gets through the air inlet opening. The high-temperature gas keeps moving and become thinner gradually. And it’s close to the ground until separated by the sinking cool air. This effect also makes the average temperature in the room decreases significantly. Only the high-temperature gas around the ceiling in this room is not affected so much by the human’s movements. It is indicated that the diversification of flow field changes the temperature distribution. When human moves, the air around the human follows his movements and the high-temperature gas spreads forward right behind the human. The movements also lead to a farther spread of the cool air that getting through the inlet window, which decreases the average temperature in the room. And the movements also promote the heat transfer effects of convection between the gases with different temperatures.
Fig. 5. The temperature distribution in XY plane when human moves (Z=1.1 m, K): (a) τ = 1.0 s, (b) τ = 2.0 s, (c) τ = 3.0 s, (d) τ = 4.0 s, (e) τ = 5.0 s, (f) τ = 6.0 s.

Fig. 6. The temperature distribution in YZ plane when human moves (X=1 m, K). (a) τ = 1.0 s, (b) τ = 2.0 s, (c) τ = 3.0 s, (d) τ = 4.0 s, (e) τ = 5.0 s, (f) τ = 6.0 s.
As for the smoke propagation analysis and fire risk evaluation, the human physiological responses to high-temperature and toxic gas are not included in this work. When a trapped people tries to get through a corridor full of fire smoke, he will inhale toxic gas and get burned by the high-temperature smoke. Then the physiological function of the human gets weakened. The loss of the physical ability or even the ability of judgement may lead to the failure of escape. So, taking the human physiological responses to high-temperature and toxic gas into the consideration of the human behavior in fire can do a great help to the formulation of evacuation routes.

4. Conclusions

In this paper, a numerical investigation method for building fire is proposed. This method is composed of fire parameters and a smoke propagation analysis. The distribution of velocity field and temperature field are quantitative calculated by using the CFD method. The effects of human movements on smoke propagation are analysed by comparing the diversifications of flow field when human moves. The flow field distribution including temperature field and velocity field before human moves is calculated by using the steady-state method, and when the human moves it is calculated by using the unsteady-state method. The diversification of flow field distribution can be analysed by comparing the results of these two methods. The results show that the movements of human may cause great change to the velocity field and temperature field in the building. The high temperature and toxic gas follow the human’s movement and lead to a great change to the risk distribution in the risk region.

In conclusion, this method is an appropriate method for the quantitative analysis of flow field distribution in building fire considering human movement. The dynamic characteristics and effects of human movements on smoke propagation can be also assessed successfully by applying the method in unsteady state. These results should be taken into the consideration for the formulation of evacuation routes. So this method can be used in practical application, and it will be an important technical basis for the program development of rescue and evacuation.

Acknowledgement

This paper was supported by National Natural Science Foundation of China (Grant No. 51076073) and the National Basic Research Program of China (973 Program) (Grant No. 2012CB719705). The authors deeply appreciate the supports.

References