Smoke spreading along corridor induced by outdoor wind: numerical simulation and parameter sensitivity analysis

Aijuan WANG¹, Brady MANESCAU¹, Khaled CHETEHOUNA¹ and Charles de IZARRA¹

¹INSa Centre Val de Loire, University Orléans, PRISME, EA 4229, F-18020 Bourges, France

aijuan.wang@insa-cvl.fr

ABSTRACT

During a fire in building of high or low height, 80% of deaths are caused by the inhalation of smoke. Considering the fact that outdoor wind is one of the important driving forces of the smoke movement of the fire, this work investigates its effects on the dynamics of smoke along corridor induced by compartment fire. This study is carried out using numerical simulation based on experimental data. The computational fluid dynamic code used is FDS. Through the analysis of mesh resolution, it is shown that the numerical results can agree well with the experimental data. Simulations are performed with different wind speeds. It can be seen that the outdoor wind has effects on the smoke stratification. Indeed, the more wind speed increase, the more the smoke stratification is disturbed. Under these conditions, smoke occupies entire corridor volume and thus presents a high risk of toxicity to people. It can be found that the temperature of the smoke increases suddenly and decreases to converge to a homogeneous value in the entire enclosure. Considering the importance of the input data used for the numerical simulation, a sensitivity analysis is made on parameters related to the material, fuel, combustion and extinction models. This analysis is carried out using a local sensitivity analysis method. It turns out that mass flux of fuel and activation energy as well as thermal conductivity of the walls have significant effects (more than 10%) on the smoke temperature near the ceiling.

KEYWORDS:
Smoke velocity, FDS, corridor, outdoor wind, sensitivity analysis
1 INTRODUCTION

In recent years, the number of high-rise building has been increasing rapidly in the world. While it brings convenience to people, it represents a major challenge for fire safety engineering due to the larger number of victims during fire. Thus, fire safety engineering of high-rise building has drawn many research interests. The researches [1][2] have shown that smoke is the main cause of death in fires due to the toxicity. Therefore, it is essential to study the behavior of smoke spreading inside the building. The behavior of smoke spreading can be affected by thermal buoyancy, expansion, wind speed, heat pressure and so on. For high-rise building, the wind velocity increases significantly with the height of the building and influences the smoke spreading along corridor in fires.

Therefore, researches have to be continued. Some experimental tests have already been done but they are very expensive and can not study many parameters. Several studies have been carried out on this issue [3][4]. In ref.[5], it is shown that the influence of the external velocity on the propagation of the smoke, and the influence of the fire power on the level of smoke stratification. In order to reduce the costs of implementing the experimental devices and to better understand the phenomena related to the smoke flow in an enclosure, numerical studies have been carried out[6][7][8].

Most of the numerical simulations focus on the propagation of smokes in a ventilated or not ventilated enclosure where the level of smoke stratification varies as a function of temperature profile or velocity. However, these studies do not take into account the effects of the outdoor wind on the smoke stratification in a corridor adjacent to a burning room with an opening. Using the data obtained by Li et al. [4], the aim of the present paper is to highlight the ability of Fire Dynamics Simulator (FDS 6) to reproduce the behavior of the smokes induced by outdoor wind.

Considering uncertainties in definition of input parameters in numerical modeling and uncertainties related to experimental measurements, a sensitivity analysis is applied in order to determine the effect of input parameters on the output [9][10].

In this work, a CFD approach is proposed to evaluate the effects of outdoor wind on the smoke spreading induced by an adjacent compartment fire. This paper is organized as follows: Section 2 presents the physical and numerical modeling. Section 3 deals with numerical results and sensitivity analysis on the results and Section 4 contains conclusions and perspectives.

2. PHYSICAL AND NUMERICAL MODELLING

2.1. Computational domain and boundary conditions

The computational domain used in this study represents 1/3 scale horizontal corridor with an internal dimension of 5.5 m (length) × 0.7 m (width) × 0.9 m (height) connected to a room having a dimension of 2.0 m (length) × 1.7 m (width) × 1.0 m (height) (cf. Fig. 1). The dimension of the door connecting the room and the corridor is 0.7 m long by 0.3 m wide, and the dimension of the window opposite to the door is 0.5 m (width) × 0.5 m (height). Several walls are made in steel with 25 mm as thickness. The outdoor wind has modeled as a vent with a fixed flow at the level of window. In the current numerical study, outside has been modeled in two openings: the first in front of the corridor and the second beside the window as shown in the figure 1. This configuration represents the one used during experiment studies of Li et al. [4]. Several simulations are carried out with a fire located in the middle of the room with 0.6 m × 0.7 m as area. The fire source has modeled as a burner with a mass flow rate using liquefied petroleum gas as fuel. Three calculations were carried out with wind speeds varying from 0 to 3.46 m/s for Heat Release Rate of 92 kW.

![Fig. 1. Geometry of computational domain.](image)

2.2. Numerical modelling approach
The simulations were carried out using the CFD code Fire Dynamics Simulator (FDS) 6. It solves the Navier-Stokes equations using an explicit finite difference scheme. Being a fire CFD code, it models the thermally driven flow with an emphasis on smoke and heat transport. It is a large eddy simulation (LES) model using a uniform mesh, and has parallel computing capability using Message-Passing Interface (MPI). In this work, the numerical modeling was made using the LES approach as turbulence model, the Eddy Dissipation Concept (EDC) as combustion model and thermal radiation model. The calculations are carried out from default models of turbulence and extinction but the combustion model used is based on the finite rate combustion. This combustion model uses Arrhenius parameters (A: pre-exponential factor and E: activation energy). The fuel used is the LPG and the mass flux given by the experimental data [4] has been used as input for all simulations.

2.3. Mesh resolution

The mesh resolution is carried out for four different mesh sizes: 20cm, 10cm, 5cm and 2.5cm. The choice of these mesh sizes is made by recommendations in the numerical studies[11][12]. Mesh resolution is made on temperature and smoke velocity at 70cm high. It is shown that for mesh size of 5cm, the prediction is similar with the experiment data (cf. Fig. 2). According to mesh size of 2.5 cm, the numerical results are better accurate. But in table 1, the time calculation of mesh size 2.5cm is 10 times longer than mesh size 5cm. In addition, the relative gap of the mesh size of 5cm is near to the mesh size of 2.5cm. So, reconciling precision and calculation time, the mesh size of 5cm will be used for the several calculations.

![Fig. 2. Grid cells influence on: (a) temperature at 70cm high; (b) smoke velocity without wind at 70cm high.](image)

**Table 1. Results of different mesh size of numerical grid**

| Numerical grid    | Number of cells | Temperature (%C) | Smoke velocity (m/s) | CPU time (h) |
|-------------------|-----------------|------------------|----------------------|--------------|
| Mesh size 20cm    | 1685            | 31.06            | 33.17                | 1.2          |
| Mesh size 10cm    | 11865           | 19.76            | 16.91                | 4.4          |
| Mesh size 5cm     | 94920           | 6.06             | 6.23                 | 9.6          |
| Mesh size 2.5cm   | 759360          | 3.80             | 3.85                 | 92.2         |

Relative Gap is obtained by:

$$RG = 100 \times \sqrt{\frac{1}{n} \sum_{i=1}^{n} (y_{pre,i} - y_{exp,i})^2}$$

(1)

Where $y_{pre}$ is a predicted value, $y_{exp}$ is an experimental value and $n$ is the number of experimental points.

Using the mesh size of 5cm, figure 3 presents the simulation of the smoke velocity field without wind velocity in the cross section $y=0.5$ m at 300 s. The cross section is the plan in the middle of the corridor. Taking this plan at the height of 70 cm, it is shown that the maximum value of the smoke velocity is near the door and
decreases with the distance as shown in the figure 2(b). In addition, taking account smoke stratification with a hot zone near the ceiling and a cold zone near the floor, it is observed that buoyancy effects give the reverse observation. Indeed, near the floor, the smoke velocity increases with the distance (distance from the doorway).

Fig. 3. Simulation of the smoke velocity field without wind velocity in the cross section y=0.5m at 300 s.

Considering tests with outdoor wind velocity of Vw=1.73 m/s and Vw=3.46 m/s, it is shown in figure 4 that prediction is similar with experiment. These comparisons highlight the ability of the solver to reproduce the smoke flow with the vortex induced by the recirculation zones.

Fig. 4. Smoke velocity with: (a) Vw=1.73 m/s; (b) Vw=3.46 m/s at 70 cm high.

3. RESULTS AND DISCUSSION

3.1. Numerical results

In this section, the influence of wind velocity on smoke temperature is presented from numerical results. The calculations are carried out with three different wind velocities (Vw=0 m/s, Vw=1.73 m/s and Vw=3.46 m/s). The choice of these three velocities is due to the fact that for velocities higher than Vw=3.46 m/s, the smoke has the same behavior.

From figure 4(a), it is observed that the smoke temperature increases suddenly with introducing of outdoor wind and converges to an average value little bigger than smoke temperature without wind. Moreover, it's presented that the more wind velocity increases, the more oxygen concentration increases. This contributes to the decrease of the smoke temperature near the ceiling.

Figure 4(b) presents the comparison between the smoke temperature near the celling and near the floor. It can be seen that the more wind speed increases, the more the smoke stratification is disturbed. When the wind velocity is 0 m/s, smoke temperature near the floor is about 5 °C and 60 °C near the ceiling. Regarding the wind velocity of 1.73 m/s, the smoke temperature near the floor is about 25 °C and 100 °C near the ceiling. From wind velocity of 3.46 m/s, the smoke temperature near the ceiling and floor are similar with an average of 76 °C. With these values, it is observed that the more wind velocity increases, the more smoke temperature near the floor increases. Then, it becomes similar with smoke temperature near the ceiling. In this context, it’s possible to happen that smoke occupies entire corridor from a value of wind velocity. Under these conditions, it exist a high risk of toxicity for people. These different observations highlight the ability of FDS to reproduce the effects of wind on the smoke movement in an enclosure.
3.2. Sensitivity analysis on the results

A local sensitivity analysis (SA) proposed by Chaos [9] is carried out in this study in order to identify the input parameters which has an influence on the output data such as smoke temperature. This methodology is based on the determination of the local sensitivity coefficient $S_j$ for different input parameters $\beta$. From the local sensitivity coefficient $S_j$, it is possible to calculate the area $S_{j, tot}$. For different parameters $\beta$, it is possible to calculate the area $A_j$ representing the ratio between the area $S_{j, tot}$ and the sum of the different areas $S_{j, tot}$ of the different parameters $\beta$. These parameters are calculated from equations 2, 3 and 4.

In this study, the input parameters $\beta_j$ are fuel mass flux (MF), the material properties (conductivity $\kappa$, emissivity $\varepsilon$, density $\rho$ and specific heat $C_p$), the Arrhenius parameters ($A$, $E_a$), the turbulence model parameter (Tur) and the extinction model parameter (Ext). $\Delta$ has been taken equal to 10% for calculation. The output data used to make the analysis sensitivity is the smoke temperature. From figure 5, it is shown that the values of $A_j$ for nine input parameters are between 4% and 28%. Considering the fact that, an input parameter has an effect on the output data when its value of $A_j$ is above of 10% (value of $\Delta$). So, the input parameters which have the more effect on the smoke temperature are fuel mass flux, activation energy and conductivity. From this observation, it’s important to define the value of MF, $\lambda$ and $E$ with a lot of precision in order to avoid big variation on the output data.

Fig. 5. Sensitivity analysis results on smoke temperature near the ceiling.
4. CONCLUSION

In this paper, a CFD approach is proposed to evaluate the effects of outdoor wind on the smoke spreading induced by an adjacent compartment fire. This approach is based on experiment data using FDS version 6. Simulations are performed for different wind velocities. It is found that the numerical results are similar to experimental data.

It is shown that the outside wind has noticeable impact on the smoke stratification. Indeed, with outdoor wind, the smoke temperature increases suddenly and decreases to converge to homogeneous value in the entire enclosure. The more wind velocity increases, the more the smoke stratification is disturbed. As a result, with a smoke present in the entire enclosure, a risk of toxic smoke can pose threat to people’s safety.

A local sensitivity analysis (SA) is carried out to highlight the influence of input parameters used in the modeling on output data such as the Heat Release Rate, temperature, velocity etc. It is found that mass flux of fuel and parameters related to combustion model can obviously affect output data such as smoke temperature near the ceiling.

From this work, it is proved that CFD FDS is able to provide information related to the smoke movement in a compartment and with the sensitivity analysis, it is shown that it’s important to define correctly the input parameter such as fuel mass flux and the parameters of the combustion model.

REFERENCES

[1] K.T. Paul, T.R. Hull, K. Lebek, A.A. Stec, Fire smoke toxicity: The effect of nitrogen oxides, Fire Saf. J. 43 (2008) 243–251. doi:10.1016/jfiresaf.2007.10.003.

[2] T.R. Hull, D. Brein, A.A. Stec, Quantification of toxic hazard from fires in buildings, J. Build. Eng. 8 (2016) 313–318. doi:10.1016/j.jobe.2016.02.014.

[3] L.H. Hu, R. Huo, Y.Z. Li, H.B. Wang, W.K. Chow, Full-scale burning tests on studying smoke temperature and velocity along a corridor, Tunn. Undergr. Sp. Technol. 20 (2005) 223–229. doi:10.1016/jtust.2004.08.007.

[4] S.C. Li, D.F. Huang, N. Meng, L.F. Chen, L.H. Hu, Smoke spread velocity along a corridor induced by an adjacent compartment fire with outdoor wind, Appl. Therm. Eng. 111 (2017) 420–430. doi:10.1016/applthermaleng.2016.09.086.

[5] M.H. Zhong, C.L. Shi, L. He, J.H. Shi, C. Liu, X.L. Tian, Smoke development in full-scale sloped long and large curved tunnel fires under natural ventilation, Appl. Therm. Eng. 108 (2016) 857–865. doi:10.1016/applthermaleng.2016.07.141.

[6] A. Li, Y. Zhang, J. Hu, R. Gao, Reduced-scale experimental study of the temperature field and smoke development of the bus bar corridor fire in the underground hydraulic machinery plant, Tunn. Undergr. Sp. Technol. 41 (2014) 95–103. doi:10.1016/jtust.2013.11.009.

[7] G. Zhao, T. Beji, B. Merci, Study of FDS simulations of buoyant fire-induced smoke movement in a high-rise building stairwell, Fire Saf. J. 91 (2017) 276–283. doi:10.1016/jfiresaf.2017.04.005.

[8] Q. Wang, F. Tang, L.J. Li, X.C. Zhang, C.G. Fan, Large Eddy Simulation on the Effect of Smoke Exhaust Openings Arrangement on the Smoke Spread in Tunnel Fires, Procedia Eng. 135 (2016) 308–314. doi:10.1016/proeng.2016.01.135.

[9] M. Chaos, Application of sensitivity analyses to condensed-phase pyrolysis modeling, Fire Saf. J. 61 (2013) 254–264. doi:10.1016/jfiresaf.2013.09.016.

[10] Y. Jiang, K. Chetehouna, S. Zhang, J. Qin, W. Bao, N. Gascoin, Sensitivity analysis of fluid properties and operating conditions on flow distribution in non-uniformly heated parallel pipes, Appl. Therm. Eng. 130 (2018) 458–465. doi:10.1016/applthermaleng.2017.11.005.

[11] B. Magnognou, J.P. Garo, B. Coudour, H.Y. Wang, Risk analysis of unburnt gas ignition in an exhaust system connected to a confined and mechanically ventilated enclosure fire, Fire Saf. J. 91 (2017) 291–302. doi:10.1016/jfiresaf.2017.03.036.

[12] I. Sellami, B. Manescau, K. Chetehouna, C. de Izarra, R. Nait-Said, F. Zidani, BLEVE fireball modeling using Fire Dynamics Simulator (FDS) in an Algerian gas industry, J. Loss Prev. Process Ind. 54 (2018) 69–84. doi:10.1016/jjlp.2018.02.010.