

## International Symposium on Safety Science and Engineering in China, 2012 (ISSSE-2012)

# Air Flow through the Door Opening Induced by a Room Fire under Different Ventilation Factors

N. Cai<sup>a</sup>, W.K. Chow<sup>a,\*</sup>

<sup>a</sup>Research Centre for Fire Engineering, Department of Building Services Engineering, The Hong Kong Polytechnic University, Hong Kong, China

### Abstract

Air flow through the door opening induced by a room fire was studied by Computational Fluid Dynamics (CFD) model Fire Dynamics Simulator (FDS) version 5 in this paper. There are two key problems in applying CFD. The first one is on design fire boundaries, and the second one the computing grid size. Consequently, many arguments were raised in fire hazard assessment for projects applying performance-based design in providing fire safe design. Prescribed input heating rate based on the experimental results was used in FDS. Three fire scenarios based on different ventilation conditions were simulated. Different boundary conditions were investigated to evaluate free open boundary conditions in applying CFD to predict heat release rate (HRR) and the air flow through the opening. The effects of open boundary conditions on HRR and pressure distribution at the opening plan under different ventilation factors were analyzed.

© 2012 Published by Elsevier Ltd. Selection and/or peer-review under responsibility of the Capital University of Economics and Business, China Academy of Safety Science and Technology. Open access under [CC BY-NC-ND license](#).

*Keywords:* CFD, Air flow rate, Ventilation factor, Free boundary

### 1. Introduction

It is important to understand the air flow rate through the openings during fires in fire safety engineering. The minimum flashover heat release rate (HRR) in a room fire can be estimated based on the air flow rate [1]. The ventilation factor plays an important role in estimating the air intake rate [2].

CFD has been widely applied in hazard assessment in applying performance-based design (PBD) for providing fire safety performance for projects with difficulties to comply with the codes. The CFD software FDS is commonly used to study fire driven fluid flow. The Navier-Stokes equations were derived to study low-speed, thermally-driven flow with an emphasis on smoke and heat transport from fires [3]. In this paper, FDS version 5 was applied to investigate the air flow rate through the door opening induced by an ISO9705 room fire. Results were used to evaluate FDS model in predicting HRR in boundary analysis.

### 2. CFD simulation setup

Experiment was conducted [4] with a gasoline pool fire in an ISO9705 room calorimeter at Lanxi, Harbin, Heilongjiang, China. A room calorimeter of length 3.6 m, width 2.4 m and height 2.4 m was constructed of brick with a cement finish as

---

E-mail address: [beelize@polyu.edu.hk](mailto:beelize@polyu.edu.hk)

in Fig. 1. The thickness of the brick wall was 0.25 m, with the cement finish of thickness 20 mm. The ceiling was constructed with 0.2 m reinforced concrete. There is a door of height 2.0 m and width 0.8 m. HRR was measured by an oxygen consumption calorimeter. This measured curve of HRR was used in FDS simulation.

A three-dimensional Cartesian coordinate system was used with length along the x-direction, width along the y-direction, and height along the z-direction. Free boundary condition was applied, which allowed fluid to enter and leave the computational domain freely. Pressure was taken as the same as the ambient pressure.

The total time for each simulation is taken to 1300 s. The time step is determined by the Courant-Friedrichs-Lewy (CFL) condition to satisfy the stability criteria [3, 5].

### 3. Grid size

The Large Eddy Simulation (LES) methodology views the transient signatures as comprising larger eddies that are resolvable by the calculation procedure and smaller unresolvable eddies which need to be modeled in a manner as comprising a time averaged component and a fluctuating perturbation about that average [6].

What determines the size of eddies that are resolvable and those are not in the LES model is the fineness of the numerical grid. The process of refining the grid system is referred to as a grid/mesh sensitivity study.

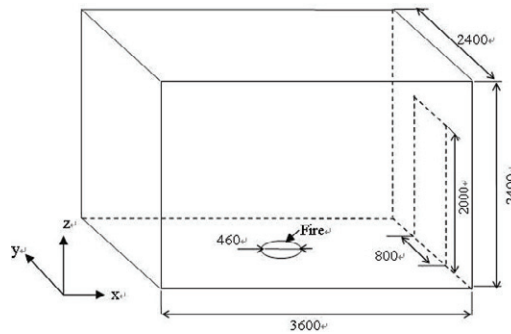


Fig. 1. Experiment room (in mm)

For simulations involving buoyant plumes, the non-dimensional expression  $D^*/\delta x$  can play a role as a measure of how well the flow field is resolved, where  $D^*$  is a characteristic fire diameter [3]:

$$D^* = \left( \frac{\dot{Q}}{\rho_\infty C_p T_\infty \sqrt{g}} \right)^{2/5} \quad (1)$$

and  $\delta x$  is the nominal size of a mesh cell. The quantity  $D^*/\delta x$  can be thought of as the number of computational cells spanning the characteristic diameter of the fire. A refined grid system can improve the accuracy of results of LES.

It is suggested by McGrattan [7] that the value of  $D^*/\delta x$  should be larger than 10 to guarantee the reliable operation of FDS. This value has been confirmed by Merci's study [8] to be acceptable for FDS simulation. Zou and Chow [9] obtained reasonable FDS predictions of temperature and radiation data using  $D^*/\delta x$  of  $\approx 14$ . Ma and Quintiere [10] found the optimum resolution of a pool fire simulation when the value of  $D^*/\delta x$  was around 20. The results of study by Hietaniemi [11] on pool fire showed that at least 20 cells should be within the diameter of the pool to get good agreement.

Stretched mesh (stretched in x- and y-directions and uniform in z-direction) system was applied in order to refine the grids in the vicinity of the fire and door where large key spatial gradients of fluid quantities were anticipated. Followed by the previous numerical study by the authors [12], grid convergence would be achieved under the grid size of 0.0375 m for the core area.

#### 4. Free open boundary

An open boundary on the exterior boundary of the computational domain can allow bi-directional flow with hot gas flowing out and cool air coming into the room through the openings. In FDS [5], the hydrodynamic pressure (head)  $H$  under the outflow condition at OPEN boundary is specified by the velocity vector, pressure perturbation and density :

$$p = \frac{1}{2} |\mathbf{u}|^2 + \frac{\tilde{p}}{\rho} \quad (2)$$

Where the pressure is set to ambient pressure by the user,  $\rho$  is the ambient density and  $\tilde{p}$  is the most recent value of the velocity on the boundary. Under the inflow condition at an OPEN vent, FDS makes the assumption that Bernoulli holds (i.e. inviscid, steady, incompressible) and that the fluid element on the boundary has accelerated. Note that the background density  $\rho$  is usually equal to the initial ambient density at the start of a calculation, but may change in time if the baroclinic correction term is included. There are many debates on specifying free open boundary conditions in applying CFD to building fire hazard assessment and should be watched carefully [13].

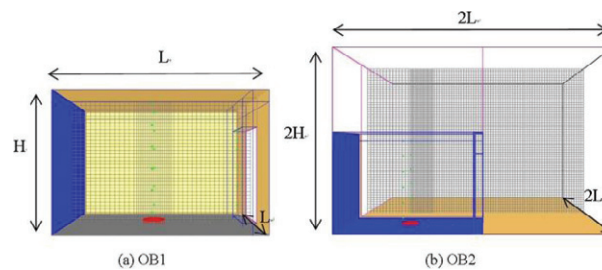


Fig.2. Plan of simulation domain

In the previous study mentioned [12], five different open boundaries were examined using the same ventilation arrangement. In this study, two free open boundary conditions labeled OB1 and OB2 with the computational domain extended to different distances beyond the room as shown in Fig. 2 were examined with three different ventilation arrangements. The impact of changing the boundary conditions on HRR and air flow will be examined case by case later.

#### 5. Ventilation factors

The ventilation factor  $V_f$  [2] is defined as:

$$V_f = A_o \sqrt{H_o} \quad (3)$$

Where  $A_o$  is the area of the openings and  $H_o$  is the opening height.

Different ventilation scenarios were investigated. Ventilation conditions were adjusted by the door height from sill to upper boundary of the door. As shown in Fig. 3, three different ventilation conditions with a fixed door of width 0.8 m but different height, labeled SC1 to SC3, used in this study are:

- SC1: Door height 2.0 m from the floor.
- SC2: Door height 1.0 m from the floor.
- SC3: Door height 0.5 m from the floor.

The door has a sill of 0.1 m height. The scenario SC1 was under the same condition of the experiment mentioned above [4]. Simulation results were compared with experimental data.

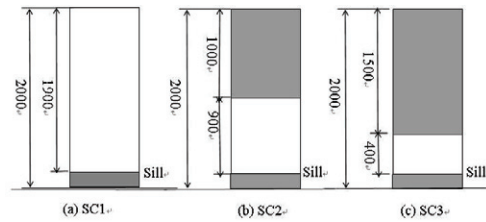


Fig. 3. Ventilation dimensions of the door (in mm)

## 6. Results

As mentioned earlier, there are still many debates on applying free open boundary. In this paper, two free open boundary conditions labeled OB1 and OB2 were examined. For condition OB1, open boundary was applied on the plan of the door. For condition OB2, the computational domain was extended. The length and height of the extended space were once of the room length and height as in Fig. 2(b). Pressure was taken as the same as the ambient pressure. Stretched mesh with cubic grids of size 0.075 m by 0.075 m by 0.075 m and finer grids of size 0.0375 m by 0.0375 m by 0.0375 m was used in these simulations as shown in Fig. 2.

With the fire load located in the center of the room, scenarios SC1 to SC3 were tested under boundary condition OB1 and OB2 respectively. Figs. 4 and 5 show the predicted results of HRR of SC1 to SC3. Except SC3, the predicted curves of HRR are very similar to that of the experimental results.

## 7. Comparisons with experiment

In order to quantify the comparison precisely, functional analysis recommended by Peacock et al. [14] on zone modeling was applied to evaluate the CFD results [15-17]. Transient predicted and measured data are expressed as vectors  $\vec{P}$  and  $\vec{M}$ . The Euclidean norm and secant inner product cosine between  $\vec{P}$  and  $\vec{M}$  are calculated:

$$\text{Norm} = \frac{|\vec{P} - \vec{M}|}{|\vec{P}|} \quad (4)$$

$$\text{Cosine} = \frac{\langle \vec{P} \times \vec{M} \rangle}{|\vec{P}| \times |\vec{M}|} \quad (5)$$

Values of norm and cosine are used to compare CFD predicted results with measured data. For a good agreement of the two curves on experiment and model, norm (a measure of the difference in the overall magnitude) is expected to approach 0; and cosine (a comparison of the shapes of the curves) is expected to approach 1.

Table 1 shows the functional analysis results of the point-to-point comparison for different boundary conditions. For SC1 which is under the same condition of the experiment, results of both boundary conditions OB1 (cosine of 0.967 and norm of 0.029) and OB2 (cosine of 0.969 and norm of 0.027) agree well with the experiment. Though the functional analysis confirms that boundary condition OB2 agrees better with experiment, extending the boundary from OB1 to OB2 gives little changes in both curve shape and magnitude. However, for SC2 and SC3, decreasing the ventilation factors would increase deviations between the predicted results of HRR and that of input. The most significant case is SC3 which has the lowest ventilation factor. Prediction in oxygen supply with the decreased opening area would decrease the heat release rate. Moreover, extending boundary to OB2 resulted in higher predicted value of HRR, as in Fig. 6.

Pressure distributions at the central line of the door in different scenarios are shown in Fig. 7. No significant differences are observed under SC1 and SC2 for both boundary conditions. However, the pressure profiles across the door under SC3 were entirely different. Though experimental pressure data were not available, the predicted pressure profile of OB1 under SC3 is hardly acceptable. It is because such pressure profile would induce almost single direction air flow through the opening.

For both boundary conditions, increasing the ventilation factors would increase the pressure difference at the opening plan, which suggests increase of the air flow rate through the opening.

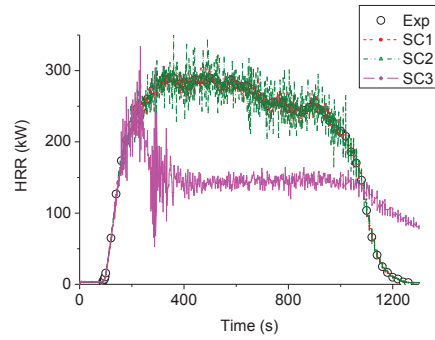


Fig.4. Comparison of the curves of HRR of SC1 to SC3 under OB1

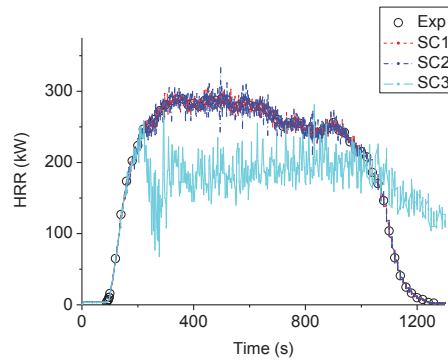


Fig.5. Comparison of the curves of HRR of SC1 to SC3 under OB2

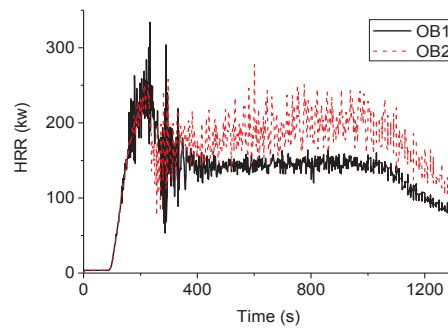


Fig.6. The curves of HRR of SC4

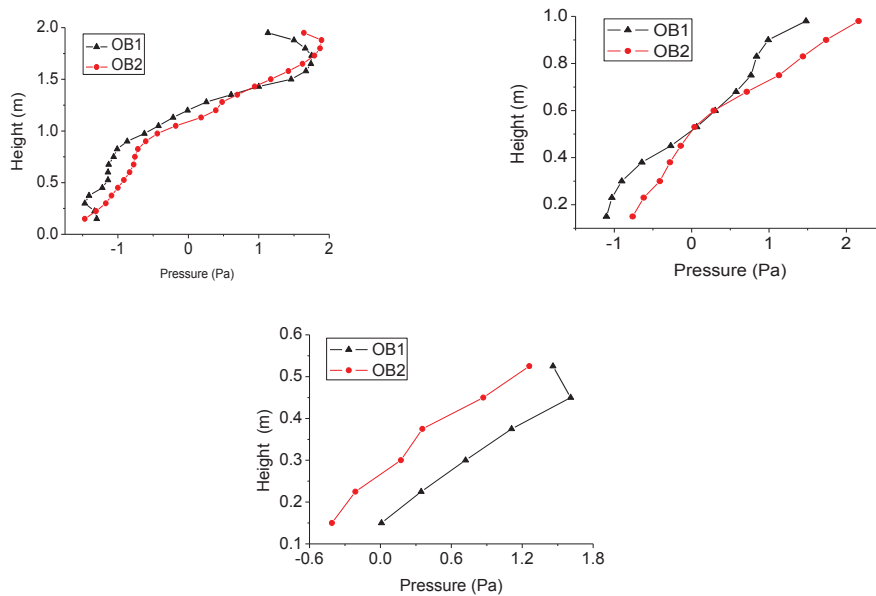


Fig.7. Pressure distribution at the central line of the door at 600s

Table1. Functional analysis results for different boundary conditions

Boundary conditions		Norm	Cosine
OB1	SC1	0.029	0.967
	SC2	0.074	0.805
	SC3	0.725	0.491
OB2	SC1	0.027	0.969
	SC2	0.028	0.951
	SC3	0.452	0.527

## 8. Conclusion

Air flow through the door opening induced by a room fire under different ventilation factors was studied by CFD. Pressure distribution at the opening plan suggests that the air flow through the opening would be more vigorous with increase of ventilation factors. The location of the opening will also have an effect on air flow. When the opening is closer to the top of the room, ventilation rate would increase due to buoyancy of the hot air.

Functional analysis was carried out to compare predicted results of HRR with the input (experimental) data under different open boundaries. The computational domain outside did not give much difference to the predicted HRR when ventilation factor is relatively large. The impact of boundary condition would be more significant on fire development with decreasing the ventilation factor. Especially when the ventilation factor is fairly small, extending the open boundary might avoid underestimating the HRR. For pressure distribution, extending the computational domain would give fairly reasonable results. Therefore, free opening boundary condition should be evaluated before being applied in CFD simulations, especially when the combustion process is included while simulating fires in rooms with small ventilation factors. Extending the computational domain to a sufficient distance beyond the opening is recommended. Further related work on validation and verification of proper description of free boundaries in FDS should be conducted.

## Acknowledgements

The work described in this paper was supported by a grant from the Research Grants Council of the Hong Kong Special Administrative Region, China for the project “Aspects of open kitchen fires in tall buildings and protection alternatives” with account number B-Q27R.

## References

- [1] V. Babrauskas, 1980. "Estimating room flashover potential," *Fire Technol* 16, pp. 94-103.
- [2] V. Babrauskas and R. B. Williamson, 1978. "Post-flashover compartment fires: Basis of a theoretical model," *Fire Mater* 2, pp. 39-53.
- [3] K. B. McGrattan, B. Klein, S. Hostikka and J. E. Floyd, 2009. "Fire Dynamics Simulator (Version 5), User's Guide," NIST Special Publication 1019-5, National Institute of Standards and Technology, Gaithersburg, MD, USA.
- [4] W.K. Chow, G.W. Zou, Y. Gao, N. Zhu, "Experiment on room fire with oxygen consumption calorimetry," *Int. J. Eng. perform. -based codes*, in press.
- [5] K. McGrattan, S. Hostikka, J. Floyd, H. Baum, R. Rehm, W. Mell et al, 2009. "Fire Dynamics Simulator (Version 5): Technical Reference Guide," NIST SP 1018, NIST Special Publication 1018-5, National Institute of Standards and Technology, Gaithersburg, Maryland, USA.
- [6] G. Cox, "Fire Modelling, 2004." Building Research Establishment Digests DG367, 2nd Edition Edition, pp.1-8.
- [7] K. B. McGrattan, H. R. Baum and R. G. Rehm, 1998. "Large eddy simulations of smoke movement," *Fire Saf. J* 30, p. 161-178.
- [8] B. Merci and K. Van Maele, 2008. "Numerical simulations of full-scale enclosure fires in a small compartment with natural roof ventilation," *Fire Saf. J* 43, p. 495-511.
- [9] G. W. Zou and W. K. Chow, 2005. "Evaluation of the field model, fire dynamics simulator, for a specific experimental scenario," *J. Fire Prot. Eng* 15, p. 77-92.
- [10] T. G. Ma and J. G. Quintiere, 2003. "Numerical simulation of axi-symmetric fire plumes: accuracy and limitations," *Fire Saf. J* 38, p. 467-492.
- [11] J. Hietaniemi, S. Hostikka and J. Vaari, 2004. "FDS simulation of fire spread comparison of model results with experimental data," *VTT Working p.4*.
- [12] N. Cai and W. K. Chow, 2012. "Numerical Studies on Heat Release Rate in Room Fire on Liquid Fuel under Different Ventilation Factors," *Int. J. Chem. Eng* 2012, Article ID 910869, p.13.
- [13] C. L. Chow and W. K. Chow, 2009. "A brief review on applying computational fluid dynamics in building fire hazard assessment," A Chapter in *Fire Safety*, I. Sogaard and H. Krogh, Eds., Nova Science Publishers.
- [14] R. D. Peacock, P. A. Reneke, W. D. Davis and W. W. Jones, 1999. "Quantifying fire model evaluation using functional analysis," *Fire Saf. J.* 33, p. 167-184.
- [15] W. K. Chow and G. W. Zou, 2009. "Numerical simulation of pressure changes in closed chamber fires," *Build. Environ* 44, p.1261-1275.
- [16] W. K. Chow, C. L. Chow and S. S. Li, "Simulating Smoke Filling in Big Halls by Computational Fluid Dynamics," *Model. Simul. Eng* 2011, p.16.
- [17] W. K. Mok and W. K. Chow, 2004. "Verification and Validation in Modelling Fire by Computational Fluid Dynamics," *Int. J. Archit. Sci* 5, p. 58-67..