International Journal on Engineering Performance-Based Fire Codes, Volume 5, Number 4, p.176-180, 2003

LARGE EDDY SIMULATION OF EXTERNAL FIRE SPREAD THROUGH OPENINGS

Lianyu Cao and Yincheng Guo
Department of Engineering Mechanics, Tsinghua University, Beijing 100084, China

ABSTRACT

This paper focuses on the application of large eddy simulation for the study of external fire spread through openings. The performance of a horizontal projection with different lengths and different positions has been investigated, including the consideration of wind effects. Based on the similarity principle, instantaneous behaviors of the opening jet plume have also been captured in this paper.

Keywords: opening jet plume, horizontal projection, LES, instantaneous flow behavior

1. INTRODUCTION

When a fire breaks out and spreads in a building, high-temperature jet plumes ejected from external windows can cause the spread of fire to the floors above. In order to minimize the risk of secondary fire at the level above the fire compartment, exterior architecture designs, vertical or horizontal projections are usually used. Research on this subject has been carried out for many years [1].

Ashton and Malhotra [2] had investigated storey to storey fire spread by conducting large-scale experiments on a four-storey building and concluded that a vertical separation of 3 ft (~900 mm) was inadequate to prevent the entry of external flame from a fire in a lower storey. Further research by National Research Council of Canada in 1974 concluded that "a vertical spandrel wall was found to be not a practical means of protection against flaming issuing from an opening". Luo et al. [3] had applied a CFD model, STAR-CD to justify the engineering design of the projections. They simulated a 7.5 MW compartment fire and presented a comparison of a 900 mm spandrel performance on stopping fire spread with a 500 mm horizontal projection. It was found that the horizontal projection would provide a better performance (hot smoke away from wall) for fire separation than the vertical spandrel (hot smoke close to wall).

Ohmiya and Yusa [4] had studied the opening jet plume behavior by carrying out compartment fire experiments with different opening sizes, various soffit lengths and various distances between the soffits and the upper end of the openings. The results showed that the nondimensional temperature distributions of jet plumes are geometrically similar, small-size (0.5 m × 0.5 m × 0.5 m) and medium-size (1.5 m × 1.5 m × 1.5 m) compartment models are nearly identical for each nondimensional soffit length. Their experiments also proved that a wider opening tends to cause a closer jet plume to the wall above the opening.

Galea et al. [5] had used computational fluid dynamics (CFD) techniques in the analysis of fire plumes emerging from windows. They examined three window configurations: narrow, wide and wide with a 1.0 m deep external protrusion (apron). The results of the simulations showed that the plume would be detached from the external wall for the narrow window, and attached for the wide window. With the adding of a 1.0 m long horizontal projection right above the wide window, the jet plume would be projected away from the facade.

More recent investigations have also considered the heat fluxes impinging on the facade above the window openings. For example, research at the National Research Council of Canada [6,7] showed a significant drop in heat transfer from a window fire plume to the building facade above it when a horizontal projection was deployed immediately above the window opening. Experiments reported by Oleszkiewicz [8] used a three-storey high burn facility with a window opening of 2.6 m wide by 1.37 m high and with compartment fire sizes of 5.75 MW and 6.9 MW. He showed that a 300 mm, a 600 mm or a 1000 mm horizontal projection installed above the window could reduce the exposure (at 1.0 m distance above the opening) by approximately 50%, 60%, 85% respectively.

Although much work has been done, the application of the research results to the building design is still limited. More detailed studies are required in consideration of actual designs because there are various combinations of opening conditions, projection conditions and ventilation conditions. The use of CFD has been supposed as an alternative means to predict the effect of a fire. The present study aims to apply FDS, based on

LES technology and developed by NIST, for the study of external fire spread through openings under through-draft ventilation condition.

2. METHODOLOGY

2.1 Hydrodynamic and Turbulence Model

An approximate form of the Navier-Stokes equations appropriate for low Mach number applications is used in FDS, together with a Smagorinsky turbulence sub-grid scale mode. Details can be found in other reference [9].

2.2 Combustion Model

The mixture-fraction combustion model [9] based on infinitely fast chemistry kinetics is adopted in FDS. Start with the most general form of the combustion reaction:

$$\upsilon_F Fuel + \upsilon_O O_2 \rightarrow \sum_i \upsilon_{P,i} \ Products$$

The mixture fraction Z is defined as:

$$Z = \frac{sY_F - (Y_O - Y_O^{\infty})}{sY_F^I + Y_O^{\infty}}$$

where

$$s = \frac{\upsilon_O M_O}{\upsilon_F M_F}$$

Note that Y_F^I is the fraction of fuel in the fuel stream and Z varies from 0 (pure air) to 1 (pure fuel). The mixture fraction satisfies the conservation law, with box and F-V filtering.

$$M_j = -\bar{\rho}(\overline{u_j Z} - \tilde{u}_j \tilde{Z}) = \frac{\mu_T}{S_{CT}} \cdot \frac{\partial \tilde{Z}}{\partial x_j}$$

where μ_T is turbulent viscosity which is simulated by Smagorinsky sub-grid scale model and S_{CT} is turbulent Schmidt number.

The heat release rate per unit volume is based on Huggett's assumption [10] of oxygen consumption:

$$\dot{q}''' = \Delta h_O \dot{m}'''$$

$$-\dot{m}''' = \nabla \cdot (\rho D \frac{dY_O}{dZ} \nabla Z) - \frac{dY_O}{dZ} \nabla \cdot \rho D \nabla Z$$

where Δh_O is the energy released per unit mass of oxygen consumed.

2.3 Radiation Heat Transfer Model

In most fire scenarios, soot is the most important combustion product controlling the thermal radiation. So it is possible to assume that the hot smoke behaves as a gray medium. The Radiative Transport Equation (RTE) for a non-scattering gray gas is:

$$s \cdot \nabla I(x,s) = \kappa(x)[I_b(x) - I(x,s)]$$

where $I(x,s)$ is the radiation intensity, **s** is the unit normal direction vector and the source term is due to the blackbody radiation $I_b = \sigma T^4 / \pi$. The wall boundaries are assumed to be diffuse and gray. Absorption coefficient $\kappa(x)$ is calculated using RADCAL narrow-band model. The RTE is solved using the Finite Volume Method, a technique similar to those for convective transport for fluid flow. More details about the thermal radiation model can be found in FDS- technical reference [9].

3. PREDICTED RESULTS AND DISCUSSIONS

The sketch of the fire compartment is shown in Fig. 1. It is a two-story like building with the dimension of 1.5 m (length) × 1.5 m (width) × 2.4 m (height). The fire source (1.2 MW) is located in the center of the first story floor. During the simulation period, the window (0.8 m wide and 0.5 m high) and the door (0.4 m wide and 0.8 m high) are both kept open. The upper story only has the right-side wall along which the fire plume emerged from the bottom opening might spread upwards.

In this paper, the horizontal projection is placed either just at the upper end of the opening (Hs = 0 m) or 0.3 m away from it vertically. The length of the projection (Ls) is either 0.2 m or 0.3 m.

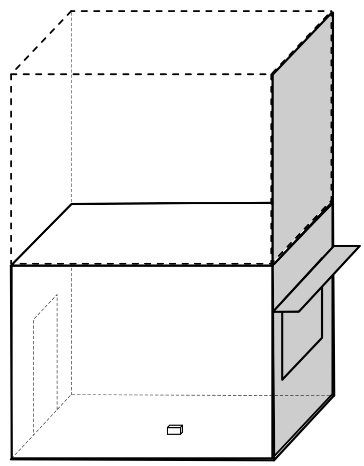

Fig. 1: Sketch of fire compartment

3.1 Performance of a Horizontal Projection

Fig. 2 shows the simulated temperature contours of the opening jet plume with and without a horizontal projection. Obviously, the 0.2 m long projection keeps the jet plume much farther away from the wall. Thus, about 40% reduced temperature distribution is obtained above the window along the vertical wall centerline, as is shown in Fig. 3. If the projection length comes to be 0.3 m, extremely low temperature distribution will be obtained. When the projection is placed 0.3 m away from the upper end of the opening, the 0.2 m long horizontal projection seems to perform much better.

Heat flux distributions along the vertical wall centerline above the projection are shown in Fig. 4. It can be seen that rather lower heat flux is achieved with the setting of a horizontal projection. However, a different projection height (Hs = 0.3 m) seems just to "put away" the heat flux distribution to higher positions.

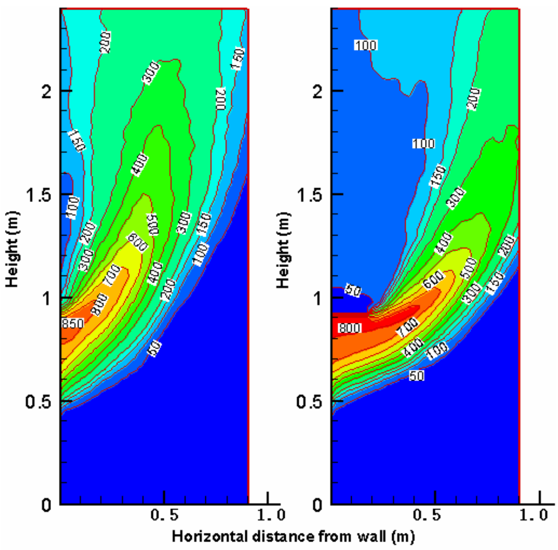

Fig. 2: Temperature distribution contours

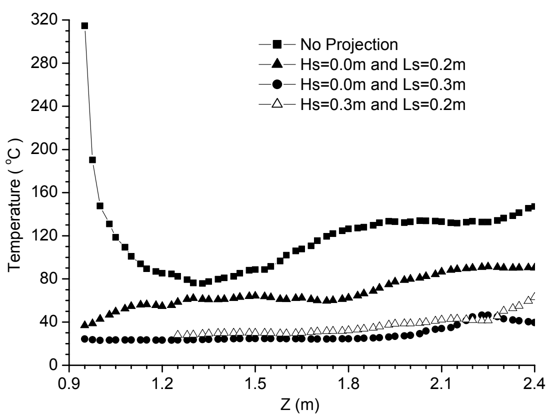

Fig. 3: Temperature distributions

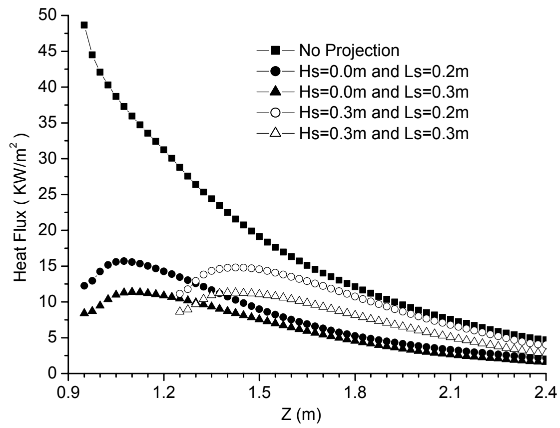

Fig. 4: Heat flux distributions

3.2 Wind Effects

Wind can greatly affects the opening jet plume behavior, as is shown in Fig. 5, in which the velocity of the wind is either -3 ms^{-1} (left) or 3 ms^{-1} (right) and the projection lengths are both 0.2 m. Obviously, when the plume is against a moderate velocity wind, it will be pushed towards the wall slightly. Therefore, temperature and heat flux distribution along the wall will be a little higher compared with the no wind situation. However, if the wind is too strong, little smoke can eject from the window. When the plume ejects down the wind, it will be driven much farther away from the wall. Thus, rather low temperature and heat flux will be achieved on the wall. While at the same time, temperature of the jet plume gets an increase of about 100°C, which reveals that more fuels have been driven out of the window and burned outside.

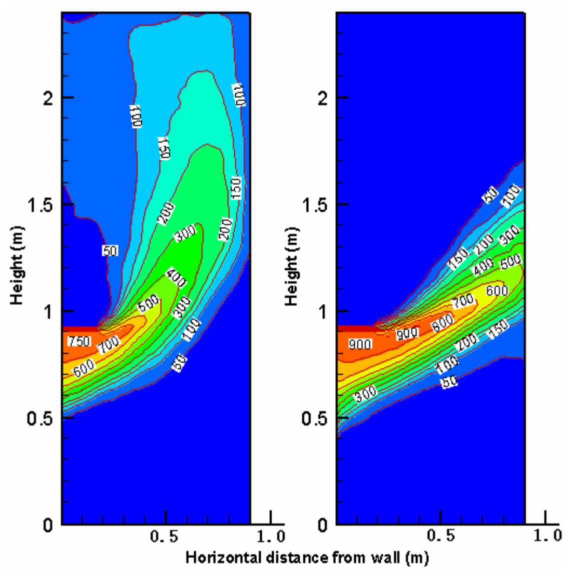

Fig. 5: Temperature distributions with wind effects

3.3 Instantaneous Behaviors of Opening Jet Plume

The above discussions focus on the time-averaged opening jet plume behaviors with and without a horizontal projection. In order to capture the instantaneous eddy structures, the computational domain should be reduced, confined to the present computer capacity. Based on the results of Ohmiya and Yusa [4], a 1/2 scale reduced geometrically similar room of the above computational domain has been used here. The heat release rate turns to be 212 KW now so as to ensure similarity principle.

Fig. 6 shows the captured vortex structures of the opening jet plume using very fine grids ($\Delta = 0.25$ cm) in the vertical section. From Fig. 6(a), vortexes at the edge of the forepart jet plume can be seen clearly. The big vortex at the top of the plume will form some slight vortexes around the bulk of the jet quickly, as is shown in Fig. 6(b) and Fig. 6(c). Fig. 6(c) also shows an interesting phenomenon: two bulks of jet plume circle with each other and form a structure a little like a double helix. This can be explained as the effects of buoyancy or else a symmetrical jet plume will be formed instead.

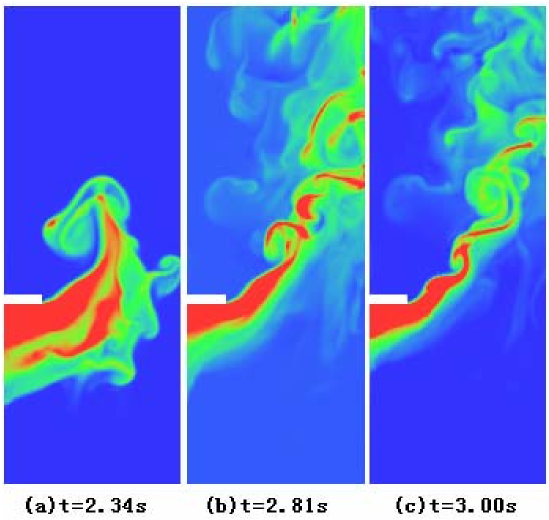

Fig. 6: Instantaneous temperature distributions

4. CONCLUSIONS

Large eddy simulation has been applied to simulate opening jet plume above. The results show that extremely low temperature and heat flux distributions can be achieved with the setting of a 0.3 m long horizontal projection at the upper end of the opening. If the projection height (Hs) turns to be 0.3 m, a 0.2 m horizontal projection is enough to obtain the same temperature distribution as low as the 0.3 m one. However, this does not help to reduce the heat flux above the window. Wind also shows great effects on the opening jet plume behavior. The plume can be driven either farther or closer to the wall according to the direction of the wind. Based on the similarity principle, very fine grids have been used to capture the instantaneous vortex structures of the opening jet plume. The results show that some slight vortexes will appear around the bulk of the jet during the beginning period. An interesting double helix structure which forms from the effects of buoyancy has also been captured here.

ACKNOWLEDGEMENTS

Authors acknowledge the financial support provided by Ministry of Science & Technology (MOST) of China under project of National Key Basic Research Special Funds with Grant no. 2001CB409600, and Beijing Municipal Commission for Science & Technology, under Grant no. H023220050120 and no. H023620020120.

REFERENCES

1. C.A. Wade and J.C. Clampett, Fire performance of exterior claddings, FCRC PR 00-03.
2. L.A. Ashton and H.L. Malhotra, External walls of building – Part I The protection of openings against spread of fire from storey to storey, FR Note No. 436 (1960).
3. M.C. Luo, L.D. Zhao, V. Cheng and R. Yau, “Performance-based fire engineering design: Application of CFD model for the prevention of external fire spread”, Preliminary Proceedings, 5th AOSFST, Newcastle, Australia (2001).
4. Y. Ohmiya and S. Yusa, “Prediction method of opening jet plume behavior in the presence of an opening soffit”, Preliminary Proceedings, 5th AOSFST, Newcastle, Australia (2001).
5. E.R. Galea, D. Berhane and N.A. Hoffmann, “CFD analysis of fire plumes emerging from windows with external protrusions in high-rise buildings”, Interflam '96, 7th Proceedings, 26-28 March 1996, Cambridge, England, Interscience Communications Ltd., London, England, pp. 835-839 (1996).
6. Oleszkiewicz, “Heat transfer from a window plume to a building façade”, Proceedings of the Winter Annual Meeting of the American Society of Mechanical Engineers, San Francisco, HDT- Vol. 123, pp. 163-170 (1989).
7. Oleszkiewicz, “Fire exposure to exterior walls and flame spread on combustible cladding”, Fire Technology, November, pp. 357-375 (1990).
8. Oleszkiewicz, “Vertical separation of windows using spandrel walls and horizontal projections”, Fire Technology, November (1991).

9. K.B. McGrattan, H.R.Baum, R.G. Rehm, A. Hamins and G.P. Forney, Fire Dynamics Simulator - Technical reference guide, NIST 6467, January (2000).

10. C. Huggett, "Estimation of the rate of heat release by means of oxygen consumption measurements", Fire and Materials, Vol. 4, pp. 61-65 (1980).