DISCUSSION ON BUOYANCY, TURBULENCE AND COMBUSTION IN FIRE FIELD MODELLING

Y. Gao
Department of Building Engineering, Harbin Engineering University, 15001 Harbin, P.R. China

W.K. Chow
Areas of Strength: Fire Safety Engineering, Research Centre for Fire Engineering
Department of Building Services Engineering, The Hong Kong Polytechnic University, Hong Kong, China

(Received 1 September 2005; Accepted 19 October 2005)

ABSTRACT

Buoyancy, turbulence and combustion in simulating a compartment fire with a field model (or application of Computational Fluid Dynamics) are discussed. The ISO room-corner fire test is taken as an example. Flaming region of the fire is taken as a combustion gas volume region with reacting and not yet reacting zones. Those zones might be in laminar flow or in turbulent flow, depending on their positions and mixing ratios of fuel and air. A model based on eddy dissipation concept which can simulate premixed, partially premixed and non-premixed combustion with buoyancy was used together with k-ε type of turbulence model with buoyancy.

1. INTRODUCTION

Combustion in a fire had been studied extensively in the literature [e.g. 1,2]. At the initial stage of an enclosure fire, a fire plume will be formed above the burning object with most of the heat released from chemical combustion reactions in the flaming region. Buoyancy is the driving force of fluid flow in the plume because of the high temperature and density differences. It is well known that both gas velocity and temperature would decrease radially outward from the central axis of the plume, but increase longitudinally upward and then decrease after reaching the maximum values for very tall spaces. Inside the combustion region, the flame temperature was measured to be about 1000 K, and the upward gas velocity varied with the heat release rate.

Building fire field model, or application of Computational Fluid Dynamics (CFD) [e.g. 1-5] is commonly used in simulating compartment fires. The combustion process can be modeled [e.g. 3,6] together with the eddy-break-up (EBU) model, eddy dissipation concept (EDC) model, probability density function (PDF) model and the flame surface density (FSD) model. Based on the flamelet hypothesis, the turbulent combustion field can be described in terms of an array of “laminar elements” or flamelets which constitute collectively to form a turbulent flame. These elements are transported, strained and wrinkled by the turbulent flow.

However, the buoyancy-induced turbulence is a complicated phenomenon still not too well understood. For example, correction method due to Rodi was commonly used. But buoyancy would decrease the turbulence kinetic energy k, leading to higher dissipation rate. As a result, k would be further reduced, giving smaller conductive heat transfer. Further, the large number of intermediate chemical reactions [e.g. 7,8] that would occur in a combustion process makes the turbulence phenomena even more difficult to understand. To simplify the calculation, simple chemical reacting system [1-3,9,10] was often used to model the combustion process in a fire. This is only good for extracting some quantities useful for engineering applications, such as assessing the hazard scenario in buildings with glass systems.

2. FLAMES IN A FIRE

The EBU model was proposed long time ago for turbulent effect in combustion with k-ε model. There, the reaction rate (or breaking frequency) \( \dot{W}_{S,T} \) is assumed to be proportional to the average value of the concentration fluctuation \( g \) and the frequency (or turbulent diffusion time \( \tau_{T} \)):

\[
\dot{W}_{S,T} = C_{EBU} \rho g \frac{\varepsilon}{k} \quad (1)
\]

where \( C_{EBU} \) is an empirical constant and \( g \) is given by the instantaneous concentration fluctuation \( Y' \):

\[
g = \overline{Y'^{2}} \quad (2)
\]

\( \overline{Y'^{2}} \) is the mean square of the concentration fluctuations. The production of turbulence kinetic energy \( P_{k} \) and its dissipation rate \( c_{k} \) are:

\[
P_{k} = \frac{1}{2} \rho \overline{V^{2}} (\overline{V^{2}} + \overline{V'^{2}}) \quad (3)
\]

and

\[
c_{k} = \frac{k}{2} \rho \overline{V'^{2}} \quad (4)
\]

where \( \overline{V^{2}} \) is the mean square of the velocity fluctuations, \( \overline{V'^{2}} \) is the turbulent kinetic energy density.

\[ c_{k} = c_{k_{1}} + c_{k_{2}} \quad (5)
\]

The production term \( P_{k} \) is made up of the two terms \( 2c_{k_{1}} \rho g \frac{\varepsilon}{k} \) and \( c_{k_{2}} \rho g \frac{\varepsilon}{k} \). The first term is larger, the second one is smaller. Consequently, the larger term is the dominant one.

The above equations are linearized. The mean terms are considered to be constant, the major contribution to the eddy dissipation rate are the turbulent kinetic energy density terms of the two terms in equation (5).

The large number of intermediate chemical reactions is large. To simplify the calculation, a simple chemical reacting system [1-3,9,10] was often used to model the combustion process in a fire. This is only good for extracting some quantities useful for engineering applications, such as assessing the hazard scenario in buildings with glass systems.
Correction due to Magunssen would hold when [11]:

\[ g \propto \sqrt{Y_1} \quad \text{or} \quad g \propto Y_2 \quad \text{or} \quad g \propto Y_3 \]

where \( Y_1 \), \( Y_2 \), and \( Y_3 \) are the concentration of fuel, oxidant or combustion products. This gives:

\[ g \propto \min\{\sqrt{Y_1}, Y_2, Y_3\} \quad (3) \]

Note that \( g \) can be deduced from the transport equation.

A deficiency in EBU is that it cannot describe chemical kinetics. Time averaged values can be expressed by the Arrhenius rate:

\[ \bar{W}_{S,A} = B\rho \frac{E}{R T} \exp(-\frac{E}{RT}) \quad (4) \]

That means the turbulent reaction rates \( \bar{W}_S \) and \( \bar{W}_{S,A} \) for \( s \) groups are related to \( g \).

Since fluid motion in a fire plume is induced by buoyancy, characteristics reaction time \( \tau_e \) is very close to the turbulent diffusion time \( \tau_T \):

\[ \frac{\tau_e}{\tau_T} \approx 1 \quad (5) \]

Therefore, both chemical reaction and turbulent mixing are important. Note that \( \tau_T \) is given by:

\[ \tau_T = \frac{\ell^2}{D_T} = \frac{\ell}{k^{1/2}} = \frac{k}{\varepsilon} \quad (6) \]

For the corrected EDU due to Magnussen [11], i.e. the “Eddy-dissipation concept”, the above conditions would be satisfied.

At different positions of the turbulent reacting flow:

\[ \bar{W}_S \approx \begin{cases} \bar{W}_{S,T} \text{ for } \tau_T < \tau_C \text{ or } \bar{W}_{S,T} < \bar{W}_{S,A} \\ \bar{W}_{S,A} \text{ for } \tau_T < \tau_C \text{ or } \bar{W}_{S,A} < \bar{W}_{S,T} \end{cases} \quad (7) \]

i.e.

\[ \bar{W}_S = \min\{\bar{W}_{S,T}, \bar{W}_{S,A}\} \quad (8) \]

This model is also known as EDC which can simulate premixed, partially premixed, and non-premixed combustion with good accuracy. Therefore, it is commonly used in the literature.

The main advantage of EBU model is that it takes the turbulent diffusion time as a key factor of reaction rate. On the other hand, this gives problems in simulating the flaming region of a fire.

For the combustion reaction of a buoyant turbulent fire plume, air entrained from the surrounding would mix with the fuel and provide the necessary oxygen. If the surrounding region cannot provide sufficient amount of air, reacting gas at high temperature would move up due to buoyancy, stretching to higher positions for getting air until the fuel is burnt out. At the same time, the premixed but not yet combusted fuel would lead to incomplete combustion once meeting air at the outside ambient.

Because of that, the following model is used in this paper:

The flaming region is considered as a bigger combustion region with ‘reacting’ and ‘not yet reacting’ gas zones. Those zones might be in laminar flow or in turbulent flow, depending on the positions in the flow field and their flow velocities. There are fuel and air of different ratio in each zone, some of them give premixed flames and some give diffusional flames. There are no clear identifications of burnt fuel nor unburnt fuel interfaces. All these can be demonstrated by referring to the numerical experiments simulated in this paper. It is difficult to list the differential equation for the sum of flame area per unit volume \( \Sigma (m^2/m^3) \), and the flame surface per unit mass \( S_f (m^2/kg) \) as in a flamelet model or an FSD model.

The core area of a fire plume might be burning without sufficient oxygen. Whether the fuel can react or not depends on the amount and the rate of air flowing out. This would determine the flame height as shown later. For unreacted fuel staying at positions with high temperature, more air would be entrained to mix with the fuel for burning. Some fuel might not burn because the temperature is not high enough at these positions as shown in later figures.

For buoyancy due to large temperature differences, and burning characteristics of fuel under different conditions of insufficient oxygen at different times and positions, air can be supplied and mixed at different rates. These factors interact with each other and give plumes of different shapes. In this paper, the flame is visualized as average, steady and turbulent. By using EDC model with EBU properties, the fire plume can be simulated by the CFD models such as PHOENICS [10]. The EDC model is a key part of those CFD models for studying chemical reaction. It is recommended that
under burning, a fire plume can be modelled with physics similar to a flamelet model.

Buoyant turbulent combustion field is suggested to be described in terms of an array of “laminar or turbulent elements”, which collectively constitute a turbulent flame (premixed partially, premixed and non-premixed combustion). For bigger combusting volumes, combustion in formal zones with laminar turbulent elements satisfies the criteria of EDC model.

3. NUMERICAL EXPERIMENTS

Combustion process in a compartment fire, especially the plume flow, had been studied quite extensively in the literature [e.g. 12]. In this paper, fire in a compartment of size following the ISO9705 [13-16] was considered. The length of the room is 3.6 m, width 2.4 m and height 2.4 m. There is a glass window of size 2.0 m by 2.0 m on one side of the wall, and a door of size 0.8 m by 2.0 m (high) in the front as shown in Fig. 1.

Two sets of simulations with different locations of fire were considered:

- Case A: The fire is located at the centre of the room.
- Case B: The fire is at the corner of the room.

The fire-induced gas flow and temperature distribution for the two cases would be very different.

The main objective of this paper is to study the combustion model under buoyancy-induced turbulence. Results are useful for fire hazard assessments, such as pressure acting on the glass panel. Therefore, to simplify the physical picture, methane was used as the fuel, instead of ethanol or methanol. The simple chemical reacting system was used by taking the ambient oxygen content as 0.232. Only the heat release rate of the combusting gas was considered, and nothing on the fuel type was studied. Case A simulations with the fire put at the centre of the room would give a better understanding on the flame itself. For Case B simulations with the fire placed at a corner, combustion and turbulence would be distributed by the wall-effect. In this paper, more attention would be paid to Case A simulations.

Simulations were performed with both the window and the door opened. The room was divided into 25, 23 and 32 parts along the x-, y- and z-directions respectively. The grid sizes were smaller in regions close to the fire.

Steady burning of methane (CH₄) was considered. The fuel was injected with a mass flow rate of 0.015 kgs⁻¹, with heat released from the combustion process. In the simulations, the total mass out-flowing rate of hot gases above the neutral plane would be balanced with the total mass in-flowing rate of cool air. Initial values for $k$ and $\varepsilon$ were $1 \times 10^{-3}$ Jkg⁻¹ and $1 \times 10^{-2}$ Jkg⁻¹s⁻¹ respectively. The relaxation factor for solving the $k$ equation was $1 \times 10^{-4}$, and for the $\varepsilon$ equation was $1 \times 10^{-5}$. The relaxation factor for pressure was taken to be 1.

![Fig. 1: Room geometry](image)
The burning of hydrocarbon in air proceeds by way of hundreds of distinguishable chemical reactions, involving tens of intermediate chemical species. Only the initial and final states are represented by:

\[
\text{Fuel} + \text{oxygen} \rightarrow \text{products}
\]

\[
1 \text{kg} \quad \text{s kg} \quad (1+s) \text{kg}
\]

where \( s \) is the stochiometric ratio. In this paper, the mixture fraction \( f_{\text{stoich}} \) is calculated from:

\[
f_{\text{stoich}} = \frac{1}{1+s}
\]

4. CASE A SIMULATIONS

Typical simulated results at the vertical planes at \( x = 1.2 \text{ m} \) and \( z = 1.3 \text{ m} \) are shown in Figs. 2 and 3 for a fire burning over a horizontal surface of size 0.5 m by 0.5 m, and height 0.2 m. For the temperature contours across the vertical plane at \( x = 1.2 \text{ m} \) as shown in Fig. 2, the flame was deflected to the back due to the motion caused by the air drawn from the door. This is a good demonstration that the fire plume is induced by buoyancy.

Temperature, pressure and velocity distributions near the fire are investigated first. For the temperature profiles at \( z = 1.3 \text{ m} \) shown in Fig. 4, the maximum temperature was 1250 K at 0.9 m high. Since there was insufficient oxygen in the flaming region, reaction occurred at lower temperature between 600 K and 800 K. The temperature curves took the form of a saddle with peak values at height below 0.9 m. Temperature decreased to 920 K as the height increased, say at \( y = 1.26 \text{ m} \) for curve 5. The depth of the saddle minimum gives the amount of oxygen received by the flame at different heights. Curve 3 is the temperature distribution curve at 0.74 m. At this height, more outside air was entrained than at 0.42 m (with temperature distribution shown in curve 2). The burning temperature at the central area was 300 K higher. For regions closer to outside, the temperature was about 1100 K with little variations because of better air supply, and thus giving a diffusional flame.

From the predicted fuel mixing ratio distributions, in regions with more injection of fuel, the fuel mixture ratio was about 0.1. For methane, fuel to air ratio was 1.0 to 9.0. For air with oxygen content of 0.232, \( f_{\text{stoich}} \) (guaranteeing complete combustion) is about 17.4. Therefore, complete combustion would be impossible due to the large amount of air required. Although fuel was added from a height of 0.3 m to 0.55 m, a large amount of it could only burn completely at higher positions. Therefore, higher combustion temperature was found at 1.0 m.

High temperature gas with fuel and mixed air moved up because of buoyancy and induced air motion. The mixing mechanism of fuel and air, heat release rate and maximum temperature are limited by buoyancy. For the above simulations, the flame expanded sideways due to high temperature with a maximum speed of 2.0 ms\(^{-1}\) as shown in Fig. 5. In Fig. 6, the flame moved upward at a speed less than 4.0 ms\(^{-1}\).

The pressure profile along the z-direction at \( x = 1.2 \text{ m} \) and \( y = 0.74 \text{ m} \) is shown in Fig. 7. The one along \( y = 1.2 \text{ m} \) and \( z = 1.3 \text{ m} \) is shown in Fig. 8. These diagrams can give some indication on how pressure varied in regions close to the fire in the compartment. The pressure difference induced by buoyancy along the vertical direction was 11 Pa. In regions closer to the bottom of the fire, the vertical pressure difference was 5 Pa. This pressure was due to the momentum of the fluid flow with certain amount of fuel mixed with air at 0.3 m to 0.5 m. Within this small region, the vertical pressure gradient is proportional to the flow rate. The pressure reduced from 2.8 Pa to the ambient value of -2.0 Pa, and the velocity increased from 0 to 3.4 ms\(^{-1}\). The burning gas entered the region at 298 K (point \( A_o \) in Fig. 9), to 700 K (point \( A \) in Fig. 9) at 0.32 m. The temperature kept constant (point \( B \) in the figure), and then increased to 1200 K at 0.95 m (point \( C \) in the figure). The corresponding points \( A_o, A, B \) and \( C \) are also shown in the velocity profile in Fig. 6 and the pressure profile in Fig. 8. Both temperature and velocity kept on decreasing after passing through point \( C \). One point to note is that from point \( A \) to \( B \), velocity rose rapidly from 0.6 ms\(^{-1}\) to 3.4 ms\(^{-1}\), but temperature was kept at the same value. From point \( B \) to \( C \), temperature rose rapidly from 620 K to 1250 K, but the velocity did not change. Obviously, the rapid increase in velocity from point \( A \) to \( B \) was because of the decrease in pressure. The change in potential energy due to a pressure difference of 5 Pa would give a change in kinetic energy for the velocity of 3.5 ms\(^{-1}\). The increase in pressure above point \( B \) (\( y = 0.55 \text{ m} \) in Fig. 8) to the room ceiling was due to buoyancy. The pressure at point \( B \) is about 2.7 Pa, which is the pressure at the flaming region at 0.5 m. Because of buoyancy, the pressure is 2.7 Pa lower than the ambient.

As seen from the velocity vectors diagrams in Figs. 2 and 3, radial air motion was induced by the high-temperature flaming region of the plume. Outside air moved into the room at -1.2 ms\(^{-1}\) to -1.6 ms\(^{-1}\) at the lower part of the door. The profile for the velocity component \( w \) along \( z \) at \( x = 1.2 \text{ m} \) and \( y = 0.74 \text{ m} \) is shown in Fig. 5. The value of \( w \) at point
F is -1.2 ms\(^{-1}\), corresponding to -2.0 Pa at the same point F in Fig. 7. For point E in Fig. 5, w is 0 ms\(^{-1}\), for pressure started to rise with velocity flowing in the opposite direction. Going back to point O’ in Fig. 5, w is 0 ms\(^{-1}\). The burning gas would be heated up and expanded, with w increased along the positive z-direction to 1.4 ms\(^{-1}\) at the surface of x = 1.2 m; and to -1.9 ms\(^{-1}\) along the negative z-direction. Those are shown as points D and C in Fig. 5, at 4.0 Pa and 3.0 Pa respectively. Averaging the pressure over these two points would give a mean pressure of 3.5 Pa. This explains why the flame expands to the nearby regions. The mean pressure surrounding the flame is -2.0 Pa, the pressure at the back of the flame (i.e. point B in Fig. 7) is lower than the pressure in the front (point E in Fig. 7). Back pressure would be induced by the gas curling round the flame, but the value is relatively low because of the large amount of air mixed in the flame. The increase in pressure in regions close to the flame from -2.0 Pa to 0 Pa at point E was due to the kinetic energy loss of the ambient air flowing in at a speed of -1.2 ms\(^{-1}\) reduced to 0 ms\(^{-1}\). Expansion of the burning gas would increase the pressure from 0 Pa to 4.0 Pa.

Predicted pressure contours along the window (x = 0 m) indicated that the pressure above the neutral plane was positive and higher than the ambient, so hot gas moved out from the upper part of the compartment. Pressure below the neutral plane was negative, cold air would move in from the window. Velocity vectors in a horizontal plane across the window at z = 1.3 m shown in Fig. 3a indicated that air moved in from outside to the room under the neutral plane, but hot gas moved out of the room above the neutral plane.

![Fig. 2: Results across the plane at x = 1.2 m](image)

![Fig. 3: Results across the plane at z = 1.3 m](image)
Fig. 4: Temperature variations along z at x = 1.2 m

Fig. 5: Velocity component w variations with z at x = 1.2 m

Fig. 6: Velocity component v variations with y at x = 1.2 m
Fig. 7: Pressure variations along z at x = 1.2 m

Fig. 8: Pressure variations along y at x = 1.2 m

Fig. 9: Temperature variations along y at x = 1.2 m
For studying the combustion process, the combustion product concentration contours across a plane at \( x = 1.2 \) m are shown in Fig. 10a, and smoke distribution in Fig. 10b. The oxygen concentration contours across a plane at \( z = 1.3 \) m are shown in Fig. 11a, and fuel injecting contours in Fig. 11b. Smoke concentration was computed following the procedures of calculating the following as listed in the user manual of PHOENICS [10]:

\[
\text{Constant} \times (f - f_{\text{stoich}}) \times T^5
\]

It is observed that at the central part of the flaming region, the fuel mixture ratio is relatively higher. Under the conditions that the oxygen concentration is insufficient and combustion is incomplete, the smoke concentration is higher.

The radiative heat flux contours across the wall surfaces, i.e. vertical planes at \( x = 0 \) m and \( z = 0 \) m, are shown in Figs. 12a and b. The six-flux-radiation model in PHOENICS was used. Following the physical characteristics of water vapor and carbon dioxide, absorption coefficient of the gas was taken as 0.2, scattering coefficient being 0.01, and emissivity of the wall was 0.9. As the room is small, radiation intensity is quite big and can be up to \( 3.5 \times 10^4 \) Jm\(^{-2}\).

5. CASE B SIMULATIONS

When the fire was placed at the corner, i.e. case B simulations in this paper, the air flow and the temperature were all different from simulations A where the fire was placed at the centre. Temperature would change the combustion reaction rates, and hence the heat release rates and the dissipation rates of turbulence kinetic energy. Using the EBU model would give a very difficult problem to solve.
Fig. 12: Radiation heat fluxes across the planes at $x = 0.0$ m and $z = 0.0$ m

(a) Across x-plane at 0.0 m in kWm$^{-2}$

(b) Across z-plane at 0.0 m in kWm$^{-2}$

Fig. 13: Velocity vectors for simulation B

(a) Across the plane $y = 0.35$ m

(b) Across the planes at $x = 0.1$ m,
    $y = 0.1$ m, $z = 0.1$ m

(c) Across the plane $z = 0.1$ m

(d) Across the plane $y = 2.4$ m
From the velocity vectors across a horizontal plane at 0.4 m high as shown in Fig. 13a, air was driven in from the lower parts of the window and the door. Upward velocity can be up to 2.33 ms\(^{-1}\) in the flaming region. Velocity vectors in the three planes at \(x = 0\) m, \(y = 0\) m and \(z = 0\) m are shown in Fig. 13b. The velocity vectors in the horizontal plane close to the wall with a window, i.e. \(x = 2.3\) m, are shown in Fig. 13c. The maximum velocity is 3.13 ms\(^{-1}\). Gas at the ceiling would move towards the door as shown in the vector diagram across a ceiling plane at \(y = 2.4\) m in Fig. 13d. The maximum velocity is 3.25 ms\(^{-1}\).

Distribution diagrams for the combustion product and mixture fraction at a plane across \(x = 2.3\) m are shown in Figs. 14a and b respectively. The difference with simulations A is that the combustion products moved upward along the wall surface to the ceiling. A certain amount of fuel was not burnt completely. An explanation is that the combustion region was so close to the wall that air would flow back quickly to the ceiling because of the space constraint. The combustion process was not complete due to oxygen starvation. Direct contact of the hot combustion product with the ceiling would heat the ceiling up. For simulations A with the fire at the centre, the temperature close to the ceiling was up to 600 K as illustrated in Fig. 9. But for the fire placed at the corner as in simulations B, temperature close to the ceiling was above 900 K as shown in Fig. 14c. Further, pressure at the corner of the ceiling would be higher than in other regions as shown in Fig. 14d.

Experimental studies on wood panels in a room of the same size as that in the ISO 9705 were reported in the literature [14,15]. There, much thicker char was found at the ceiling and on the wall close to the square burner. From the combustion viewpoint, the flame at the corner had a smaller contact area with outside air, thus leading to incomplete combustion. A large quantity of unburnt fuel was found at the ceiling. Therefore, the heat release rate was greater and the temperature was higher. Note that the combustion process of the wood panel itself would also generate heat and thus increase the temperature.

![Graphs showing various distributions](image)

**Fig. 14: Results for simulation B**
Velocity components $w$, differential pressure and temperature profiles at the central doorway ($x = 1.2$ m and $z = 3.6$ m) are shown in Figs. 15, 16 and 17 respectively. The neutral plane was found at the height of 1.05 m. The maximum outward velocity $w$ above the neutral plane was 1.2 m$^{-1}$, maximum differential pressure was about 2.0 Pa, and temperature was 420 K. The maximum inward velocity was -1.2 m$^{-1}$, differential pressure was -1.0 Pa, and temperature was close to the ambient value of 25°C. It can be seen from Fig. 16 that from 2.18 m to 2.4 m high, the air velocity reduced by 0.5 m$^{-1}$ and differential pressure was 5.5 Pa. This is consistent with the pressure induced by buoyancy.
6. CONCLUSION

The following conclusions can be drawn:

- In the flaming region of a buoyancy-induced fire plume, there are different combustion zones. Each zone would satisfy the conditions for applying EDC model. The turbulent combusting flow field was simulated in depth by PHOENICS [10].

- There are no clear interfaces of burnt fuel nor unburnt fuel as seen in Figs. 2, 3, 10 and 11. The flame height can be determined from whether the fuel is burnt or not in Fig. 2. Also, in Fig. 11, some fuel could not be burnt because the temperature was not high enough.

- More detailed thermal studies on a room with similar size as that quoted in the ISO 9705 test with the fire placed at the centre and at the corner were carried out. The velocity, temperature and pressure profiles in the flaming region were studied.

- From the simulations, high temperature gases emitted from combustion would be expanded. High speed of 2.0 ms\(^{-1}\) at the flame boundary would be reduced to 0 ms\(^{-1}\) rapidly. When the flame was kept at 1200 K, upward velocity induced by buoyancy would be from 0.6 ms\(^{-1}\) to 3.4 ms\(^{-1}\).

- For fire simulations [6] in a room of similar size as that in the ISO 9705 test [13], the neutral plane was kept at a height of 1.05 m. The air pressure was higher than the ambient in regions above the neutral plane. The differential pressure would increase to somewhere between 7.0 Pa and 8.0 Pa. Hot gas would flow out of the room at a velocity of 1.2 ms\(^{-1}\) to 1.8 ms\(^{-1}\), and the maximum air temperature was 147°C. The differential pressure near the door was 1.0 Pa. The pressure below the neutral plane would be less than the ambient pressure. Air would move in at a maximum velocity of -1.2 ms\(^{-1}\), and the temperature was the same as the ambient.

- For the fire placed at the corner of a room (Case B simulations), the upper ceiling temperature would be higher than placing the fire at the centre (Case A simulations). The space constraints on the combustion were discussed in this paper. The burning gas moved up with higher speed for simulations B, but the combustion process was incomplete at the lower part. Therefore, most of the combustion reactions took place at the ceiling, so giving higher heat release rates. This agreed qualitatively when compared with some experiments in the literature [14-16].

The above discussions are useful for applying CFD in fire hazard assessments, such as the behaviour of glass system.
ACKNOWLEDGEMENT

Appointment of Professor Gao was partly funded by the RGC project “Fire Safety for Glass Façades in Green and Sustainable Buildings” with number B-Q803.

REFERENCES