A STUDY OF COMPARTMENT FIRE UNDER FORCED VENTILATION: EXPERIMENTAL AND COMPUTATIONAL FLUID DYNAMICS MODELING

M.Z. ABU BAKAR[1] AND Y. WU[2]

[1]*School of Chemical Engineering, Universiti Sains Malaysia,*

Engineering Campus, 14300 Seberang Perai, Penang, MALAYSIA

[2]*University of Sheffield, Department of Chemical Engineering, United Kingdom*

E-mail: chmohdz@eng.usm.my

Abstract: This paper discusses on the dynamic behavior of the flame and smoke inside a compartment fire. The compartment can be referred to a room, road tunnel, duct, compartment or a building. A series of small scales experiments were carried on four rectangular ducts that have the same height (250 mm) but different widths (125 mm, 250 mm, 500 mm and 1000 mm). Fire simulations on the same compartments were performed to investigate the effectiveness of Computational Fluid Dynamics (CFD) in predicting fire phenomenon. CFD was found capable to predict fire phenomenon such as flame shape, flame height and flame tilt angle similar to the experiment.

Keywords: *Smoke flow, fire plume, compartment fire, buoyancy*

1. INTRODUCTION

Fire is an extremely complex phenomenon with various hazards to human, properties and environment. It embraces nearly all the effect found in subsonic chemically reacting flow. Fluid dynamics, combustion, kinetic, radiation and in many cases multi-phase flow effects are linked together to provide the extreme complex physical and chemical phenomenon. It is this complexity that delayed the development of fire research as a science until approximately the 1950s. Due to this complexity, attempts have been made in the mathematical modeling to study, understand and foremost to visualize the basic phenomenon of fire from first principles via solution of the basic conservation equations.

In a typical compartment fire, once the flames have reached the ceiling, they can no longer travel upwards and must therefore travel horizontally. Since they are very hot and therefore light gases, they travel under the ceiling and this situation can elongate as much as 5 – 10 times of an open fire [1]. This elongation arises because the mixing of the air into the flame under the ceiling is by a much slower process than the flame travels vertically. Therefore, in order to entrain enough air to burn all the volatile fuel, the horizontal flame has to be much longer. This phenomenon is shown in Fig. 1a to 1c.

The behavior of a fire in a compartment has been studied by a number of researchers [2-5]. Apart from the dynamic behavior of the fire, the study on the movement of the combustion products such as the smoke has equal importance. This is because smoke is one of the major factors that cause injuries and deaths.

The main objective of this work is to study the dynamic behavior of compartment fire. Parameters such as flame height and flame tilt angles for various fire sizes and ventilation velocities will be investigated by using both experimental and CFD modeling.

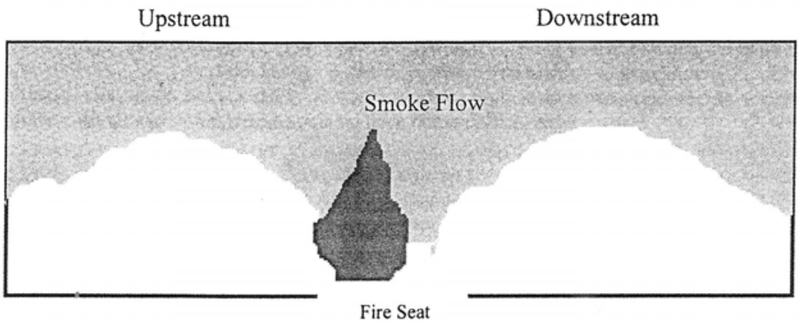

Fig. 1a: Illustration of a compartment fire without ventilation.

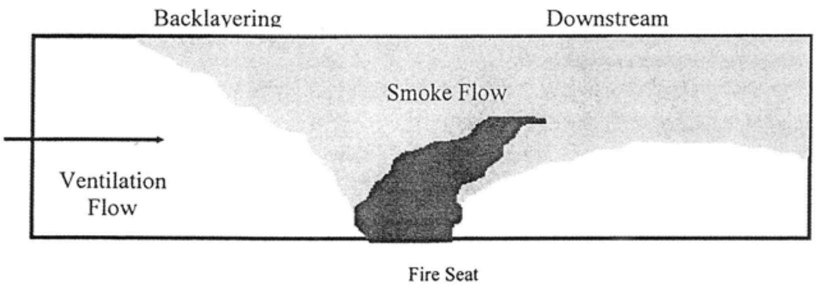

Fig. 1b: Illustration of a compartment fire with low ventilation.

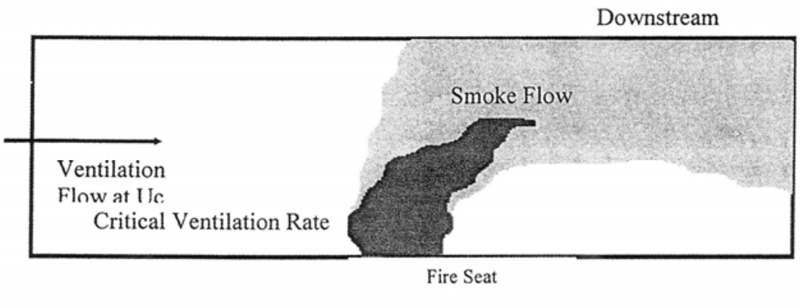

Fig. 1c: Illustration of a compartment fire with critical ventilation.

2. EXPERIMENTAL WORK

The schematic front view and the cross-section of the compartment are shown in Fig. 2. The majority of the model, including the base, was constructed from Perspex (PMMA) of thickness 6.25 mm. To prevent heat damage, the section closet to and downstream of the fire was made from 18 SWG (1.25 mm thick) stainless steel. Additionally, the section closet to the fire was cooled throughout the tests by a water spray. The entire compartment, of approximately 15 m length, was mounted on a series of brackets, which in turn were attached to wall-mounted frames. The height of each compartment was 250 mm and the width varying from 125 mm to 1000 mm.

Propane gas was used as a fuel, metered through a rotameter. This was introduced via a 106 mm diameter porous bed burner with its top surface set flush with the compartment floor. The propane flow rate was varied between 1 and 20 litres per minute, producing fires of 1.4 to 28 kW. These fire sizes correspond to fires of approximately 2.5 to 50 MW in a compartment of diameter around 5 m when the scaling procedure was applied. All products of combustion were exhausted to atmosphere at the downstream end of the compartment.

The ventilation air supply to the compartment was channelled in through a 101.6 mm steel pipe fitted with an orifice plate of throat diameter 71.8 mm, constructed in accordance with BS 1042. The flow was driven by a compressed air. The detailed structure of the air supply system can be found in reference [6]. The orifice plate provides a useful method of determining the total volumetric flow of air. K-type thermocouples were used to measure the temperatures inside the compartment.

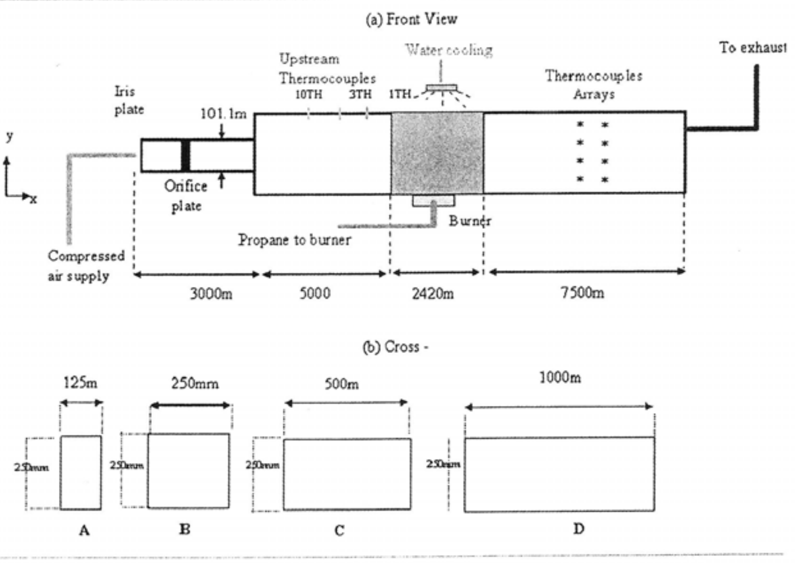

Fig. 2: Schematic front view and cross-section of experimental rig.

3. COMPUTATIONAL FLUID MODELLING

Three-dimensional simulations of smoke flow in the compartments have been carried using Computational Fluid Dynamics software, known as FLUENT. The combustion was considered to be a stoichiometric reaction of propane and air defined as

$$C_3H_8 + 5O_2 \Rightarrow 3CO_2 + 4H_2O \tag{1}$$

Two combustion models namely, the Magnussen and Hjertager [7] eddy-dissipation model and the mixture fraction/PDF approach have been used to model the combustion process. It was shown that the PDF approach used less CPU time and provides a better prediction of the flame shapes. Therefore most of the CFD simulations were carried out using the PDF approach.

The standard *k*-ε turbulence model was used to model the flow inside the compartments due to its simplicity and effectiveness. The standard *k*-ε turbulence model [8] includes basic modifications for buoyancy based on Ljuboja and Rodi [9] in the *k*-ε equation. The standard *k*-ε model is a two-equation eddy viscosity turbulence model which transport equations for two variables: *k* the turbulence energy, and ε the rate of viscous dissipation of turbulence energy. The turbulent effective viscosity, μ_t , is related to *k* and ε, by a velocity scale ($k^{1/2}$) and a length scale ($k^{3/2}/\varepsilon$) by the expression:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \tag{2}$$

and the velocity and length scales are predicted at each point in the flow via the solution of transport equations for k and ε:

$$\frac{\partial}{\partial x_i}(\rho u_i k) = \frac{\partial}{\partial x_i}\frac{\mu}{\sigma_k}\frac{\partial k}{\partial x_i} + G_k + G_b - \rho\varepsilon \tag{3}$$

$$\frac{\partial}{\partial x_i}(\rho u_i \varepsilon) = \frac{\partial}{\partial x_i}\frac{\mu}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x_i} + C_1\frac{\varepsilon}{k}(G_k + (1 - C_{3\varepsilon})G_b) - C_2\rho\frac{\varepsilon^2}{k} \tag{4}$$

Turbulence is generated according to G_k, where:

$$G_k = \mu_t\left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j}\right)\frac{\partial u_j}{\partial x_i} \tag{5}$$

and G_b is the generation due to buoyancy:

$$G_b = -g_i\frac{\mu_t}{\rho\sigma_h}\frac{\partial \rho}{\partial x_i} \tag{6}$$

where σ_h is the turbulent Prandtl number, $\frac{\mu_t C_p}{k_t}$.

and $C_{1\varepsilon}$, $C_{2\varepsilon}$, C_{μ}, σ_k and σ_ε are empirical constants, with values, 1.44, 1.92, 0.09, 1.0 and 1.3 respectively. $C_{3\varepsilon}$ is used to take account of the buoyancy effects. Woodburn and Britter [10,11] used $C_{3\varepsilon}$ equal to 0.20 and showed that the inclusion of the modified buoyancy gave better predictions between the measured and predicted results. The present study tested the influence of the value of $C_{3\varepsilon}$ in the prediction of backlayering and found that $C_{3\varepsilon}$ = 0.25 was optimal.

The total simulated compartment length was 8.1 m with the exclusion of a downstream section of 3.0 m length. The longitudinal computational domain was divided into three segments. Segment 1 was the upstream section of 5.0 m length. Segment 2 was the burner section of 0.1 m length and segment 3 was the downstream section of 3.0 m length. The first plane of the longitudinal domain was set to be the inlet of the ventilation flow and the last plane was set as the output of the smoke flow to the exhaust. The wall of the compartment was set to be a solid containing 1 cell. A non-uniform grid distribution was made to avoid a very large number of computational cells, while maintaining a sufficient degree of accuracy in the solution. The longitudinal and vertical grids were set at 102 and 28 cells, respectively. The half cross-sectional cells varying from 8, 14, 28 and 38, dependent on compartment widths. The total cells are, 22 848, 39 984, 79 968 and 10 8528 for the Compartments A, B, C and D, respectively. Grid sensitivity tests have been carried out and the results showed that the grid distributions for the compartments were sufficient.

Figures 3 and 4 show the longitudinal grid distribution for the compartments and the cross-sectional grids for each compartment.

4. RESULTS AND DISCUSSION

(A) EXPERIMENTAL

4.1 Behavior of the Flame

McCaffrey [12] proposed fire plume theory by studying a fire plume above a 30 cm square porous burner. A free fire plume consists of three distinct regimes. The three regimes are:

- Near fire, above the burner, where there is persistent flame and accelerating flow of burning gas
- A region in which there is intermittent flaming and a near-constant flow velocity (intermittent zone)
- The buoyant plume, which is characterised by decreasing velocity and temperature with height.

Figure 5 shows a photograph of a fire in one of the compartment at 15.0 kW. There are two distinguished regions that can be observed; the flame and the buoyant smoke flow. With the presence of longitudinal ventilation flow, the fire plume deflects at certain angle from the vertical. The fire plume still consists of three distinguished regions, similar finding from McCaffrey [12]. The present work considered that the temperature in the persistence flame regime is greater than

500 °C. The temperature in the intermittent regime is between 250 °C to 500 °C. The buoyant plume is considered to have temperature less than 250 °C.

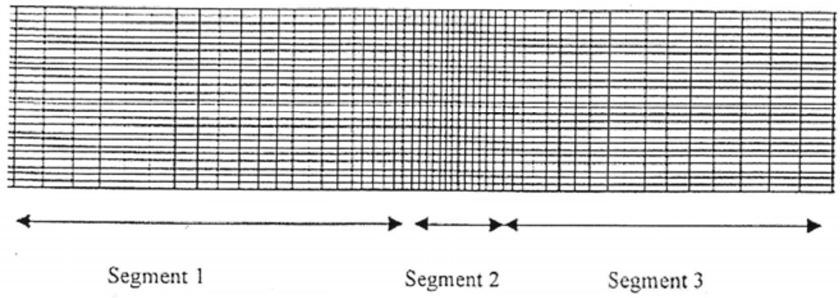

Fig. 3: Longitudinal grid distribution.

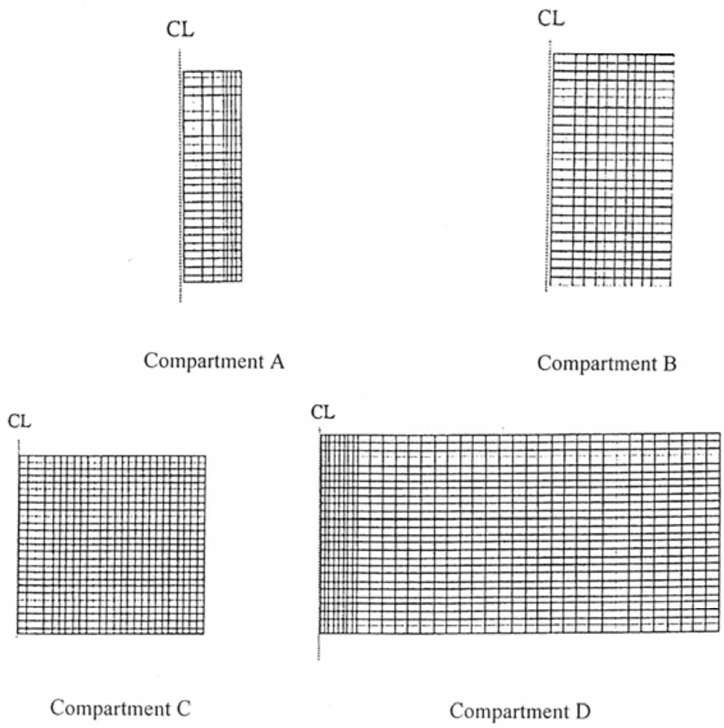

Fig. 4: Cross-sectional grid distribution.

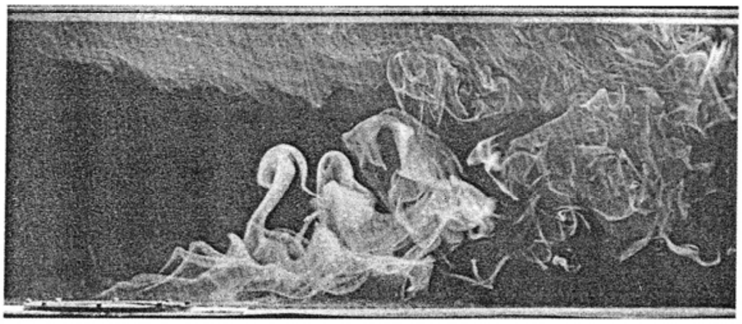

Fig. 5: Photograph of a fire in along compartment.

4.2 Flame Height

Unlike in an open fire, it was observed that the flame heights inside the compartment depend on the interaction between the fire plume with the compartment geometry and the ventilation flow. An attempt was made to illustrate the flame height with two main parameters; the fire sizes and ventilation velocity. Since the exact regimes associated with the fire plume were difficult to determine, both temperature values of 250 °C and 350 °C were selected to justify the boundaries for the intermittent regimes, while the boundary for the persistent regime remains at 500 °C.

Figure 6 depicts the measured temperature contour in compartment B at three fire sizes (3.0kW, 7.5 kW and 15.0kW). It can be observed that as fire size increases, the fire inside the compartment also grows in size. The maximum temperature recorded at 3.0 kW, 7.5 kW and 15.0 kW were in the order of 586 °C, 679 °C and 746 °C, respectively. The results show that for a small fire (3.0 kW), both intermittent regimes indicated by 250 °C and 350 °C is approximately at 125 mm above the compartment floor. Whilst, the persistent regimes in the three compartments lay low. However at 7.50 kW fire, the intermittent regimes indicated by the 250 °C contours have definitely reached the ceiling, while the intermittent regimes indicated by 350 °C have nearly reached the ceiling for all compartments. The persistent flames in the three compartments are approximately at 125 mm above the compartment floor. Finally at 15.0 kW fire, both intermittent regimes have already reached the ceiling, while the persistent flame have not yet reached the ceiling.

4.3 Flame Angle (α)

The measured flame tilt angles from the vertical for three fire sizes at least at two ventilation velocities are shown in Table 1. The tilt angle for 3.0 kW fire at the critical ventilation velocity (0.48 m/s) was 70°. When the fire size was increased to 7.50 kW, the tilt angle at the critical ventilation velocity (0.56 m/s) further increased to 73°. Finally at 15.0 kW, the tilt angle further increased to 76°. A similar pattern occurred in both compartments **A** and **D**. In general, it was found that the deflection angles of the fire plumes in all compartments were greater that 45 °.

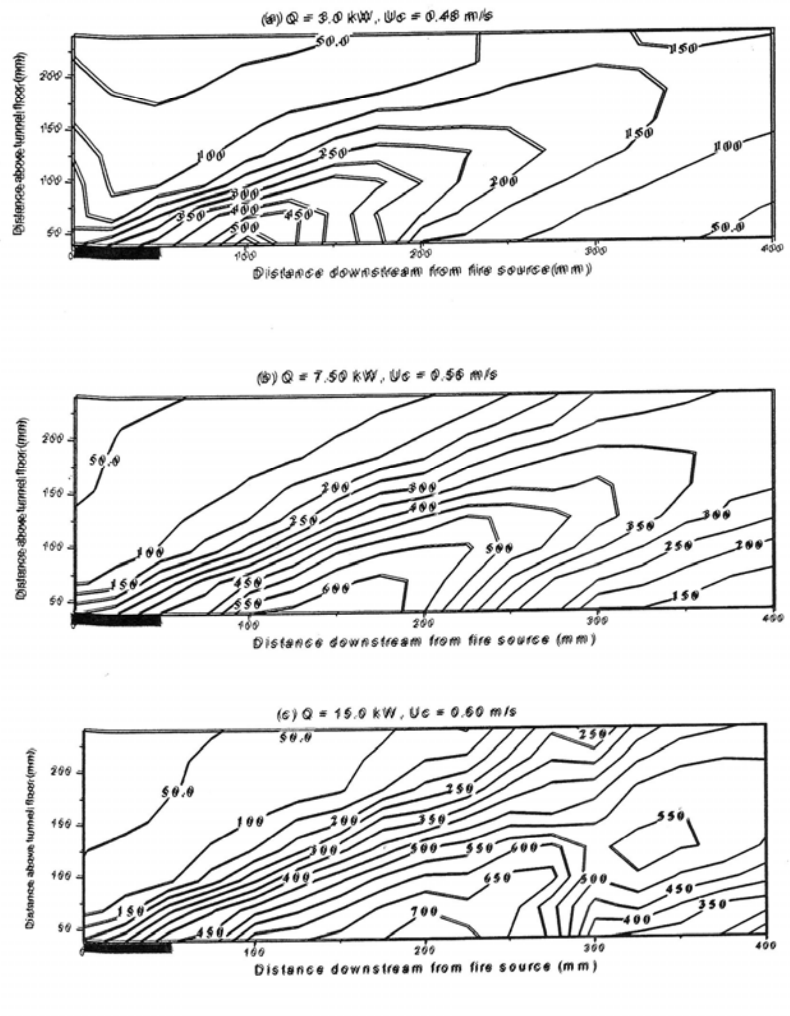

Fig. 6: Measured temperature contours at fire region for 3.0 kW, 7.50 kW, 7.5 kW and 15.0 kW fires.

Table 1: Measured plume tilt angle at various ventilation velocities and fire sizes.

Compartment B			Compartment D			Compartment A		
Q (kW)	V (m/s)	α (°)	Q (kW)	V (m/s)	α (°)	Q (kW)	V (m/s)	α (°)
3.0	0.35	60	3.0	0.18	47	3.0	0.25	65
	0.48	70		0.40	60		0.46	75
	0.96	78		0.50	64			
7.50	0.31	68	7.50	0.28	57	7.50	0.25	70
	0.56	73		0.54	65		0.48	79
15.0	0.34	73	15.0	0.35	60	-	-	-
	0.60	76		0.65	70	-	-	-

(B) COMPUTATIONAL FLUID MODELLING

4.4 Behavior of the Flame

Figure 7a to 7c show the simulated temperature distribution for compartment B at the same fire sizes 3.0 kW, 7.5 kW and 15.0 kW, respectively. With the interaction between compartment geometry and the ventilation flow, the fire plume rises above the compartment floor and deflects at certain angle from the vertical. The sizes and shapes of the fire plume can be clearly seen.

Figure 7 also indicates that CFD simulations predicted a slightly higher temperature in the flame area. The maximum temperature reached approximately 2200 K, which was quite high for propane and air burner combustion. This is caused by the combustion models employed by the FLUENT.

The turbulent combustion models are based on fast chemistry concepts, which overestimates the reaction rates. Therefore, the temperature is over predicted in the flame area. This problem is well known by the CFD simulation community. Both the scientists and CFD commercial users have pointed out the needs of better turbulence models. However, this is not the scope of the present study. Therefore, this problem will not be discussed further.

Another limitation of the combustion models is that the combustion is directly determined by the present of the fuel, therefore the CFD simulation can only predict continuous flame. The intermittent flames existing in the real fire plumes could not be predicted by the CFD simulations.

4.5 Flame Height and Flame Width

Based on the new temperature boundary for each regime as being defined in the previous section, the flame height in each compartment was measured. The results showed that for a small fire (3.0 kW), the intermittent flame was approximately one third of the height of the compartment. For medium size fire (7.50 kW), the intermittent flame was approximately three quarter of the height of the compartment. Finally at higher fire size (15.0 kW), the intermittent flame already reached the compartment ceiling.

The CFD results also showed that the persistent flame never reached the ceiling at critical condition even at higher fire size as shown in Fig. 7 for 15.0 kW fire. The persistent flame (>1500 K) only elongated further downstream. The variations of the flame heights predicted by CFD were almost similar to the experimental results, previously discussed in Section (A).

4.6 Flame Angle (α)

The measured tilt angles of the plume are shown in Table 2. Similar to the experimental results, CFD predicted the increase of fire plume tilt angles with the fire sizes. In addition, CFD also predicted that the fire plume tilt angle decreases with the increase of compartment aspect ratio.

It can be observed in Table 2 that the plume in compartment **A** has the highest deflection angle from the vertical. In contrast, the plume in compartment **D** has the smallest deflection angle from the vertical. The deflection angles vary from 75° to 56° in compartment **A** to compartment **D** at 3.0 kW. The deflection angles at 15.0 kW fire vary from 87° to 60°. The overall comparison shows that at specific compartment and fire size, the predicted plume tilts angle was slightly lesser than the experimental results.

Table 2: Predicted fire plume tilt angles.

Fire Size	Compartment A		Compartment B		Compartment D	
(kW)	Measured	Predicted	Measured	Predicted	Measured	Predicted
3.0	75°	75°	70°	65°	60°	56°
15.0	79°	87°	73°	65°	70°	60°

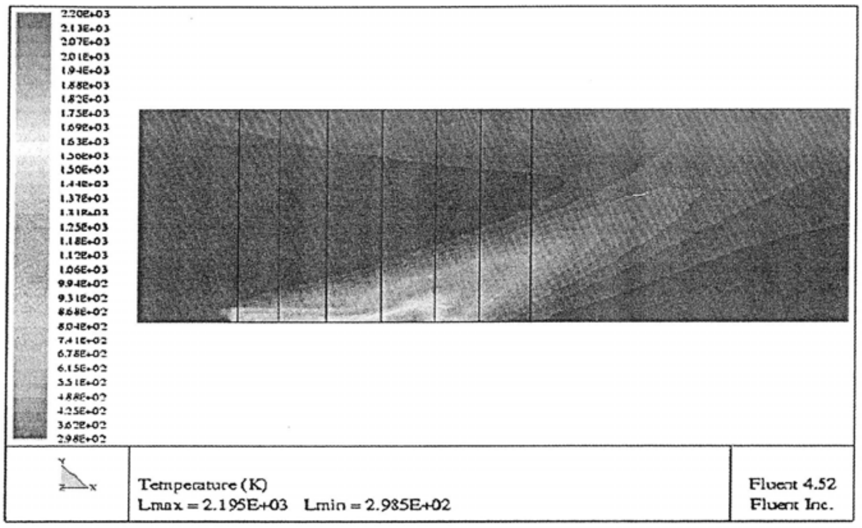

Fig. 7a: Temperature distribution in compartment B for 3.0 kW fire.

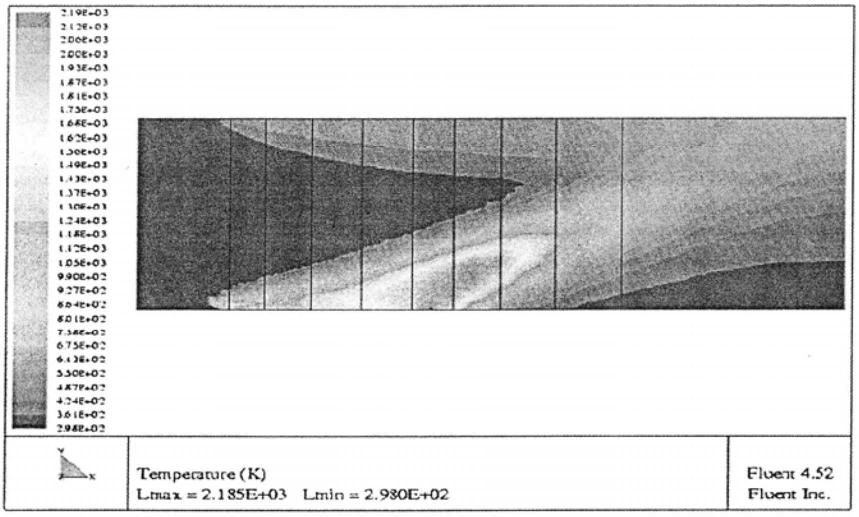

Fig. 7b: Temperature distribution in compartment B for 7.50 kW fire.

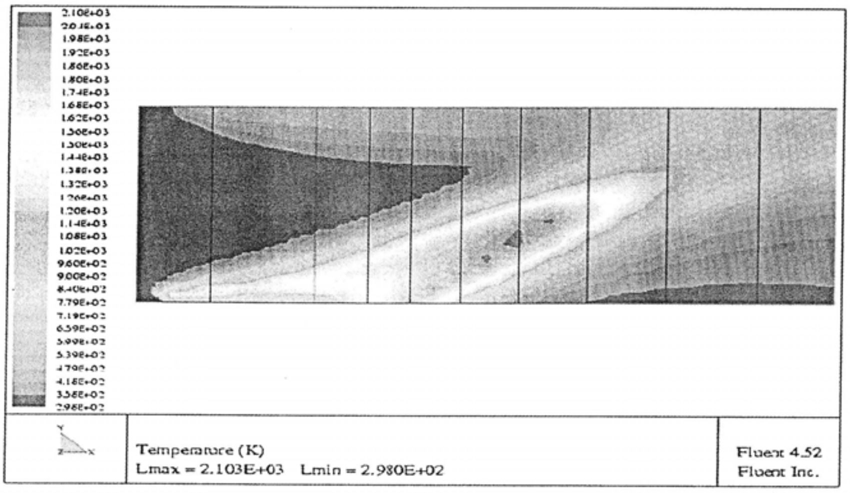

Fig. 7c: Temperature distribution in compartment B for 15.0 kW fire.

5. CONCLUSION

The dynamic fire in four compartments has been studied by performing both experimental and Computational Fluid Dynamic modeling. The results show that the distribution of the fire plume inside the compartment depends greatly on the interaction between the fire with the ventilation flow and compartment walls.

The compartment walls also gave significant effect on the distribution of the fire plume. When the compartment is narrow, the flame has to elongate further downstream for the air entrainment due to the limited spaces at both sides of the fire. In contrast, when the compartment is wide, there are greater tendencies for entrainment from both sides due to more spaces available. Therefore, it would be expected that the local acceleration near the fire seat is greater when the compartment is wide. As a result, the fire would have less deflection angle from the vertical.

Computational Fluid Dynamics was able to predict the behavior of compartment fires similar to the experimental works. The utility of CFD will be the driving force to study fire phenomenon since this technique consume less time and enables us to model fire in a complex geometry.

NOMENCLATURE

k	the turbulence energy m^2s^{-2}
ε	the rate of viscous dissipation of turbulence energy m^2s^{-3}
μ	the turbulence effective viscosity $kgm^{-1}s^{-1}$
C_1	constant
C_2	constant
C_μ	constant
$C_{3\varepsilon}$	constant
σ_k	constant
σ_ε	constant

REFERENCES

[1] A.J.M, Heselden, "Studies of Fire and Smoke Behavior Relevant to Tunnel", Building Research Establishment, CP66/1978, pp 1-14, 1978.

[2] M.Z, Abu Bakar, "Control of Smoke Flow in Tunnel Fires" PhD Thesis, University of Sheffield, UK, 1999.

[3] Y.,Wu and M.Z.A.Bakar, "Control Smoke Flow in Tunnel Fires Using Longitudinal Ventilation Systems – A Study of The Critical Velocity", Fire Safety Journal, Vol 35, page 363-390, 2000 John Wiley, UK.

[4] G.T, Atkinson and Y. Wu, "Smoke Control in Sloping Tunnel" Fire Safety Journal, Vol 27, pp 322-335, 1996.

[5] D.D, Drysdale, " An Introduction to Fire Dynamic", 1985, Wiley and Son.

[6] BS 1042: Section 1:1 "Measurement of Fluid Flow in Closed Conduit", 1989.

[7] B.F. Magnussen and B.H. Hjertager, "On mathematical modelling of turbulent combustion with special emphasis on soot formation and combustion", Proceedings of 16th International Symposium on Combustion, The Combustion Institute, pg 719-729, 1976.

[8] B.E.Launder and D.B. Spalding, "The numerical computation of turbulent flows", Journal Computational Methods Appl Mech Engineering, Vol 3 pg 269-289, 1974.

[9] M. Ljuboja and W. Rodi, "Calculation of turbulent wall jets with an algebraic Reynolds stress models", Journal Fluids Enging Trans, ASME, 1980.

[10] P.J. Woodburn and R.E Britter "CFD Simulations of a tunnel fire- Part 1", Fire Safety Journal, Vol 26, pg 35-62, 1996.

[11] P.J. Woodburn and R.E Britter, "CFD Simulations of a tunnel fire- Part 2", Fire Safety Journal, Vol 26, pg 62-90, 1996.

[12] B.J. Mc Caffrey, "Purely buoyant diffusion flames: some experimental results", National Bureau of Standards, NBSIR 79-1910, 1979.