13th Australasian Fluid Mechanics Conference
Monash University, Melbourne, Australia
13-18 December 1998

PREDICTED SMOKE SPREAD IN MULTI-COMPARTMENT BUILDING USING CFD

H. L. Li[1], J. D. Li[2], S. R. Kennett[3] and M. C. Luo[1]
[1]CESARE, Victoria University of Technology
PO Box 14428 MCMC, Melbourne, Victoria, Australia 8001
[2]School of the Built Environment, Victoria University of Technology
PO Box 14428 MCMC, Melbourne, Victoria, Australia 8001
[3]Ship Structures and Materials Division, Aeronautical and Maritime Research Laboratory
PO Box 4331, Melbourne, Victoria, Australia 3001

ABSTRACT
A new approach in using computational fluid dynamics (CFD) to investigate the smoke spreading in multi-storey building is currently under development. In this approach, the CFD method is applied to each compartment individually. The velocity, temperature and species concentrations are calculated for each compartment starting with the burning compartment. Each doorway provides the boundary conditions for the next compartment. This paper presents a comparison between the predicted results from this new approach and the previous experimental results from a single level three compartments fire environment. The numerical results obtained were found to agree well with the experimental results, and justify the approach used.

INTRODUCTION
About two-thirds of all deaths resulting from compartment fires can be attributed to the effects of smoke (Pitts, 1994). Prediction of smoke spread in multi-compartment building is useful in the design of fire safety system. In the past two decades, substantial work has been taken in understanding the smoke spread in compartment buildings using both experimental and theoretical techniques (Beard, 1991). Recently, CFD (also know as field modelling) is becoming increasingly popular as a tool in fire research (Richard *et* al., 1993). The CFD method provides detailed information of fire related parameters in the simulated environment. However, it is still a very challenging task to predict the smoke spread in multi-compartment buildings. The interaction of complex geometry boundary conditions and several different physical processes, such as turbulence, combustion radiation, buoyancy makes prediction difficult. Traditionally, when using CFD to simulate the dispersion of heat, smoke and other toxic gases from a fire in multi-compartment buildings, the building will be considered as one computation domain (Luo & Beck 1994). The usefulness of CFD is limit by a number of factors, which include:

1) The size of the computer memory will restrict the number of compartments and the size of the building to be investigated.
2) The complexity of the internal building geometry which is difficult to model in a single computation domain.

A research program at the Centre for Environmental Safety and Risk Engineering (CESARE), Victoria University of Technology, is being undertaken to develop an understanding of, and the capability of predicting the smoke spread inside multi-storey compartment buildings. The research described in this paper represents a new approach of using CFD to investigating the fire and smoke spread in the multi-storey building. In this method, the computation domain only applies to each compartment during calculation. The velocity, temperature and species concentrations were calculated starting from the burning compartment. The calculated results at each doorway provide the boundary conditions for the calculation of the next compartment. The predicted results are compared with experimental data of Luo & Beck (1994).

THEORETICAL MODEL
The CFD model is described by three-dimensional transport equations for the conservation of mass, momentum, and enthalpy. The general expression for the conservation equations is:

$$\frac{\partial}{\partial t}(\rho\Phi)+\frac{\partial}{\partial x_i}(\rho u_i\Phi)=\frac{\partial}{\partial x_i}\left(\Gamma_\Phi\frac{\partial\Phi}{\partial x_i}\right)+S_\Phi \qquad (1)$$

Here x_i are the coordinate variables with i=1,2, and 3 and the corresponding velocity components are u_i, ρ the density, Γ_Φ the diffusivity (or viscosity), S_Φ the source, and Φ is a general fluid property variable and it can be velocity, pressure P, turbulent kinetic energy k, energy dissipation ε, enthalpy h, mixture fraction f and mixture fraction variance g.

The following sections describe the major sub-models used in the CFD model

Combustion
For turbulent combustion, the well know mixed-is-burnt model (or equilibrium limit, Bilger, 1976) is used. The fuel concentration and hence the heat release at any particular location was calculated by assuming a Gaussian distribution for the mixture fraction fluctuation with the mean and rms values being calculated from (1).

For a simple diffusion flame, the fuel and oxygen are combined in fixed proportion by mass to yield the product:

$$F + rO \rightarrow (r+1)P \tag{2}$$

Thus 1kg fuel (F) reacts with r kg of oxidant (O) to yield ($r+1$) kg of product (P). It is assumed that the turbulent diffusivities of all species are equal, and the reaction of the fuel gas with the oxidant is one-step and infinitely fast in comparison with mixing rates.

The fuel used in this study is propane (C_3H_8). The combustion reaction for propane is:

$$C_3H_8 + (2+2a+b)O_2 \xrightarrow{\Delta H_{Comb}} aCO_2 + bCO + 4H_2O \tag{3}$$

The net specific energy (NSE) is 46 MJ/kg for pure propane. Experimentally, because of impurities in the fuel, and incomplete nature of the combustion due to soot formation, the assumed enthalpy (ΔH_{Comb}) is 44 MJ/kg. The fuel supply rate specified in the CFD program was set to match the operating conditions of modeled experiment.

Carbon dioxide and carbon monoxide are the products of interest in the combustion in equation (3). The local CO and CO_2 concentrations were calculated by the empirical relationships of Takada & Yung (1992), i.e.,

$$\frac{Y_{CO}}{Y_{CO_2}} = \frac{28a}{44b} \quad \text{and} \quad \frac{44b}{28a} = \frac{60Y_{O_2}}{23} \tag{4}$$

In this study it is assumed that the combustion does not occur when the local oxygen concentration is less than 10%. The effect of this assumption is normally restricted to the fire plume where the fuel concentration is high.

Radiation

The radiation sub-model is based on the discrete transfer method of Lockwood & Shah (1980). In the method, representative rays are tracked within the calculation region, from emission at one boundary until interception at another. The change in intensity I for a given ray may be expressed by the following form,

$$\frac{dI}{ds} = -KI + \frac{K\sigma T^4}{\pi} \tag{5}$$

Where s is distance, σ is Stefan-Boltzmann constant, K is the combined particle and gas absorption coefficient, and it is obtained using the following empirical relationship (Boyd, 1986):

$$K = 0.32 + 0.28\exp(-T/1135) \tag{6}$$

Here T is temperature in Kelvin.

Numerical Scheme and Boundary Conditions

Equation (1) was solved using the control volume method (Patankar, 1980). QUICK scheme was used to discretize the equations. A Hybrid scheme was applied near the boundary. Because the simplicity of the geometry structure in single compartment, a simple rectangular grid has been used to subdivide the physical domain and where considered necessary the staggered gird method and SIMPLER (Patankar, 1980) procedure have been used.

Boundary conditions depend on the specified problems to be solved. In general, at the wall, the velocity, turbulent kinetic energy and dissipation rate were set to zero (Jones & Launder, 1972). Boundary conditions for pressure need only to be specified at a single location. This is in general at the inlet or exit of the compartment. Mixture fraction and its rms.-values are zero in the fresh air and the fuel. Temperature at the boundary needs to be coincide with the compartment surface temperature or, if known, with the heat flux at the wall. In the calculation, both mass flux and heat flux have been balanced for the physical domain.

RESULTS

In this study, the CFD model was used to simulate the three compartment fire experiment by Luo & Beck (1994). The layout for level 1 used in that experiment building is illustrated in Figure 1. The plan dimensions of the prototype layout are 15.6×6.9 m with a ceiling height of 2.4 m.

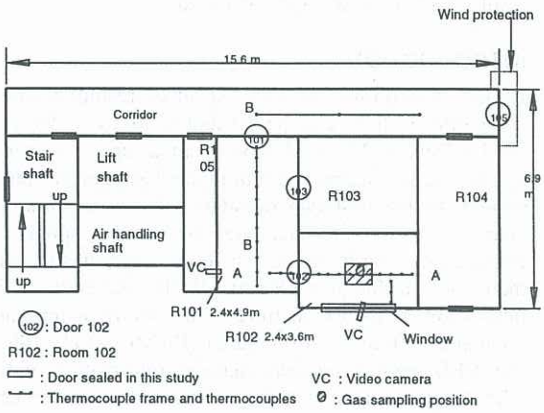

Figure 1: Layout of the first floor of the Experimental Building-Fire Facility and instrumentation

During the experimental, Room 102 was designated as the burning room, representing the room for the fire origin. Door 102 connects Room 102 (burning room) to Room 101; Door 101 connects Room 101 to the corridor. Doors 101 and 102 were open during the experiments. Door 105 is located at one end of the corridor, and was opened to outside. The dimensions for all the doors are 0.8×2.0 m. The energy release estimated from the fire is about 300KW. The temperature and species concentration measurement errors were estimated to be 10% and 15% respectively (He, 1998). For more experimental details, see Luo & Beck (1994).

In the CFD process, each room was calculated separately. In each room, the computational domain is bounded by the walls and the balanced boundary at the door. The doorway will pass flow from one room to another. Burning Room102 was processed first for 500 iterations to build the flow pattern at the doorway, then Room 101 and Corridor was calculated subsequently for 500 iteration each. Afterwards each room was calculated 10 iterations according to the above order for another 100 times until convergence were obtained in all rooms (Table 1). It took a DIGITAL ALPHA STATION (250 4/266) about 14 hour CPU time to achieve the steady state case for 20,000 cells in each computational domain. In the numerical calculation, the temperature of the wall surface of burring room has been set to 200 *°C* as observed in the experimental data. Because only one opening exits for the burning room, both fresh cold air and hot air pass through the door. The temperature for the fresh air entering the building at door 105 has been given as 20 *°C*. The mass flow rate for air out of the door was balanced and equated to fuel supply rate (0.0065 kg/s).

	Room 102	Room 101	Corridor
First Run	500	500	500
Afterward	10	10	10

Table 1: CFD Iteration number fore each room.

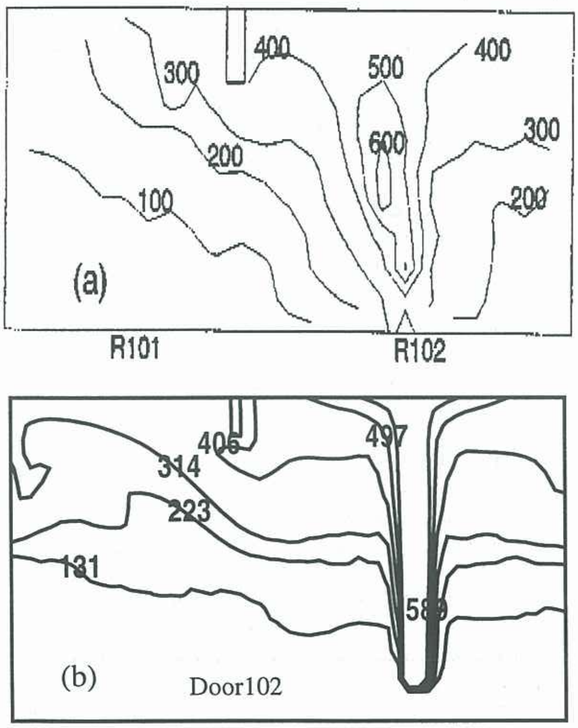

Figure 2: Temperature contours (*°C*) in Room102 and 101 on a vertical section across the centerline of D102 (A-A section of Figure 1): (a) measured; (b) predicted.

Figure 2 shows the temperature field predicted by the CFD model and the temperature field of experiment data. It can be seen that the CFD temperature field in general agrees with the experimental data on a vertical section through Room 102 and 101 (A-A section of Figure 1). The predicted temperatures near the door and roof of the room are very close to the experimental results. However, the predicted highest temperature is higher than that measured. This could be due to the fact that in calculation, the flammability limits for propane were not taken into account.

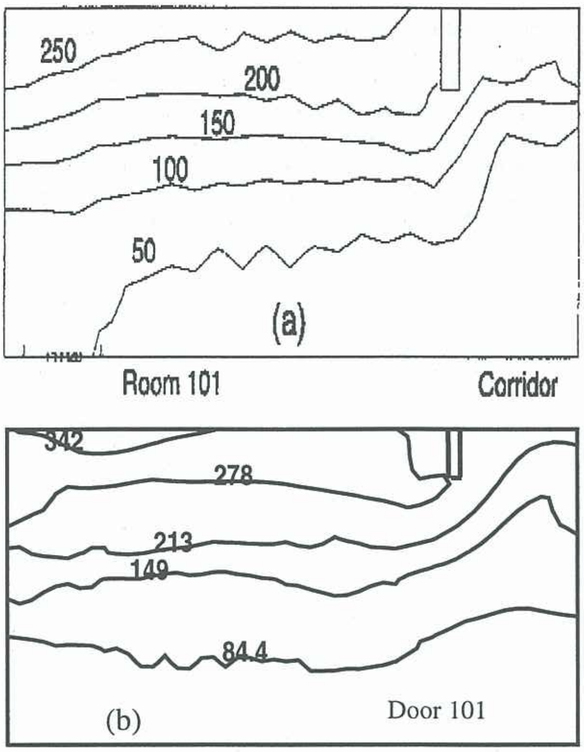

Figure 3: Temperature contours (*°C*) in Room101 and Corridor on a vertical section across the centerline of Door 101 (B-B section of Figure 1): (a) measured; (b) predicted.

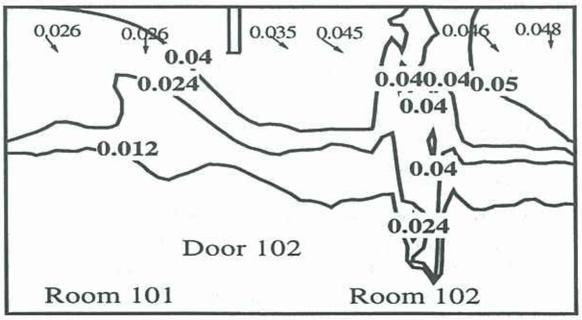

Figure 4: Predicted CO_2 distributions and measured results on a vertical section across the centreline of Door 102 (A-A section of Figure 1); predicted: solid contour lines; measured: data with small arrow mark.

The measured and predicted temperatures on a vertical section through Room101 and the corridor (B-B section of Figure 1) are illustrated in Figure 3. Both the predicted and experimental results are consistent. The measured results give more even temperature distribution than predicted. The location of the 50 *°C* temperature contour for the predicted results is about 0.5 m lower than that obtained from the measurement. Overall, the CFD predicted higher temperature than that measured which mainly due to the smoke with higher predicted temperature flow out from burning room. This shows that using doorway as a drive force can provide satisfied results as multi-rooms fire temperature distribution concerned.

In this CFD study, the smoke spread in the building was monitored by tracking the temperature and CO_2 distributions. Figure 4 shows the CO_2 distribution of predicted measured results at a vertical section in Room 102 and 101. The arrows in the figure marked the measured CO_2 at 2m level above floor. The maximum predicted CO_2 concentration reached 5% in the plume area. The predictions are in general agreement with the measurements. The measured results are more evenly distributed than the predicted results.

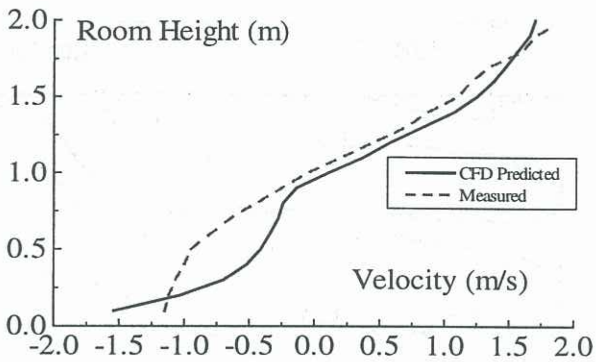

Figure 5: Comparison of the predicted and measured velocities at Door 102: (a) measured; (b) predicted.

Figure 5 shows the comparison for the predicted and measured velocity in the normal to the door direction at the center of Door 102. The predicted distribution is consistent with the experimental results although some difference exists, especially near the bottom of the door. It is felt that the boundary conditions at the door for the experiments and that used in the CFD calculation may be different. However, this will not affect using doorway as driving force to predict the flow in the next rooms

CONCLUSION

A new approach has been developed in using CFD for the prediction of smoke spreading in multi-compartment buildings. Preliminary results calculated for three compartments have been compared with the experimental results. In general, the CFD model gave good predictions of both the magnitude and the trends for the quantities measured in the experimental results although some difference exists in the predicted temperature, velocity and CO_2 in comparison with the experimental results. This may be due to the difficulty in specifying the boundary condition and the limitation of the CFD model. This difference between the prediction and the experimental data for the velocity field will be addressed further because the velocity distribution at the doorway plays a vital role in the current method.

Following this success with the three compartments, this CFD method will be compared to the experimental results for smoke spreading in transient-state and multi-storey compartments. The new CFD approach has the potential to revolutionize the ability of CFD programs as they are applied to multi-storey.

ACKNOWLEDGMENTS

The authors wish to thank the Aeronautical and Maritime Research Laboratory for its Ph.D. scholarship. The kind support provided by Professor V. Beck is also gratefully acknowledged.

REFERENCES

Beard A., "Limitations of Computer Models", Fire Safety Journal 18, 375-391, 1991.

Bilger R. W., "Turbulent Jet Diffusion Flames", Prog. Energy Combust. Sci. 1, 87, 1976.

Boyd, R. K., "Computer Modelling of A Coal Fired Furnace", Ph.D. Thesis, Department of Mechanical Engineering, University of Sydney, Australia, July 1986.

Boyd, R. K. and Kent J. H., "Three-Dimensional Furnace Computer Modelling", In Twenty-first Symposium (International) on Combustion. The Combustion Institute, pp. 265-274, 1986.

He Y., "CESARE-RISK: Fire growth and smoke spread models – Further development and experimental validation", Internal Report, CESARE, Victoria University of Technology, 1998.

Jones, W. P. & Launder B. E., "The Prediction of laminarisation with a two-equation model of turbulence", Int. J. Heat and Mass Transfer, 15, 301-314, 1972.

Kerrison L, Galea E. R, Hoffmann N. and Patel M. K., "A Comparison of a FLOW3D based fire field model with experimental room fire data", Fire Safety Journal 23, 387-411,1994.

Lockwood F. C. & Shah N. C., "A New radiation solution method for incorporation in general combustion prediction procedures", In Eighteenth Symposium (International) on Combustion. The Combustion Institute, 1980, PP 1405-1414.

Luo M. C. & Beck V., "The Fire environment in a multi-room building comparison of predicted and experimental results", Fire Safety Journal, 23(4), 413-438, 1994.

Minkowycz W. J., Sparrow E. M., Schneider G. E. and Pletcher R. H., Handbook of Numerical Heat Transfer, John Wiley & Son, New York, 1988.

Patankar S. V., Numerical Heat Transfer and Fluid Flow, McGraw-Hill, 1980.

Pitts W. M., "Application of Thermodynamic and Detailed Chemical Kinetic Modeling to Understanding Combustion Product Generation in Enclosure Fires", Fire Safety Journal 23, 271-303, 1994.

Richard D. P., Walter W. J. & Richard W. B., "Verification of a Model of Fire and Smoke Transport", Fire Safety Journal 21, 89-129, 1993.

Takada, H. & Yung, D., "Simplified Fire Growth Models For Risk-cost Assessment in Apartment Buildings", Journal of Fire Protection Engineering, 4(2), 53-66, 1992.