Twelfth Australasian Fluid Mechanics Conference
The University of Sydney, Australia 1995

MATHEMATICAL MODELLING OF FIRE INDUCED TURBULENT CEILING JETS

Behdad Moghtaderi and Vasily Novozhilov
Department of Mechanical and Mechatronic Engineering
The University of Sydney
Sydney, NSW
Australia

ABSTRACT

A mathematical field model has been used to simulate the characteristics of fire induced turbulent ceiling jets. The model involves the numerical solution of basic equations governing three-dimensional steady and transient flows with prescribed boundary conditions. The effect of turbulence, combustion and radiation are included with various sub-models.

Comparison of the model predictions and experimental data of Motevalli *et al,* 1990, are summarised in this paper. Apart from the validation exercise, the effect of the buoyancy treatment in the turbulence model is also investigated in detail.

The model predicts the flow behaviour quite well. Temperature and velocity distributions inside the ceiling jet show very good agreement with data. The calculations have highlighted the importance of the turbulence modelling on the predicted temperature stratification.

INTRODUCTION

In the event of a fire, hot gases in the fire plume rise vertically above the burning fuel and impinge on the ceiling. The ceiling surface causes the plume to turn sideways and move horizontally beneath the ceiling to the other areas remote from the fire source. This relatively rapid gas flow under the ceiling surface which is driven by the buoyancy forces is called the "Ceiling Jet". The study of fire induced ceiling jets is of considerable interest, since much of the hardware associated with detection and suppression of fires is installed within this layer.

Studies quantifying the ceiling jet flows have been conducted since the 1950s. The early studies were based on simple theoretical models, empirical relations and dimensional analysis. However, with the rapid development of computational fluid dynamics (CFD) in the last twenty years, the field modelling technique becomes the convenient way of doing such studies. It is well known, however, that the field models need to be extensively validated against experimental data to assess their reliability.

Some field model studies of ceiling jet flows have been reported since the 1980s (*i.e.* Satoh, 1988; Chow *et al*, 1993, etc) but most of them were not verified with experimental data. In this paper, a field model developed at the University of Sydney is validated against some experimental data. The importance of the buoyancy forces from the turbulence modelling point of view will be also examined.

THEORETICAL MODEL

A general description of the computational model known as FIRE has been given in earlier work (Novozhilov *et al*, 1995). It suffice to say that it is a CFD code for two or three-dimensional combustion situations. It can be applied to solid, liquid or gas fuelled fires. Transport equations of mass, momentum and energy are solved by a finite volume scheme (Patankar, 1980). Turbulence is modelled by the two equation k-ε model with terms for buoyancy effects. Combustion can be modelled by the mixture fraction concept (Bilger, 1989) or by the eddy breakup model (Spalding, 1971). The conservation equations are cast into the following form:

$$\frac{\partial}{\partial t}(\rho\phi) + \nabla.(\rho u\phi) = \nabla.(\Gamma_\phi \nabla\phi) + S_\phi \qquad (1)$$

where ϕ is the conserved variable; ρ is the density; u is the gas velocity; Γ_ϕ is the diffusion coefficient for ϕ and S_ϕ is the transport equation source term.

Radiation heat exchange is modelled using the discrete transfer method of Lockwood *et al*, 1982. Heat losses through the solid boundary can be calculated by an integral model or the numerical solution of the heat conduction equation.

THE EXPERIMENT

A series of experiments were conducted by Motevalli *et al*, 1990, to investigate fire induced turbulent ceiling jet flows. The experimental setup (Fig. 1) consists of a large ceiling with a diameter of 2.13 m and two different heights (0.5 and 1 m). The ceiling was constructed from a fibreboard type material and was insulated on the top by fibre glass insulation. Small fires of 0.5-2 kw were produced with a 27 mm premixed methane-air burner which was located on the floor in the centre. Detailed velocity and temperature measurements of the ceiling jet were made simultaneously using the cross-correlation velocimetry technique. The steady state experimental results corresponding to the fire strength of 2 kw and ceiling height of 1 m are considered here.

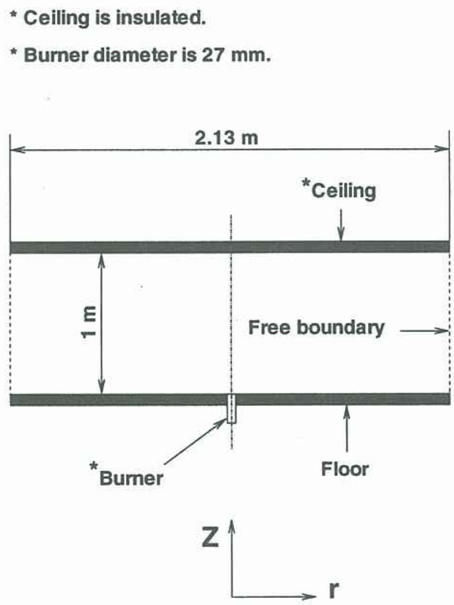

FIGURE 1 EXPERIMENTAL SETUP OF Motevalli *et al* 1990.

COMPUTATIONAL DETAILS

Due to symmetry only a quarter of the geometry is considered in this study and it is modelled by a Cartesian grid. The standard flow grid consists of 39 cells in the X, 39 in the Y and 59 in the Z direction. The geometry also has been modelled with a 45×45×70 and a 60×60×90 grid to determine the effect of grid fineness on the predictions. Increased grid resolution near the ceiling and in the burner region enable the fire plume and the ceiling jet flow to be represented more accurately. The radiation grid need not be as fine and 15, 15, and 10 cells are used in the X, Y, and Z directions, respectively. Free boundaries are sufficiently remote that they exert no unphysical effect on the solution. Since fuel and air are premixed, combustion need not be modelled. The burner delivers hot gas products at the flame temperature which is about 2200 K at atmospheric pressure for methane in air (Glassman, 1977). This is the major difference between this work and that of Kumar *et al*, 1994, in which they used a diffusion flame for simulating the similar experimental data.

Approximately 700 iterations are required to reach a converged solution for the conditions described. A converged solution is defined when the global error for the enthalpy is 0.4% or less and the mean residual for each of the velocity components is less than 0.001 m/s.

RESULTS AND DISCUSSION

This section presents the results of this study in which the FIRE code has been applied to simulate the experiment of Motevalli *et al*, 1990.

The standard flow grid was found to be sufficient for the present study. Predicted velocity profiles changed by less than 6% with the very fine grid, so the standard grid was used for all predictions.

Figures 3 and 4 illustrate the predicted and measured temperature and velocity fields inside the ceiling jet as a function of r/H which is defined as the ratio of the radial location (r) to the ceiling height (H). The gas temperature rise in these figures is the difference between the actual gas temperature in the ceiling jet and the ambient temperature which is 25 °C.

The comparison between experiment and calculation appears quite reasonable. In particular, as Fig. 3 indicates, the energy balance which is a critical factor in any attempt to simulate the fire, has been well predicted. There is however, a tendency to under-predict both velocity and temperature profiles. It can be seen that the worst discrepancy between the predictions and the experimental data happens at radial locations close to the fire source (*i.e.* r/H = 0.26). This can be attributed partly to the problems associated with numerical solution (*i.e.* assumptions made, numerical diffusion, ect) and partly to some uncertainties in the velocity measurements because of the turning of the flow. In all cases, however, the maximum errors for the velocity and excess temperature profiles are less than 14% and 8% respectively.

As both of the figures 3 and 4 show, the bigger is the r/H ratio the more stratified is the ceiling jet flow. This can be explained in terms of the effects of the buoyancy forces on the flow. An important parameter in characterising these effects is the Richardson number (Ri) which is defined as the ratio of the buoyancy forces acting over a height corresponding to the jet thickness, to the momentum flux in the jet as follows:

$$Ri = \frac{(\Delta\rho g \delta)}{(\rho_c V_c^2)} \quad (2)$$

Here, $\Delta\rho$ is the difference in the density between the fluid in the jet to the fluid adjacent to it, g is the acceleration of the gravity, δ is the jet thickness, and ρ_c and V_c are the density and characteristic velocity in the ceiling jet. When the Richardson number is less than unity the flow is dominated by momentum effects and the jet entrains cooler fluid from below. Once the Richardson number exceeds unity, further entrainment into the ceiling jet is suppressed because the increase of buoyancy leads to reduced turbulence and reduction in the mixing. Consequently the ceiling jet moves in the form of a stratified layer with constant thickness. The calculated Richardson numbers in this study are in a good agreement with the values calculated from the experimental data (Fig. 2). In particular, the critical Richardson number (Ri = 1) is reached at (r/H = 0.7) which is quite consistent with the value reported by Motevalli *et al*, 1991.

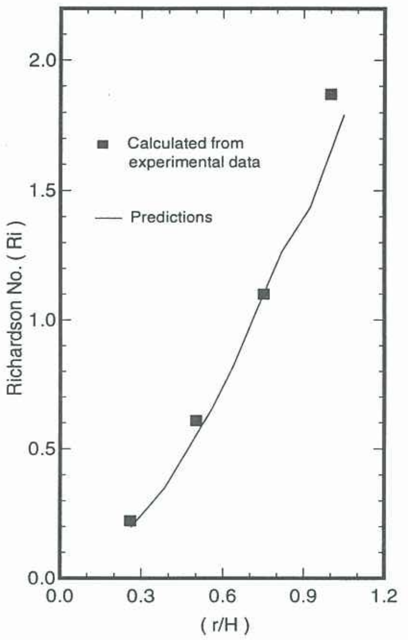

FIGURE 2 RADIAL PROFILE OF THE Richardson NO.

In order to investigate the effects of the buoyancy terms on the turbulence model predictions, a number of additional calculations were performed. Results are summarised in Fig. 5 for two radial locations (r/H = 0.5 and r/H = 0.75) on both sides of the radial location of the critical Richardson number.

Compared to the base case, there was no significant difference in the prediction when the buoyancy term in the ε equation was ignored. However, with no buoyancy effects in either k and ε equations, a significant change, occurred at (r/H = 0.75) where Ri is greater than unity and flow is dominated by buoyancy forces. In this case the stratified layer almost disappeared and was replaced by a more uniform thermal layer. This is similar to the results reported by Fletcher *et al*, 1995.

CONCLUSIONS

This paper has provided a general outline of the mathematical and physical basis of a field model known as FIRE that has been used for the prediction of fire induced ceiling jet flows. The validation studies that have been performed indicate that the major flow characteristics resulting from the fire source can be predicted by the model. Quantitative comparison is good for both velocity and temperature data, particularly for the regions away from the turning point of the flow.

It was also noted that the buoyancy modifications to the turbulence model do improve the realism of the predictions, so they should not be ignored.

ACKNOWLEDGMENT

This work was funded via an Australian Research Council Collaborative award with the WorkCover Authority of NSW and Wormalds. We also would like to thank Professor J.H. Kent from the University of Sydney for his support and assistance.

REFERENCES

Bilger, R.W., 1989, "Turbulent Diffusion Flames", *Ann. Rev. Fluid Mech.*, Vol. 21, pp. 101-135.

Chow, W.K., 1993, "Numerical Studies on the Transient Behaviour of a Fire Plume and Ceiling Jet", *Mathl. Comput. Modelling*, Vol. 17, No. 9, pp. 71-79.

Fletcher, D.F., Kent, J.H., Apte, V.B. and Green, A.R., 1995, "Numerical Simulation of Smoke Movement from a Pool Fire in a Ventilated Tunnel", *Fire Safety Journal*, No. 3, pp. 305-325.

Glassman, I., 1977, "Combustion", Academic Press.

Kumar, S. and Yehia, M., 1994, "Mathematical Field Modelling of Transient Ceiling Jet Characteristics in Fires", *Proceedings, The First Joint International ASME-ISHMT Conference*, Bambay, India.

Lockwood, F.C. and Shah, N.G, 1981, "A New Radiation Method for Incorporating in General Combustion Prediction Procedure", *Proceedings, 18th Symposium (International) on Combustion*, The Combustion Institute, pp. 1405-1414.

Motevalli, V. and Marks, C.H., 1990,"Transient Characteristics of Unconfined Fire-Plume-Driven Ceiling Jets", National Institute of Science and Technology, NIST-GCR-90-574.

Motevalli, V. and Marks, C.H., 1991,"Characterising the Unconfined Ceiling Jet Under Steady-State Conditions: A Reassessment", *Fire Safety Science - Proceedings of the Third International Symposium*, pp. 301-312.

Novozhilov, V., Moghtaderi, B., Fletcher, D.F and Kent, J.H., 1995, "CFD Modelling of Wood Combustion", *Proceedings of the Second Asia-Oceania Symposium on Fire Science and Technology*, Khabarovsk, Russia.

Patankar, S.V., 1980, "Numerical Heat Transfer and Fluid Flow", Hemisphere Publishing Corporation.

Satoh, K., 1988, "A Numerical Study of Ceiling Jets Based on 'T' Pattern Flames", *Fire Safety Science - Proceedings of the Second International Symposium*, pp. 159-168.

Spalding, D.B., 1971, "Mixing and Chemical Reaction in Steady Confined Turbulent Flames", *Proceedings, 13th Symposium (International) on Combustion*, The Combustion Institute, pp. 649-657.

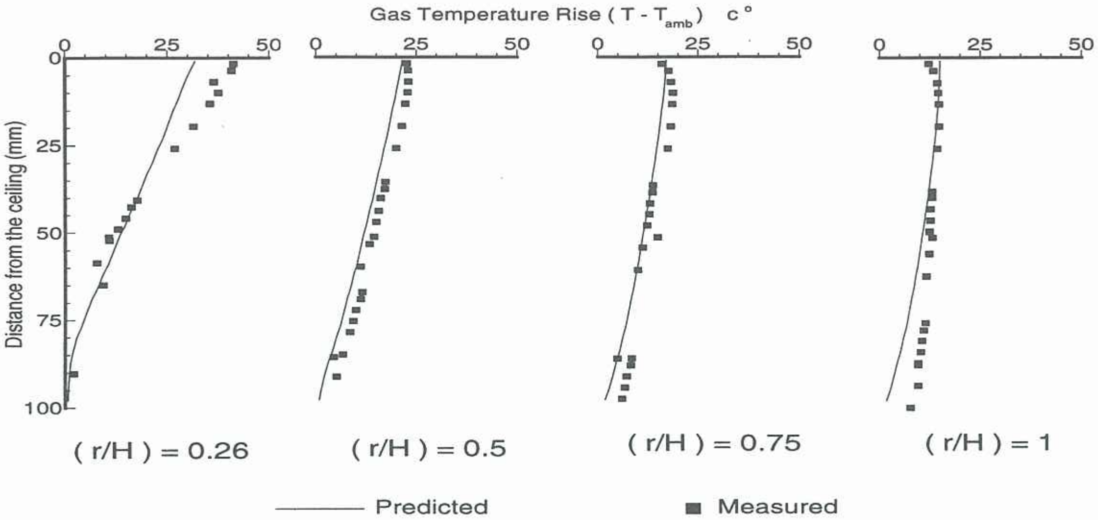

FIGURE 3 COMPARISON OF THE PREDICTED AND MEASURED TEMPERATURE PROFILES.

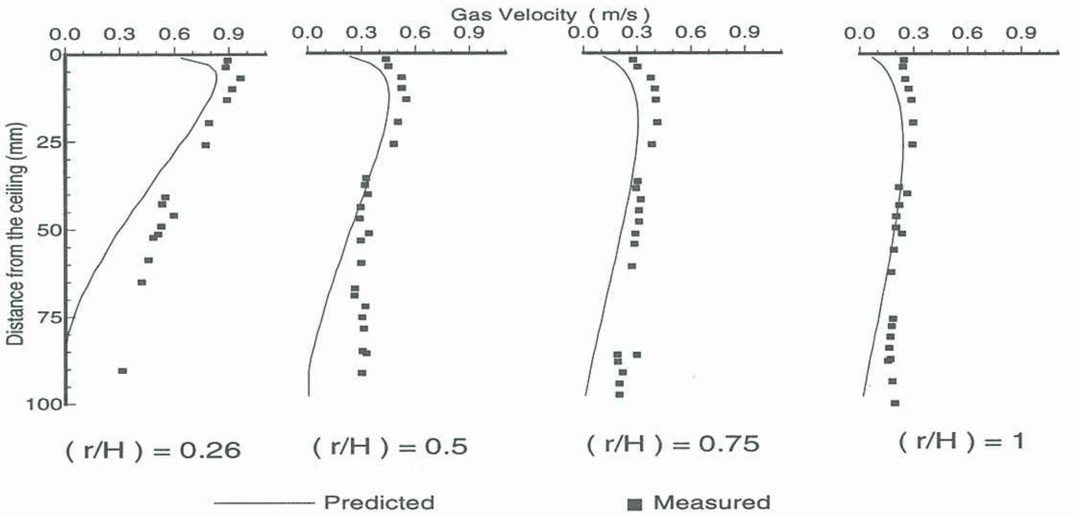

FIGURE 4 COMPARISON OF THE PREDICTED AND MEASURED VELOCITY PROFILES.

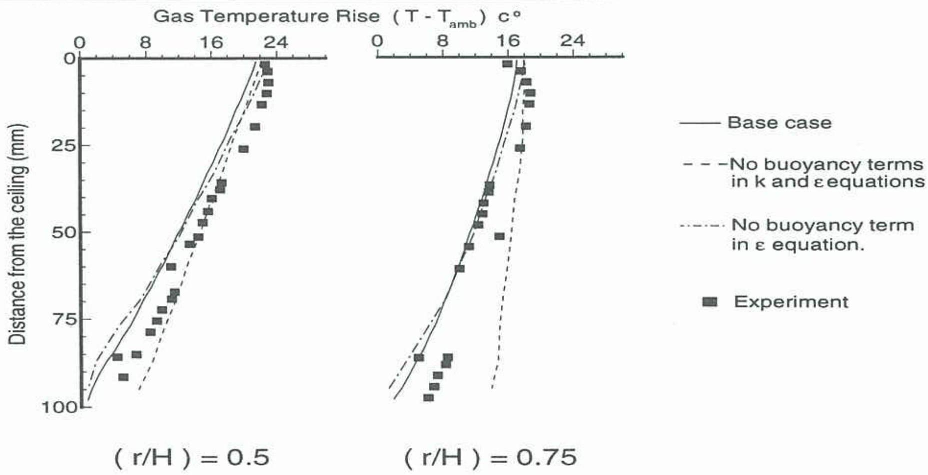

FIGURE 5 EFFECTS OF BUOYANCY TERMS ON PREDICTIONS OF TEMPERATURE PROFILES.