5713

If you don't like the graphics, go home and watch the radio

N RHODES, MSc, CEng, FIMechE
Mott MacDonald Group, Croydon, UK.

SYNOPSIS This paper, based on Computational Fluid Dynamics (CFD) techniques, describes a mathematical model which has been used for the prediction of smoke movement. The model provides a numerical solution of the basic equations governing two- and three-dimensional transient flow with prescribed boundary conditions, and includes the effects of turbulence and combustion. The application described in the paper is to a landbridge structure covering the Vancouver Advanced Light Rapid Transit and British Columbia Railway lines. The study has determined the smoke movement in order to evaluate evacuation scenarios and identify whether additional ventilation has to be provided. Animation techniques have been applied to the results in order to illustrate the consequences of design changes.

NOTATION

S_ϕ - source term
$\vec{v}_\phi$ - velocity vector
t - time
ρ - density
ϕ - the dependent variable
Γ_ϕ - diffusion coefficient

1 INTRODUCTION

Now, at last, it is no longer necessary to accept a poor understanding of any fluid flow problem. The techniques of CFD can address the three-dimensionality, transience and turbulence of the flow, and provide a foundation for building more realistic models of virtually any process.

The difficulty of solving the fundamental equations of fluid motion for realistic situations has imposed a restriction on engineering designers. To overcome this restriction, we have ventured through the avenues of barely understandable mathematical solutions on the one hand and empirical approximation on the other. These tools have served us well enough, but from time to time engineers have had to rely on design codes and loose approximations which are not always well founded.

CFD has uncovered some of the flaws in "design-code thinking", and supported others. For example, smoke movement in a large atrium is different from a household room. Thus, empirical models derived from small scale experiments can fail when applied to larger situations.

There are, of course, approximations and difficulties, but I believe these are far outweighed by the advantages. The consideration of the full scale in any novel design problem and the deeper understanding available from post-processing of CFD results are two benefits. CFD tools are becoming easier to use, but they still require some specialisation on the part of the user. Like any other tool, it can be used badly or well, with commensurate results.

This paper describes the use of CFD for predicting smoke movement arising from accidental fires. The basic physics and chemistry of fires are complex and difficult to predict with complete confidence; they involve unsteady, turbulent flow behaviour coupled with dominating thermal effects. Computer simulations of the full scale problem can address these complexities. The specific features of a design can be included, and it is relatively easy to examine different cases. Versatility is perhaps the greatest strength of computer simulations - the boundary values, such as the smoke ventilation arrangements can be altered as easily as the geometry of the structure. Great care is required in setting up such a model and ensuring the virtual absence of numerical inaccuracies, as well as a knowledge of the mathematical models employed and their particular limitations. Reference 1 provides further information on validation of models.

In the context of the detailed design process, the problem is a very broad one, and involves the consideration of system ventilation, personnel evacuation and escape routes, alarms and instrumentation, the formulation of emergency procedures and operator training. Liaison with the

Fire Brigade and other bodies is essential to ensure that all eventualities are covered and all points of view are considered. A rigorous evaluation of smoke behaviour through the use of modelling can provide an invaluable input to this larger requirement.

The analysis of CFD calculations can be time-consuming, requiring the generation and inspection of many plots of, for example, smoke concentration, velocity vectors and so on. The designer has to absorb the information obtained from literally millions of data points in a transient calculation. Work recently carried out by the author and his team has utilised animation techniques in conjunction with CFD to allow a quicker and more precise comprehension of the flow behaviour and the resulting spread of smoke from fires. The animation provides a visualisation of the smoke behaviour, giving both an overview and an insight into the most important features of any given scenario. The effect of control measures, such as emergency ventilation, can be assessed and various design options compared.

The paper describes the main features of the physical situation of the Vancouver ALRT, the mathematical model employed for prediction and the animation produced to provide a clear understanding of the flow behaviour.

2 PHYSICAL SITUATION

The Advanced Light Rapid Transit (ALRT) landbridge is a structural enclosure some 150m in length by about 50m wide. It covers a railway, an emergency service road and the ALRT track. Variations in width, cross-sectional area and ceiling height result from the slope of the ALRT track and other architectural features are shown (refer to Figure 1). The landbridge is open to atmosphere at its Western end which coincides with an ALRT station, and at the Eastern end where there is an open cross-section between the exit and a road bridge, effectively forming a vent.

3 MATHEMATICAL MODEL

The equations that need to be solved for any fluid-dynamic problem can be written in the following general form:

$$\frac{\partial}{\partial t}(\rho\phi) + \mathrm{div}\,(\rho\vec{v}_\phi - \Gamma_\phi \mathrm{grad}\,\phi) = S_\phi$$

The computer solves a set of these equations, which are obtained by integrating over a control volume. The solution methods are well established, reliable and widely used. Further details can be found in Reference 2. The dependent variable can represent the fluid velocities in each coordinate direction, enthalpy, species concentrations, turbulence quantities, radiation fluxes and so on. The number of differential equations that can be used to represent a problem need not be limited when such a computational approach is adopted. The limitation is more likely to be in the degree of understanding of the relevant physics, and whether suitable exchange coefficients and source terms can be formulated.

In the present study a three-dimensional model was developed, in which equations were solved for the three velocity components, enthalpy, pressure and smoke concentration. The effects of turbulence were included by using a two-equation turbulence model in which the kinetic energy of turbulence and the rate of dissipation of turbulence energy were solved. Auxiliary relationships were utilised to determine the temperature and density.

The equations are solved numerically on a grid comprising 50 cells in the axial direction, 17 cells vertically and 22 cells laterally. Thus all of the above variables are predicted at each cell location. Timesteps of 15s and 30s were used in the transient, which covered an overall time of 67 minutes. The model includes the effects of buoyancy, and heat transfer and wall friction are included through the use of standard wall functions.

3.1 Boundary Conditions

The fire location is presumed to be in the middle of an ALRT train situated at the lower end of the landbridge. Thus the actual fire source is 40m from the lower portal. The figure below illustrates the calculation domain and the main boundary locations.

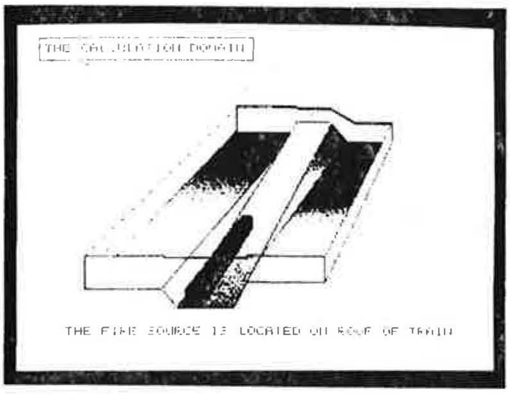
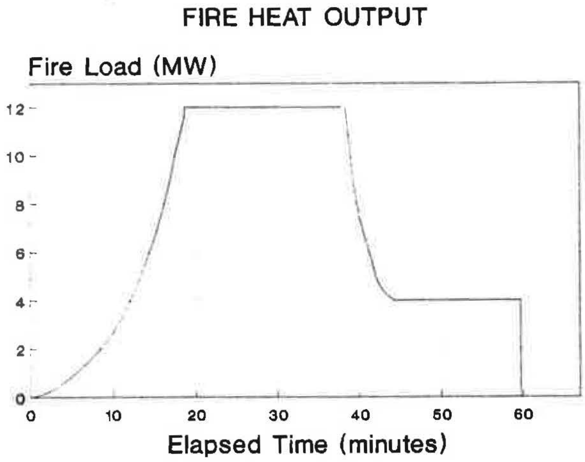


Fig 1 The calculation domain

Apart from the wall friction and heat transfer boundary conditions mentioned previously, the following additional boundaries are imposed:

At the fire source the heat load is prescribed, which varies with time (see graph). In addition, the smoke concentration at the fire source is set to unity.



A sink term is provided to absorb the combustion air. It is assumed that this air is entrained into the burning vehicle.

At the portals a zero uniform pressure is prescribed.

4 PRESENTATION OF RESULTS

The results of the study are presented in the form of smoke contour plots at a number of times during the transient. These are taken from the animation of the CFD prediction.

Figure 2 shows a cross-section in the plane of the fire at times of 2, 4, 8, 12 and 18 minutes. The fire develops slowly in the early stages, and after 2 minutes smoke has risen between the train roof and the ceiling. At subsequent times it can be seen that smoke has spread upwards to cover the ceiling above the B.C. railway track. No smoke has spread to the other side. As the fire develops, the smoke layer thickens and becomes both deeper and more concentrated above the B.C. railway. The curtaining effect of two vertical members can be seen at this section. These are solid structural supports about 1m deep and act as smoke curtains, containing the higher concentration of smoke above the fire and inhibiting lateral spread. At 12 minutes the smoke layer is considerably deeper and of higher concentration. However, the region above the roadway is still completely clear.

Although not shown in these figures, this situation remains the same whilst the fire output is at 12MW. When the fire diminishes, the smoke gradually clears away.

Figure 3 shows the smoke concentration contours 0.2m below the ceiling. Note that the absolute ceiling height varies due to the geometry variations; our results also follow this variation. At 2 minutes the low concentration contour shows a small region above the fire source. The support structures have contained the smoke sufficiently to promote more rapid spreading along the inverted channel formed by this feature. Above the B.C. railway, the extent of smoke spread is noticeable. At 4 minutes the effect of the support structure can be seen, particularly in that it stops smoke spreading across the roadway. Smoke at low concentration occupies the whole ceiling region above the B.C. railway. At 10 minutes this begins to increase to higher concentrations. As with the previous cross-section, the smoke begins to dissipate when the fire source diminishes.

Figure 4 shows two side views of the fire development, the upper plane taken through the train, and the lower one through the B.C. railway.

At 2 minutes, smoke is only evident above the train. By 4 minutes it has spread along the inverted channel and has begun to appear above the B.C. railway, having spread laterally. The situation gradually worsens as the fire develops. At 12 and 18 minutes the smoke layer is thickest above the B.C. railway and ALRT tracks towards the West Portal. Evacuation strategies would clearly need to be devised to avoid these regions.

The illustrations are taken directly from a computer animation of the flow. In the still photograph form, they revert to the commonplace graphic which CFD can readily produce and which requires mental integration in a transient process such as the one described. They are available on floppy disc or video for replay of the moving image.

The animated result opens up the science of CFD modelling to a greater audience. The user can more easily identify errors in model formulation, safety authorities can perceive hazardous situations with greater depth and clarity of concept and safety managers can judge the influence of remedial actions such as forced ventilation.

Animation can also play an important part in operator training. Straight replay of simulated smoke movement is far more memorable than a document full of graphs of smoke concentration. They may ultimately provide inputs to 'virtual reality' simulations where the operator can experience a complete involvement in the accident scenario.

5 CONCLUDING REMARKS

This paper has described a study carried out to predict the behaviour of smoke in the Vancouver ALRT landbridge. A mathematical model has been developed to represent the main features of the landbridge, and applied to predict the smoke for a particular fire scenario and still air conditions.

The geometry of the landbridge and the gradients which cause the smoke to move in preferred directions result in a problem which is too complex to solve using any simple method. The combination of CFD and animation techniques has enabled an understanding of the flow behaviour to be easily gained.

The fundamental basis of the model allows a rigorous assessment of the design options without empirical approximations. For example, the smoke layer above the B.C. railway could be alleviated by introducing a vent in the ceiling -the effect of this could easily be investigated.

Animation of the CFD predictions provides a concise and easily understandable form of presentation of the results. This has benefits not only to engineers concerned with this type of study, where communication of ideas can be significantly improved, but also fire and safety officers, station managers, operators and staff, providing a valuable training aid for safety management.

6 REFERENCES

1. Rhodes, N, Clark I W and Else, K. 1991 "Prediction of Smoke Movement Using CFD". IMechE EuroTech Conference Paper C413/067

2. Burns, A D and Wilkes, N S. 1987 A Finite Difference Method for the Computational of Fluid Flows in Complex Three-Dimensional Geometries. AERE-R 12342.

3. Rhodes, N, Lowndes J F L and Symons, G E M. 1991 "Fire Modelling for Good Design". Inst. of Energy Publ. FIRE AND EXPLOSION HAZARDS.

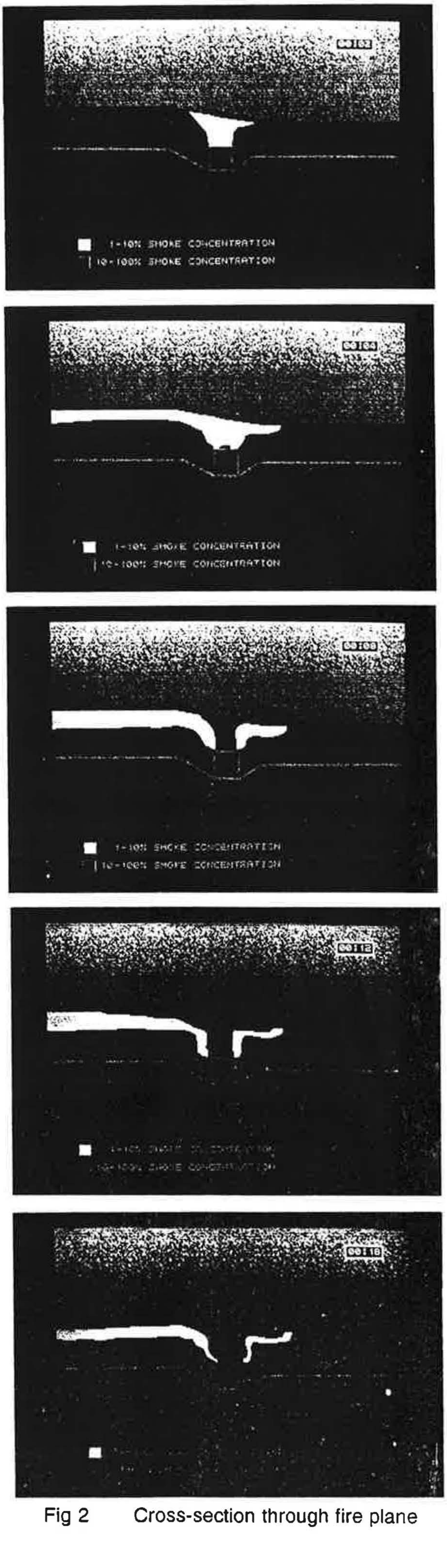


Fig 2 Cross-section through fire plane

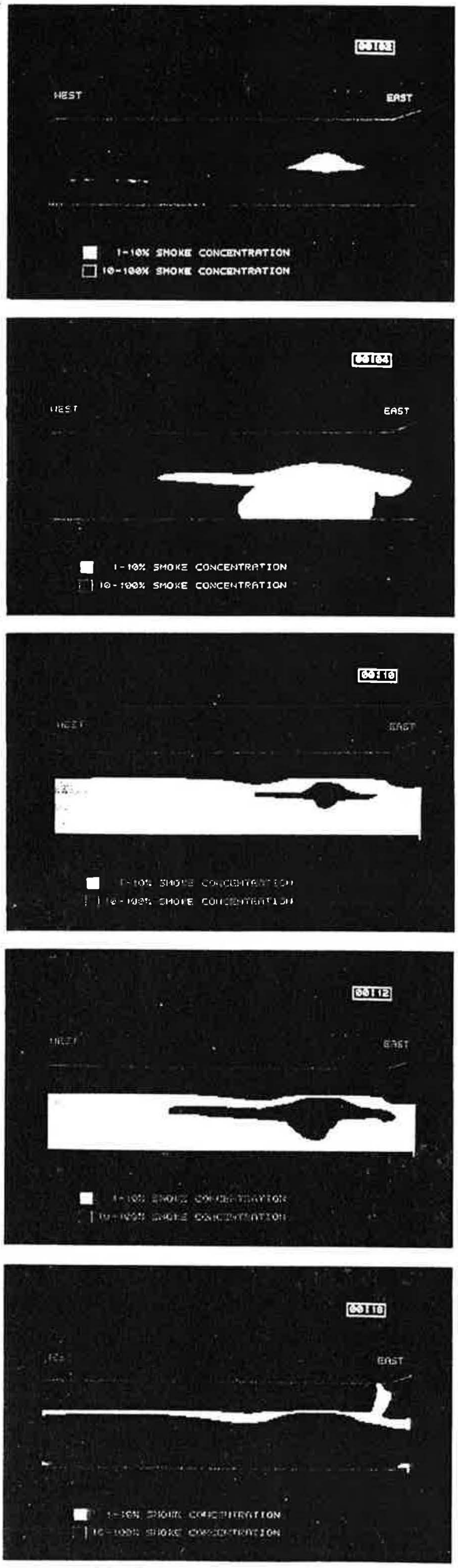


Fig 3 Plan view just below ceiling

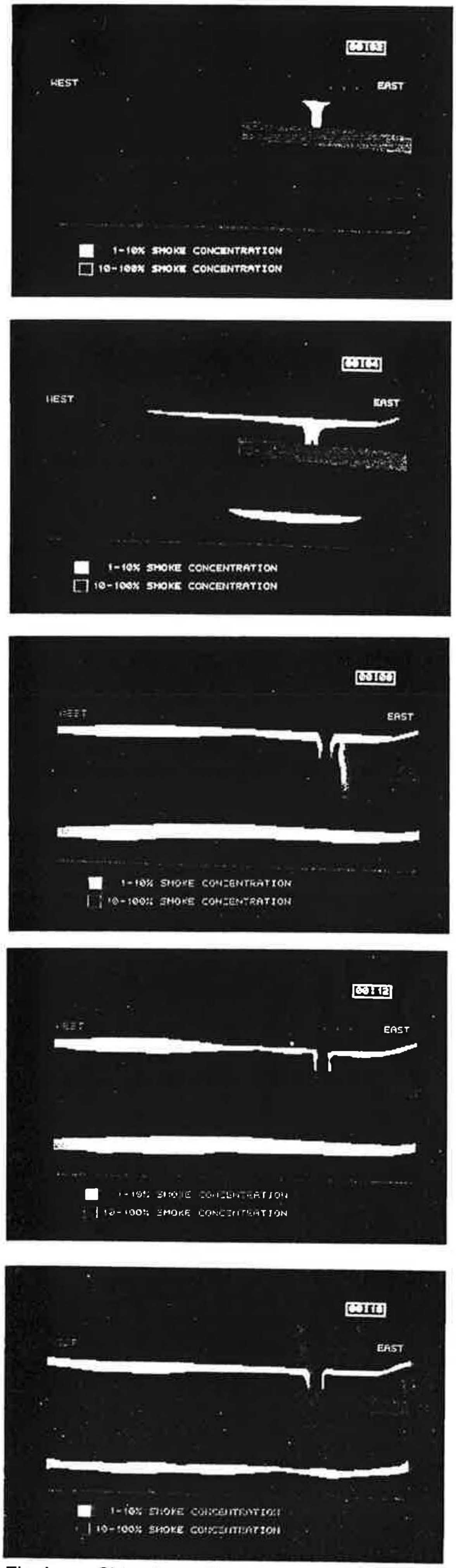


Fig 4 Side views (fire plane and BC railway)

The role of CFD modelling in design for safety and environmental management

S V EMSILE, BSc, CEng, MIMechE, MCIBSE
Division of Thermo-Fluid and Environmental Engineering University of Strathclyde, Glasgow, UK.

SYNOPSIS Faced with ever increasing Health & Safety and Environmental legislation, the Environmental Engineer must use appropriate tools and techniques to design ventilation systems which protect the health of workers and minimise damage to the external environment. This paper presents the results of a practical study using an established CFD code where the objective was to either upgrade or replace an existing ventilation system in an analytical laboratory where problems with control over employee exposure to mercury vapour were being experienced. The benefits and some of the problems of applying CFD techniques in this study are outlined.

Due to factors outwith the author's control, the full results of the study were unavailable by the publication deadline for submission of the paper. Interested readers who are unable to attend the seminar may write to the author at the above address to obtain a copy.