J. of Fire Prot. Engr., 1 (1), 1989, pp 11-22

On the Field Modelling Approach to the Simulation of Enclosure Fires

Edwin Galea
Centre for Numerical Modelling and Process Analysis
Thames Polytechnic
London SE 18, England

SUMMARY

This paper considers the mathematical field modeling approach to the description of enclosed fires. Field modeling is emerging as the "new technology" for the simulation of the fire in enclosure scenario. A number of field modeling packages are discussed as well as their application to aircraft cabin fires, fire-sprinkler interaction, shopping mall fires, atria fires and fires in sports stadia.

INTRODUCTION

Uncontrolled combustion, or fire, often occurs on colossal scales such as in forest fires or conflagrations resulting from massive oil spills. These areas are attracting research interest[1,2], however, central to current research is the enclosure fire. This broad category encompasses domestic, aircraft cabin, airport terminal, tunnel, indoor shopping centre, warehouse and multi-storey office blocks. By the very nature of enclosure fires, i.e., non-frequent occurrence, the almost total destruction of evidence and the extremely complex nature of the physics and chemistry involved, only the barest minimum may be learned from fire accident investigations. To uncover details concerning the fire dynamics involved and the potential fatal hazards responsible for preventing the occupants from escaping, it is necessary to perform simulations of possible fire scenarios. These simulations may be either experimental fire tests or numerically based mathematical computer models.

EXPERIMENTAL OR MATHEMATICAL MODELS?

Experimental simulations require the construction of either a full scale test compartment or its scaled down model, with an internal structure fitted in as complete a manner as possible. Extrapolating results from scale models can be dangerous because reliable scaling rules do not always exist. The construction of full-size completely fitted enclosures such as the interior of a Boeing 747 or tall atria buildings such as the new Lloyds of London building can be extremely expensive, even impossible. Furthermore, several fire tests may be required to take into account different fire scenarios and to accommodate the large variety of available furnishings. Among a number of establishments, the Fire Research Station (FRS) in Britain and the US Federal Aviation Authority (FAA) have conducted full-scale fire tests in enclosures, such as domestic rooms, road tunnels[3,4,5] and a variety of aircraft fuselages including, DC-7, B-737 and a Lockheed C-133[6,7,8].

Much has been learned from these investigations about how fires begin and propagate over a variety of fuels. However, instead of relying on fire accident investigations or experimental fire simulations with real fires, the computer offers a more fundamental approach to the study of the spread of fire and its associated hazards. Mathematical modelling by computer also offers a cheaper and more general alternative, provided that the models can be reliably validated.

MATHEMATICAL MODELS

Mathematical modelling of compartment fires has been underway for a number of years. There are two fundamentally different approaches to the modelling of fires; probablistic and deterministic. The probabilistic[9,10,11] or stochastic approach involves the assessment of the probable fire risk in an enclosure by associating a finite probability to all fire influencing parameters such as distribution of fuel, number and extent of enclosure openings, human behaviour, etc. This approach, while useful in suggesting the likelihood of a fire in a given enclosure tells us little about the production and distribution of fire characteristics such as smoke concentration, temperature, smoke movement, etc. If this information is desired, the deterministic modelling approach must be used. This can employ two contrasting strategies, zone and field modelling. Zone modelling is the established approach while the field modelling formulation is emerging as the "new technology" for the modelling of enclosure fires.

Zone Modelling

Zone modelling pre-dates the use of digital computers for the treatment of fire problems. Modern zone models can trace their ancestry back to the early sixties when Thomas *et. al.*[12] constructed simple three zone roof venting models. This model was not "computerized" but presented in the form of nomograms. In zone modelling, the hypothetical burning enclosure is divided up into several distinct zones or regions of consistent fire behaviour. The number of zones may be as few as one but typically there would be four or five. The possible zones in a compartment fire might be the flaming combustion zone, the rising thermal plume above the combustion zone, the hot gas layer accumulated under the ceiling, the lower cold layer of air, etc. Within these zones, experimentally based empirical expressions are used to describe the physical behaviour of the fire. The interaction between zones is carried out in such a way as to maintain the observance of fundamental conservation rules (say energy and mass). These calculations can be performed in a matter of minutes even on moderate sized computers. A number of proprietary zone modelling packages are available, for instance: The Harvard Model[13]; The National Bureau of Standards Models[14] (NBS) and the Dayton Aircraft Cabin Fire Model[15] (DACFIR). These packages have been applied to a number of fire in enclosure simulations. DACFIR uses the zone modelling approach to describe the cabin atmosphere in wide body aircraft subject to an internal fire. The cabin environment is divided into an upper layer containing hot combustion products and a lower layer comprising cold uncontaminated air. The model generates a single value of fire characteristics such as air temperature and density, and smoke concentration for each zone. But the representation of these zones by single values is inaccurate because full-scale experiments have revealed that temperature and smoke concentration gradients can exist within each broad zone[16,17]. Aircraft cabin contents, such as passenger seats, are excluded from the DACFIR atmosphere model. Numerical experiments[18,19,20] suggest that in some fire scenarios, obstacles influence the manner in which fire hazards are distributed within the cabin by changing the air circulation currents.

Although research groups in the USA and UK are continuing to refine and develop zone models, several weaknesses stand out. Firstly, it is not a trivial matter to formulate a zone model as the number and location of zones representing the fire situation is not always obvious. Secondly, in certain applications some doubt may be cast on the validity of the empirical expressions used to describe the physical behavior within and between zones. Thirdly, considerable effort is required to determine inherent zone related empirical constants. Finally, the underlying assumption that fires can be resolved in this manner may not always be valid. While the above points may not apply to the application of zone modelling to domestic sized rooms they become relevant when dealing with tall buildings or buildings of large area or enclosures with complex internal structures.

Field Modelling

The main strengths of the computer modelling of fire in terms of large simple zones is the technique's conceptional simplicity and its modest use of computer resources. Opposed to this are the considerable limitations the technique has exhibited. Contrasting the zone approach is the field modelling strategy. In comparison to zone models, field models appear the more versatile and involve a minimum of empiricism. However, at their current stage of development, field models cannot be used with much confidence in the combustion region or in situations involving spreading flames. Field models also require considerably more computer power than their zone modelling predecessors. The reasons for this are twofold. Firstly, at the heart of the fire simulation problem lies one of the most difficult areas in Computational Fluid Dynamics (CFD): the numerical solution of recirculating, three-dimensional turbulent buoyant fluid flow with heat and mass transfer. Secondly, the computer power required to solve these problems is considerable. Until the recent advent of fast, inexpensive hardware and powerful, robust and reliable CFD software, mathematical simulation of the fire in enclosure scenario was strictly the preserve of cruder analysis procedures or the research mathematician. Field models differ from zone models in that they employ CFD software that can describe and predict the flow of hot turbulent fire gases across a whole field of points in the enclosed compartment.

Classical fluid dynamics is concerned with the mathematical description of the physical behaviour of fluids (gases or liquids). The equations governing the behaviour of fluids have been known for over 150 years. They consist, in general, of a set of three dimensional, time dependent, non-linear partial differential equations: the Navier-Stokes equations. In their most general form, the Navier-Stokes equations cannot be solved by analytical methods.

CFD involves the practical computer solution of the Navier-Stokes equations[21]. Even with modern computers, the exact solution of the equations governing turbulent flow is not possible. The equations describing the turbulent motion and the solution procedures to solve these equations are known; however, today's computer technology cannot provide the storage capacity or the computational speed required to allow their practical solution. The problem lies in the very nature of turbulence. The physical processes which control the growth and decay of turbulent motion are occurring on scales much smaller than the overall flow scales. Eddies responsible for the decay of turbulence in a gaseous flow are typically about 0.1mm. In order to describe the flow, it is necessary to work down to these small scales. This results in tremendous storage overheads and computational speed penalties. If the CFD product is to be of any use to the engineer, the turbulent nature of the flow cannot be ignored. This problem, for the most part, has been overcome by the development of semi-empirical turbulence models[22,23,24]. These consist of differential or algebraic equations and associated constants. For most engineering applications the solution of these equations, together with the time averaged Navier-Stokes equations, are sufficient to model the behaviour of real turbulent fluids.

Many CFD products have been developed in academic and industrial research centers to simulate specific fluid flow processes. These codes do not have widespread application as they are process-specific. General-purpose CFD products must provide an environment for the simulation of complex processes involving fluid flow, heat transfer and chemical reactions. A few general-purpose CFD products have been produced and are commercially available: PHOENICS[25], marketed by CHAM Ltd; FLUENT[26], marketed by CREARE Ltd and STAR-CD, developed from TEACH[27] and marketed by Computational Dynamics Ltd. Other products such as FLOW3D[28] developed at Harwell Laboratories and CASCADE developed at Thames Polytechnic, London are not commercially available but are being used in a variety of applications.

The new field models use these CFD techniques to solve the fundamental equations of motion and conservation for the fire at discrete points in time and space. To facilitate this, the volume of the fire compartment is divided into thousands of small volumes or computational cells. The appropriate number is dependent upon the type of fire enclosure, the order of accuracy required and, ultimately, the speed of the computer and the size of its memory. A small room may require around 2000 cells, while the interior of a large passenger aircraft requires in excess of 30,000. Using CFD products such as PHOENICS or FLUENT, the equations describing the fire system are solved simultaneously in each cell to obtain the various parameters of interest such as temperature, pressure, gas velocities, smoke concentration, etc. Thus, the model can display quantitative differences in the physical parameters throughout the computational grid. This approach requires an enormous number of calculations to be performed, thereby necessitating considerably more computer muscle than the cruder zone model alternative. Using 30,000 cells in a simulation of a fire on board a large passenger aircraft would require some tens of hours of computer time on a VAX 11/780 say. The greater sophistication and minimal use of empiricism found in the field models, however, makes them a more generally valid and versatile tool. In most applications, the greater accuracy of field models is being used to reveal the possible spread of heat and smoke in fire enclosures. When this is so, the field modeler is not concerned with the details of the combustion process and so treats the fire simply as a prescribed source of heat and smoke[18,19,20]. When the model needs to take into account the chemicals released by the fire or, in situations where it is necessary to investigate how conditions in the enclosure affect the combustion process, a more detailed combustion model must be implemented.

The combustion process is extremely complex. The change from reactants to final products includes many intermediate reactions involving the formation and interactions of numerous short lived species and free radicals. In most instances, these intermediate products and their rates of creation and destruction are not known. Turbulence further complicates the situation by influencing the mixing of reactants and products. Consequently, combustion is assumed to follow a global, one-step chemical reaction mechanism[29,30,31] in which fuel reacts with oxidant to give product. The rate of reaction is controlled solely by the turbulent mixing of fuel and oxidant which is determined from calculated flow properties. This approach, while only approximating the combustion phenomena, does give satisfactory results for relatively simple fuels such as CH_4 and polyurethane foam.

Field Models in Practice

While the practical use of field modelling is a relatively recent development, packages implementing this approach have been produced, for example, JASMINE[29-35], developed by the FRS and CHAM Ltd; UNDSAFE[18,19,36,37], developed by the University of Notre Dame in Indiana and SAFEAIR[20,38,39,40] and CLYTIE[41,42], which are currently under development at Thames Polytechnic in London.

UNDSAFE is a general purpose fire code. Early versions of the code were limited to two-dimensional single compartment enclosures[36,37]. A version of this code was modified to simulate the in-flight fire scenario[18], however these investigations were limited to two-dimensional studies. Satoh and Kurioshi[19] have performed a three-dimensional simulation of an aircraft cabin fire using a suitable modified version of the UNDSAFE code. This study however, lacked an accurate description of the aircraft cabin geometry.

SAFEAIR makes use of the CFD package PHOENICS. It is capable of simulating the spread of smoke and heat in three dimensions within a burning aircraft fuselage. One of the criticisms levelled against field models is that their application is limited to cases involving relatively simple rectangular or cylindrical fire enclosures. SAFEAIR overcomes this restriction by using specially distorted grids known as Body Fitted Co-ordinates (BFC)[43,44], see Figure 1. Through their use, the code is able to construct relatively realistic aircraft shapes as well as taking into account the contents of the cabin such as passenger seats, hat racks, overhead lockers, cabin dividers. BFC grids can be imagined as regular grids which have been squeezed, stretched, bent and twisted into the desired shape. Computational cells within the solution mesh retain their six faces, and individual cells, originally in contact remain so.

Thus far SAFEAIR has been used in two types of application. The first was an attempt to reproduce the full-scale experimental results from fire tests conducted by NASA, in 1982, in an empty BOEING-737 fuselage at the Johnson Space Center in Texas[7] (see Figure 2). Temperatures generated by SAFEAIR agreed

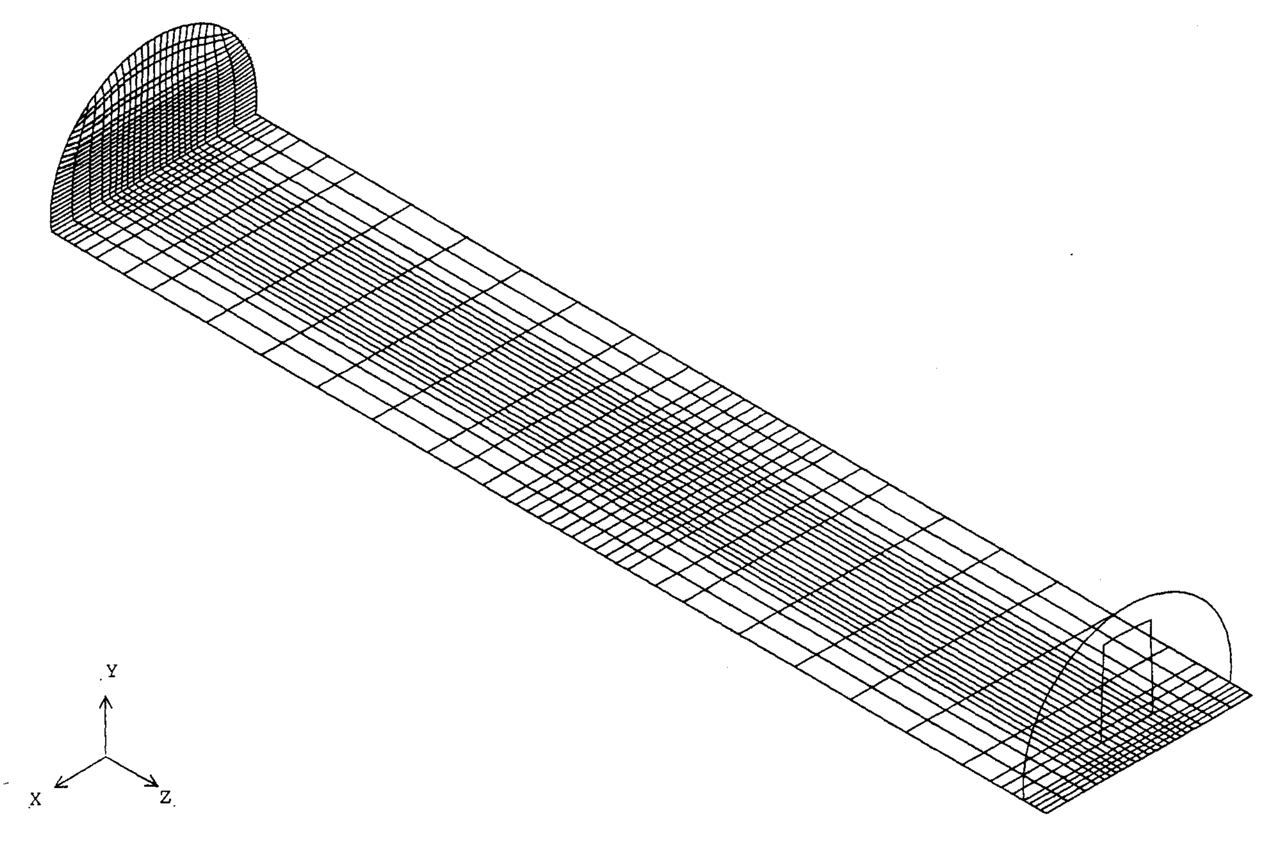


Figure 1. Body Fitted Co-ordinate Mesh. The BFC mesh depicted is used in the SAFEAIR code to represent the aircraft fuselage geometry

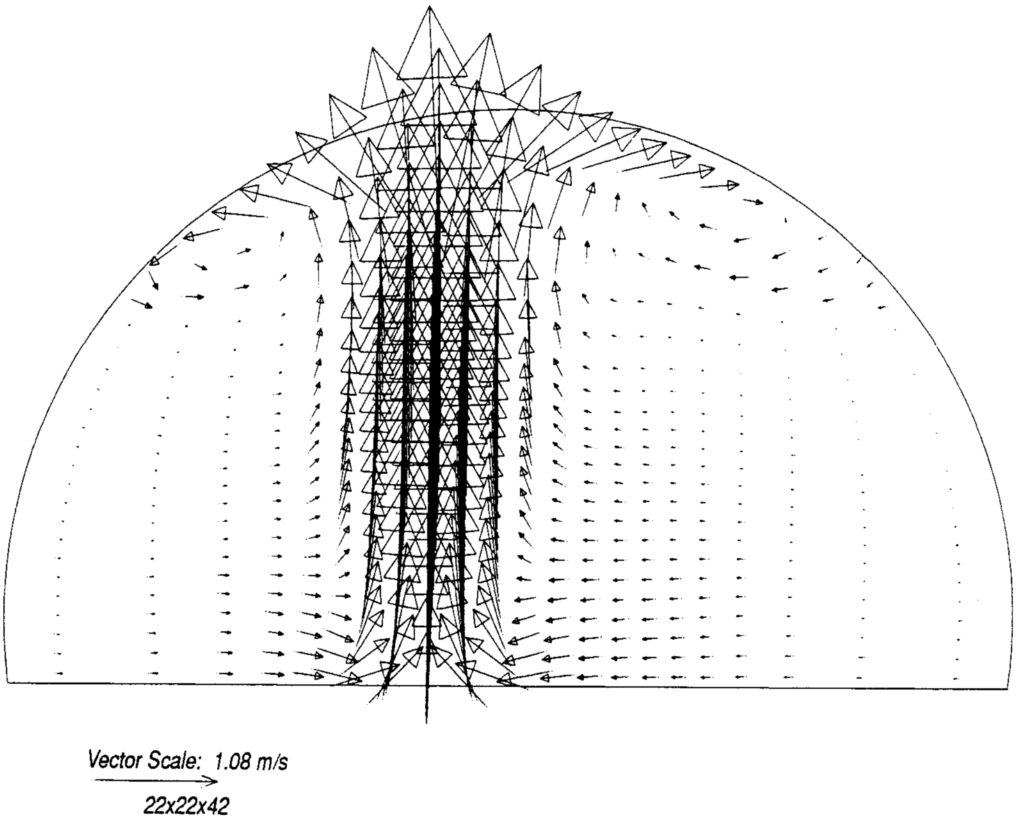


Figure 2. Fire Plume. SAFEAIR generated velocity field in a cylindrical cross-section passing through a 50.7kW heat source showing entrainment of air, rising thermal plume, impingement of plume at ceiling and the resulting re-circulation regions.

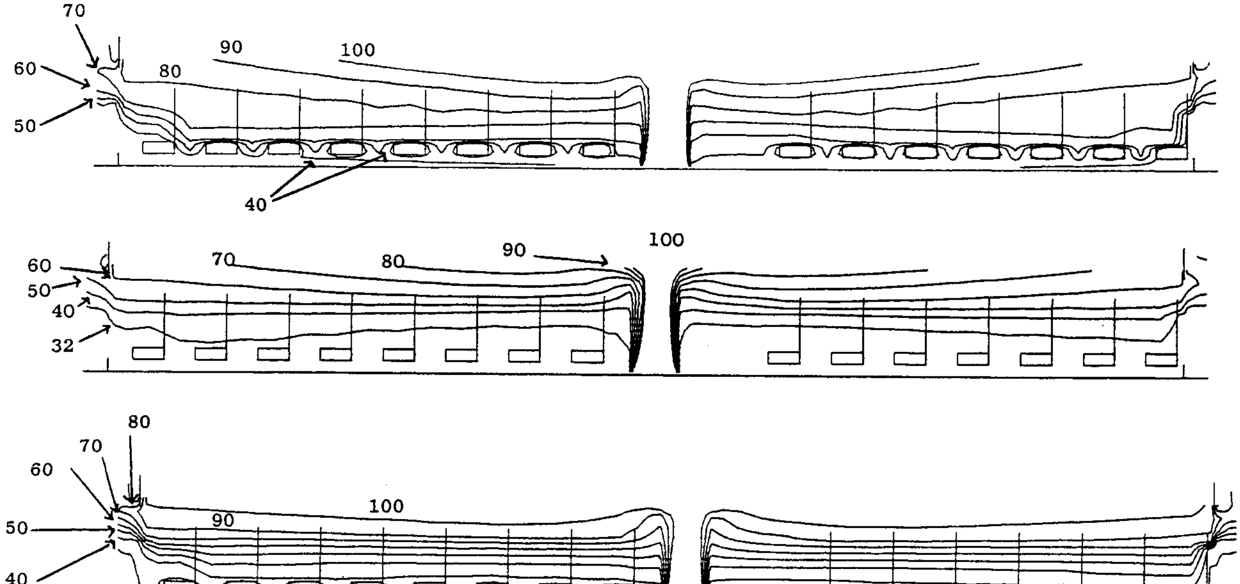
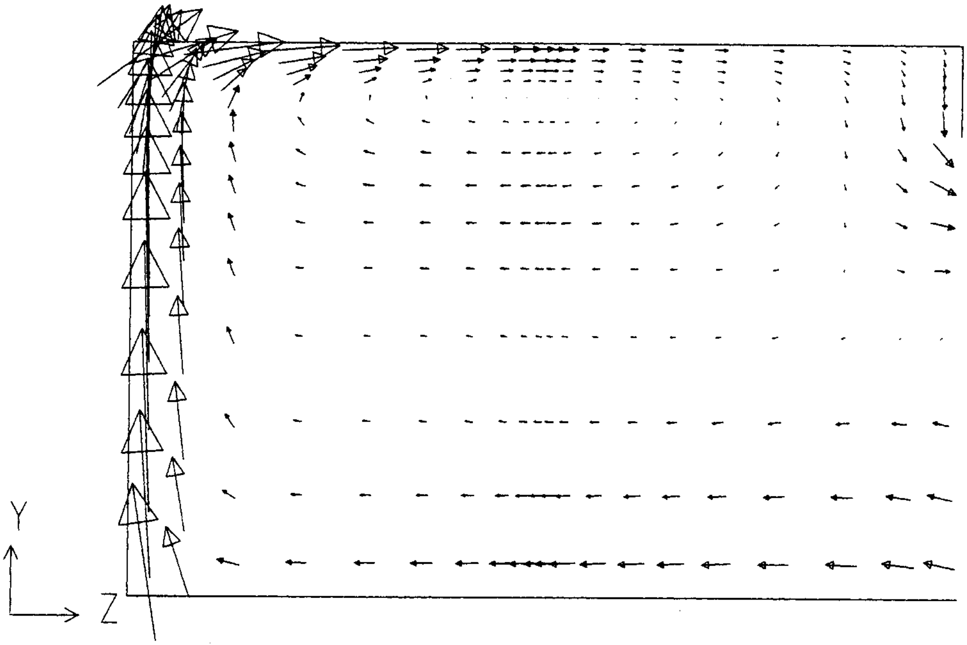
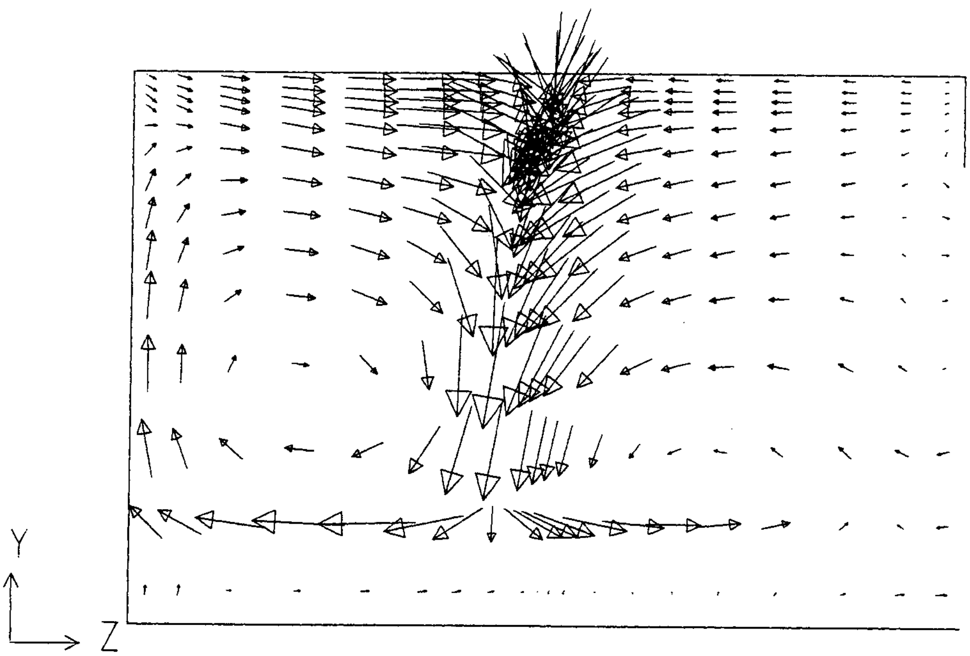


Figure 3. Aircraft Cabin Fire. SAFEAIR generated temperature coutours (degrees C) along the length of an aircraft fuselage passing through a centrally located 50.7 kW heat source. The series depicts three different venting scenarios: (a) no venting, (b) forward forced venting (cool air enters the cabin through ceiling vents and warm air is extracted from floor vents, and (c) reverse venting (i.e., opposite to case (b).)

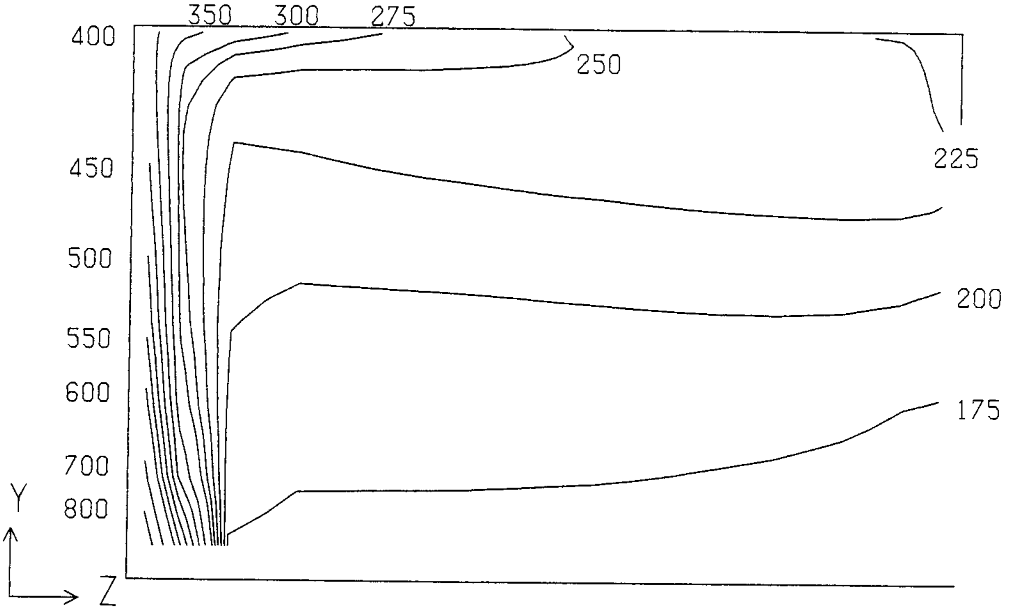


Office Fire Scenario
Figure 4 (a)
Vector scale: 2.87 m/s

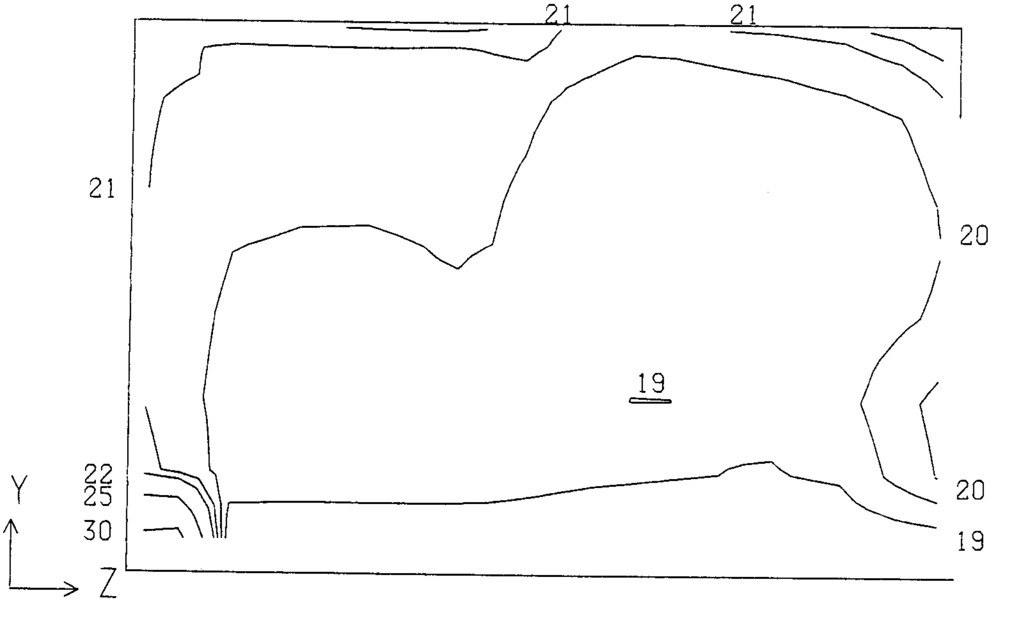


Office Fire Scenario
Figure 4 (b)
Vector scale: 2.87 m/s

Figure 4. Sprinkler-fire interaction. CLYTIE simulation of a 30 kW fire in a room of dimensions 3.66m x 2.44m x 2.44m. These figures represent longitudinal sections passing through the heat source (located near the rear wall), the sprinkler (centrally located on the ceiling and the open doorway. Figures (a) and (b) represent velocity fields before and after sprinkler activation respectively while (c) and (d) represent temperature contours (degrees C).



Office Fire Scenario
Figure 4 (c)



Office Fire Scenario
Figure 4 (d)

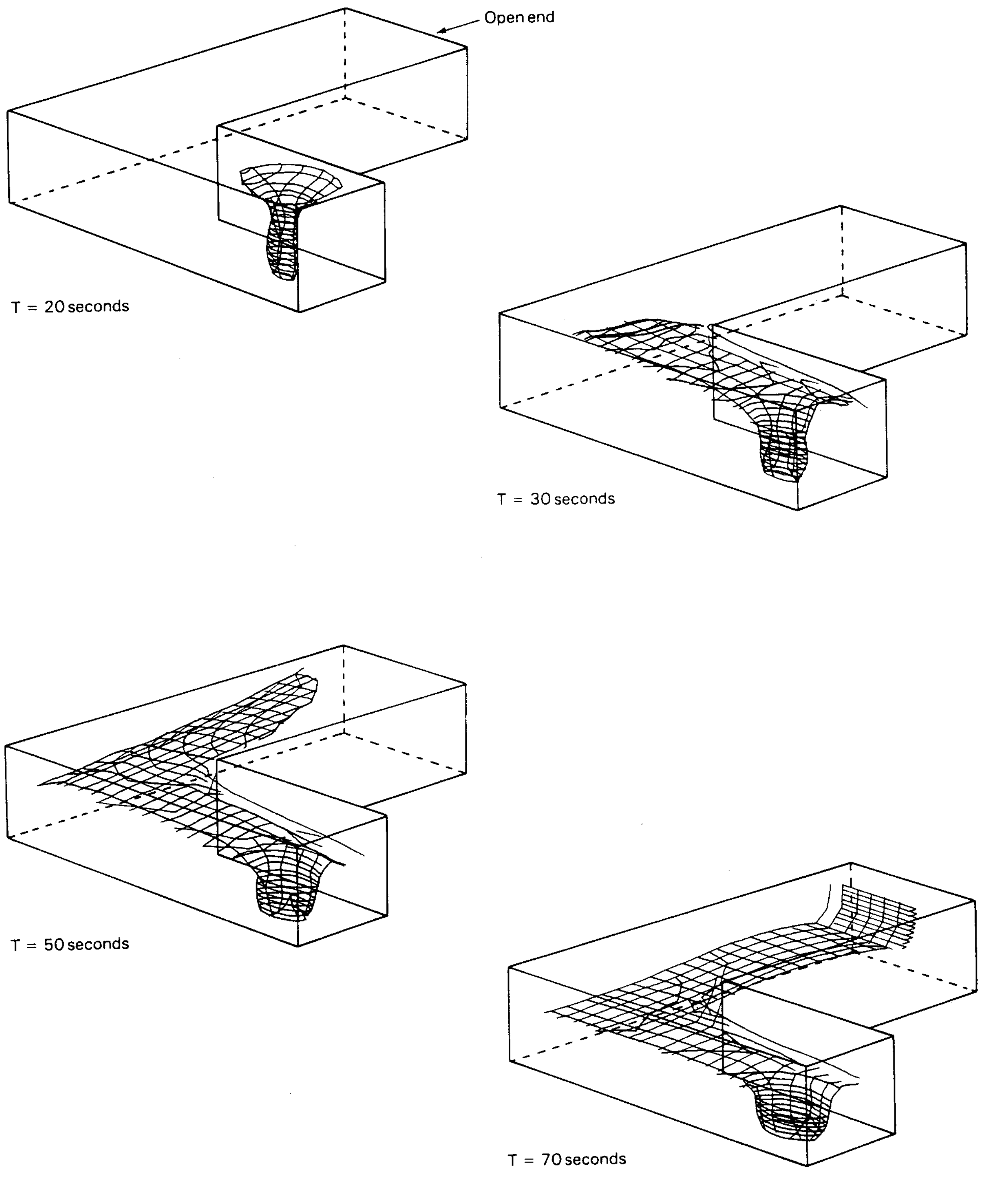


Figure 5. Shopping mall fire. JASMINE generated shopping mall fire simulation. The figures show the development of the 100° C rise temperature surface with time. (CROWN COPYRIGHT)

reasonably well with the experimental values throughout the cabin[38,39]. Maximum deviations of about 15% in the far field were observed when comparing numerical and experimental data. These calculations involved 20,328 computational cells and required in excess of 40 hours of CPU (NORSK ND-560, comparable to VAX11/785). Grids in excess of 20,000 cells are required if quantitative results are desired; however, as little as 4,200 cells will produce qualitative results. To establish confidence in the mathematics, physics and programming that make up the model, such validation is essential.

The second application involves using SAFEAIR as a predictive tool[20,40]. The model has been used to study the effects of various openings in the fuselage such as doors, hatches and ruptures and internal cabin partitions on the temperature distribution within the burning aircraft. Simulations have also been performed involving the interaction of the aircraft's air conditioning system with cabin obstacles such as seats, ceiling panels and overhead stowage bins in the event of an onboard fire. In certain fire situations, SAFEAIR suggests that the aircraft's ventilation system has a major effect on the temperature distribution within the burning fuselage. With the system extracting hot air from the floor vents and injecting cold air from the ceiling vents, as is found in most commercial passenger aircraft, temperatures in the vicinity of the seat bases increase by 20°C over the temperature found in the non-venting case. In the reverse flow situation, predicted temperatures are just above the ambient temperatures. High up in the cabin, in the vicinity of the ceiling, temperatures are also greatly reduced in the reverse venting situation (see Figure 3). The use of this venting strategy could lead to the control of the rate of spread of fire within the cabin. Such control is particularly pertinent to the in-flight fire scenario.

CLYTIE[41,42] uses the field modelling approach to simulate the activation and interaction of a sprinkler device within a three dimensional fire enclosure. Early results are displayed in Figure 4. Like SAFEAIR, CLYTIE utilizes the CFD product PHOENICS. The sprinkler-fire problem is considerably more complicated than the straight forward fire in enclosure situation. There are now two physical phases which must be incorporated into the overall mathematical description. These are the gas phase, involving the general fluid circulation of the hot combustion products and the liquid phase, representing the water droplets which have been injected into the fire compartment and evaporate. CLYTIE will address such questions as, would the sprinkler have been activated earlier had it been placed at a different location: How many sprinklers are necessary to extinguish a given fire? How long will it take to extinguish a given fire? What is the optimum average droplet size to extinguish the fire? This last question is of considerable importance to the design of fire fighting strategies. If the sprinkler is to be used to extinguish the fire then large drops are required to penetrate the rising fire plume and attack the fire source. However, if it is desired to cool the atmosphere and surrounding combustibles in order to prevent the further spread of the fire, a fine spray of water is required.

The FRS-CHAM model, JASMINE[29-35], developed over a period of ten years, is one of the most sophisticated and widely used field modelling tools available. Like the previous two codes, it uses the CFD package PHOENICS to solve the fluid flow equations. It has been used in a wide range of applications including the study of fires in domestic rooms, hospital wards, road tunnels, airport terminal buildings and large sports stadia and shopping malls. Large open single storey shopping malls are becoming increasingly popular. JASMINE[33] has been used to examine the manner in which a smoke layer produced by a fire in such a structure grows and spreads (see Figure 5). JASMINE reveals how air circulates in from the open entrance, under the smoke layer, to feed oxygen to the fire. The model also shows how the smoke layer deepens as the fire evolves. This knowledge is of great importance to fire fighting and rescue operations. One of the most important features of the FRS code is that it has been validated against a number of experimental fire tests. This means that a fair degree of confidence can be placed in predictive simulations performed using the code.

Ove Arup and Partners, the large group of consulting engineers, have made use of JASMINE[45], and, using PHOENICS have developed their own code for use in the analysis of smoke movement in large buildings. The field modelling approach is particularly useful in predicting the movement of smoke in non-standard buildings for which empirical data is non-existent. Field modelling has been used to study the movement of smoke in large atria structures such as the 74 metre high atrium of the new Lloyds of London building. These large cavernous buildings pose many questions to fire safety engineers and certification authorities. Given a ground floor fire scenario, one of their major concerns is the speed at which smoke may envelop upper open regions of the building.

A recent application of JASMINE[35] by the FRS and CHAM has been the simulation of fire in a large air-supported structure, see Figure 6. The building is a 1/6th scale model of a 60,000 seat sports stadium which Shimizu Construction Ltd of Japan are planning to construct. The model sports stadium, covered by an air supported dome, measures 34m by 28m and at its apex is 11.6m high.

The Shimizu corporation performed a series of fire tests within the structure. Results from JASMINE simulations reproduce the qualitative nature of the experimental data. In the far field, numerical and experimental results agree to within 15%. However, in the flaming region results differ greatly. This is due, in part, to the coarse nature of the solution mesh in this region (only 12 or so cells). These calculations demonstrate that in a building of this type the familiar well mixed hot upper layer does not form. This is primarily due to a strong radial jet emanating from the point of impact on the plume with the ceiling. This successful validation is of considerable importance as it extends the use of JASMINE and field modelling to fire predictions in large scale enclosures.

FUTURE DEVELOPMENTS

Mathematical field models have a demonstrated ability to simulate the actual spread of smoke and heat within a variety of fire enclosures. The technique is beginning to be used by architects and engineers in the early stages of building design to examine the structure's response to fire and associated fire hazards. In this way, possible life threatening features of building design may be eliminated before construction. The technique may also be applied to existing structures in order to find ways of reducing the threat to life in the event of fire.

Current research is concerned with increasing the efficiency of the numerical procedures which lie at the heart of field models. This is essential to reduce the high overheads incurred in using

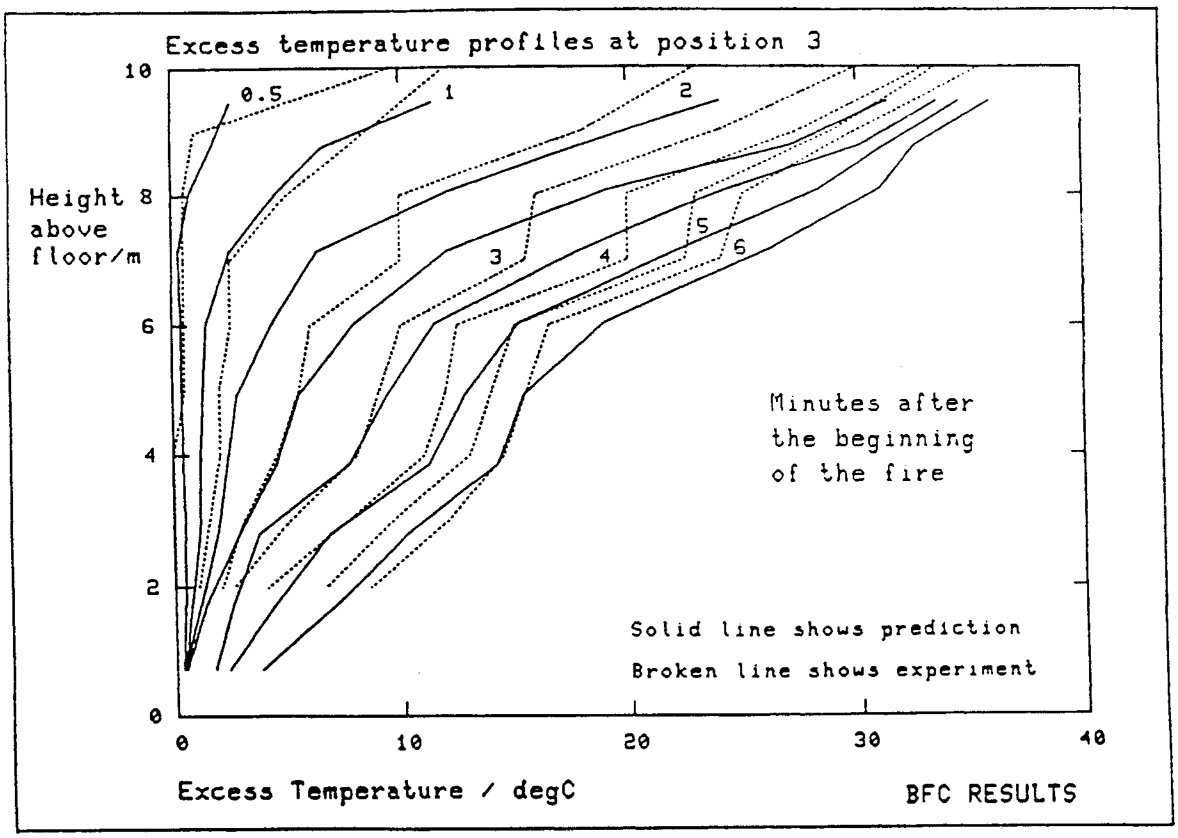


Figure 6. Sports Stadium Fire. Comparison of JASMINE generated and experimental temperature profiles during the first 6 minutes of the fire located 11m from the fire.

field modelling in realistic fire simulations. In addition to this, suitable "expert system" pre- and post-processors must be developed which enable the non-specialist architect or engineer to easily use the product and interpret the results produced by it. There is also considerable activity in the field of fire science. At present, field models cannot be used with a great deal of confidence in the immediate vicinity of the fire source. This is due to our lack of detailed understanding of the chemical and physical processes involved in: the flaming region of fires, the combustion of realistic fuels and the spread of flame over solid combustible surfaces. Current research is directed towards a better understanding of these processes.

Concluding Remarks

The continued enhancement of existing models will increase their applicability to problems that have hitherto been impossible. With this further development, field modelling should become an integral part of the design processes of all populated enclosures.

References

1. Thomas P.H., "Some Aspects of the Growth and Spread of Fire in the Open," *Forestry*, **40** (2), 1967, pp 139-164.
2. Thomas, P.H., "Rates of Spread of Some Wind-driven Fires", *Forestry*, **XLIV** (2), 1971.
3. Thomas, P.H., "Studies of Fires in Buildings Using Models, Part 1, Experiments in Ignition and Fires in Rooms", *Research*, **13**, 1960, pp 69-77.
4. Smith, P.G., "Bed assembly tests in a simulated 6-bed ward", Fire Research Station Report CR29/85, 1985.
5. Fiezlmayer, A.H., "Fire experiments in a tunnel, Ministry for Buildings and Technology", **50**, Vienna, 1976.
6. Hill, R., Borino, P.M. and Johnson, G.R., "Aircraft cabin compartmentation concepts for improving post-crash safety", Report FAA-NA-76-12, 1976.
7. Kuminecz, J.F. and Brickner, R.W., "Full-scale flammability data for validation of aircraft fire mathematical models", NASA Tech Mem 58244, 1982.
8. Eklund, T.I. and Sarkos, C.P., "The thermal impact of external pool fires on aircraft fuselages", *J Fire and Flammability*, **11**, 1980.
9. Ramachandran, G., CIB Workshop on Mathematical Modelling of Fire Growth, Paris, July, 1981.
10. Mandelbrot, B., "Random walks, Fire Damage Amount and other Paretian Risk Phenomena", *Operations Soc America*, **12**, 1968, 582.
11. Ramachandran, G., "Statistical Methods in Risk Evaluation", *Fire Safety Journal*, **2**, 1979, 125.
12. Thomas, P.H., Hinkley, P.L., Theobald, C.R., and Simms, D.L., "Investigations into the flow of hot gases in roof venting", *Fire Research Tech Paper No 7*, HMSO, 1963.
13. Mitler, H.E., "The Physical basis for the Harvard Computer Code, Home Fire Project", Tech Report No. 34, Harvard University, 1978.
14. Zuboski, E.E., Kubota, T., "A computer model for fluid dynamic aspects of a transient fire in a two-room structure", DOC NBS Grant No 5-9004, Cal. Inst. Technol., Pasadena, CA.
15. Reeves, J.B. and MacArthur, C.D., "Dayton Aircraft Cabin Fire Model, Vol 1, Basic Mathematical Model", FAA-RD-76-120, I, 1976.
16. Sankos, C.P., Hill, R.G. and Howell, W.D., "The development and application of a full-scale wide body test article to study the behaviour of interior materials during a post-crash fuel fire", *J Fire and Flammability*, **13**, 1982, 172.
17. Galea, E.R. and Markatos, N.C., "A review of mathematical modelling of aircraft cabin fires", *App Math Modelling*, **11** (3), 1987, 162.
18. Yang, K.T., Lloyd, J.R., Kanury, A.M. and Satoh, K, "Modelling of turbulent buoyant flows in aircraft cabins", *Combustion Sci Tech*, **39**, 1984, 107.
19. Satoh, K. and Kurioshi, T., "Three-dimensional numerical simulations of fires in aircraft passenger compartments", *24th Jap Aviation Symp*, 2C-8, 1.
20. Galea, E.R. and Markatos, N.C., "Aircraft Cabin Fires: A Numerical Simulation", 12th IMACS World Congress, Paris, 1988.
21. Patankar, S.V., *Numerical Heat Transfer and Fluid Flow*, McGraw Hill, NY, 1980.
22. Launder, B.E. and Spalding, D.B., *Mathematical Models of Turbulence*, Academic Press, London, 1972.
23. Spalding, D.B., *Turbulence Models*, Imperial College, London, Report HTS/76/17, 1976.
24. Launder, B.E. and Spalding, D.B., "The numerical computation of turbulent flows", *Comp Methods Appl Mech Engrg*, **3**, 1974, 269.
25. Spalding, D.B., "A general purpose computer program for multi-dimensional one- and two-phase flow", *Mathematics and Computers in Simulations*, North Holland (IMACS), Vol XXIII, 1981, 267.
26. Hutchings, B., "Solution of Natural Convection Problems using Fluent", *Fluent Users Newsletter*, **1** (1), 1986, 6.

27. Gosman, A.D., Launder, B.E., Reece, G.J., *Computer Aided Engineering, Heat Transfer and Fluid Flow*, John Wiley and Sons, New York, 1985.

28. Burns, A.D., Ingram, D., Jones, I.P., Knightley, J.R., Lo, S. and Wilkes, N.S., "FLOW3D: The development and applications of Rebase", Harwell Report AERE/R/12693, 1987.

29. Kumar, S., Hoffman, N. and Cox, G., "Some validation of Jasmine for fires in hospital wards", *Numerical Simulation of fluid flow and Heat/Mass Transfer Processes*, Springer-Verlag, Berlin, 1986, 159.

30. Markatos, N.C. and Pericleous, K.A., "An investigation of three-dimensional fires in enclosures", *Revue Generale de Thermique*, **266**, 1984, 67.

31. Kumar, S. and Cox, G., "Mathematical modelling of fires in road tunnels", *5th Int Conf on Aerodynamics and Ventilation of Vehicle Tunnels*, 1985, Lille, 61.

32. Markatos, N.C. and Cox, G., "Hydrodynamics and heat transfer in enclosures", *Physico-Chem Hydrody*, **5** (1), 1984, 53.

33. Markatos, N.C., Malin, M.R. and Cox, G., "Mathematical modelling of buoyancy induced smoke flow in enclosures", *Int J Heat Mass Transfer*, **25** (1), 1982, 63.

34. Cox, G. and Kumar, S., "The mathematical modelling of fire in forced ventilated enclosures", *18th DOE Nuclear Airborne Waste Manage and Air Cleaning Conf*, US Dept of Energy Conf 840806, 1985, 629.

35. Pericleous, K.A., Worthington, D.R.E. and Cox, G, "The field modelling of fire in an Air-supported Structure", *2nd Int Symp on Fire Safety Science*, Tokyo, 13-17 Jan 1988.

36. Yang, Y.T. and Chang, L.C., "UNDSAFE-I: a computer code for buoyant flow in an enclosure", NBS-GCR-77-84, 1977.

37. Liu, V.K. and Yang, K.T., "UNDSAFE-II: a computer code for buoyant turbulent flow in an enclosure with thermal radiation", NBS-GCR-76-150, 1978.

38. Galea, E.R. and Markatos, N.C., "Prediction of fire development in aircraft", 2nd Int Phoenics Users Conference, London, Nov 1987.

39. Galea, E.R. and Markatos, N.C., "Modelling of aircraft cabin fires", *2nd Int Symp on Fire Safety Science*, Tokyo, 13-17 June 1988.

40. Galea, E.R. and Markatos, N.C., "Progress in Mathematical Modelling of Aircraft Cabin Fires", *Disaster Management*, Vol 1, 1988, 1.

41. Hoffman, N., Galea, E.R. and Markatos, N.C., "Mathematical Modelling of Fire Sprinkler Systems", To appear in *Applied Mathematical Modelling*. .

42. Hoffman, N., Galea E.R. and Markatos, N.C., "A Computer Simulation of Fire-Sprinkler Interaction: A Two-Phase Phenomena", 12th IMACS World Congress, Paris, 1988.

43. Hedberg, P.K., Rosten, H.I. and Spalding, D.B., The Phoenics Equations, CHAM/TR/99, 1986.

44. Malin, M.R., Rosten, H.I., Spalding, D.B. and Tatchell, D.G., "Applications of Phoenics to flow around ships' hulls", *2nd Int Symp Ship Viscous Resistance*, SSPA, Goteburg, Sweden, 1986, 16.

45. Waters, R., "Air and smoke movement within a large enclosure", *Numerical Simulation of fluid flow and Heat/Mass Transfer Process*, Springer-Verlag, Berlin, 1986.