USING MATHEMATICAL MODELS TO PREDICT THE DEVELOPMENT OF AIRCRAFT CABIN FIRES.

E.R.Galea and N.Hoffmann.
Fire Safety Engineering Group
University of Greenwich
Wellington Street
London SE18 6PF
U.K.

1. SUMMARY

Computer based mathematical models describing aircraft fire have a role to play in the design and development of safer aircraft, in the implementation of safer and more rigorous certification criteria and in post mortuum accident investigation. As the cost involved in performing large-scale fire experiments for the next generation 'Ultra High Capacity Aircraft' (UHCA) are expected to be prohibitively high, the development and use of these modelling tools may become essential if these aircraft are to prove a safe and viable reality. By describing the present capabilities and limitations of aircraft fire models, this paper will examine the future development of these models in the areas of large scale applications through parallel computing, combustion modelling and extinguishment modelling.

2. INTRODUCTION

The mathematical simulation of fire has a wide, and as yet largely untapped, scope of application within the aviation industry. The function of mathematical models is to provide insight into complex behaviour by enabling designers, legislators and accident investigators, to ask 'what if' questions.

Fire models could be used to determine the impact of the spread of fire hazards such as smoke, heat and toxic gases resulting from an accident and hence predict the development of life threatening conditions within the cabin. Fire models also have application in the development of fire protection and fighting strategies such as the development of water mist systems for aircraft. Validated fire models have the potential to be used by:

Aircraft Designers, to assess the impact of new aircraft cabin layouts on the spread of fire hazards such as smoke under various fire scenarios. Fire models could be employed as design aids for the next generation UHCA, bringing fire considerations into the early stages of aircraft design,

Accident Investigators to determine the impact of the spread of fire hazards such as smoke, heat and toxic gases resulting from an accident and hence predict the development of life threatening conditions within the cabin; and finally,

Legislators, to assess the suitability of new designs and fire protection and fighting devices such as water misting systems.

3. FIRE FIELD MODELS, WHAT ARE THEY?

Fire field modelling [1] has been a reality for twenty years, however its recent success in uncovering details of the fire mechanism responsible for the Kings Cross tragedy [2] highlight its value as a fire analysis tool.

At the heart of the fire field 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. Field models differ from their simpler zone model [1] counterparts 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.

The equations which describe a field model consist, in general, of a set of three dimensional, time dependent, non-linear partial differential equations: the Navier-Stokes equations. These are essentially the same set of equations that aircraft designers use to design aerodynamic shapes such as wings. Fire field models employ CFD software 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 5000 cells, while the interior of a small passenger aircraft requires in excess of 50,000.

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. Using a three-dimensional framework of Body Fitted Co-ordinates (BFC), it is possible to construct realistically shaped fire enclosures. These could be as different as a spacious populated enclosure such as an aircraft cabin [3,4] or the confined environment of a cable duct.

4. THE APPLICATION OF FIRE FIELD MODELS TO AIRCRAFT FIRE SCENARIOS.

Since 1985 the Fire Safety Engineering Group (FSEG) of the University of Greenwich (UOG) has been developing aircraft cabin fire field models [3-8]. Over the years this work has been supported by various groups, including the U.K. Civil Aviation Authority (CAA), the U.K. Engineering and Physical Sciences Research Council (EPSRC) and British Aerospace Plc.

Field models with a flow domain describing the internal passenger space and the surrounding external region of the fuselage could be used to determine aircraft venting performance, both natural and forced. This would be useful for evaluating the impact of the spread of fire hazards such as smoke, heat and toxic gases resulting from either an in-flight or post-crash fire. The technique can also be used as a tool to aid in the design of cabin

---

Presented at PEP 88th Meeting on Aircraft Fire Safety. Dresden, Germany, 14-18 Oct. 1996.

ventilation systems in order to determine optimal levels of ventilation performance for maximum passenger comfort.

External post-crash fuel fires could be examined and their interaction with the prevailing environmental conditions and fuselage orientation determined. Once the external fire has gained access to the internal regions of the aircraft, either through, an existing rupture, open doorway or 'burn-through', field models can be used to determine the development of the hazardous cabin fire environment.

One of the earliest aircraft cabin fire calculations produced by the FSEG compared model predictions with measured data derived from a series of experiments produced by the Johnson Space Centre [9]. These experiments were conducted in an empty B737 fuselage, with a fire source consisting of a fuel pan containing 4.5 l of Jet A-1 fuel located on the cabin floor just off the cabin centre. The B-737 had the forward and aft sections removed and replaced with bulkheads containing doors.

This simple geometry was modelled accurately in three-dimensions using a BFC grid consisting of 20,328 computational cells and the fire was represented by a simple constant heat source [3] representing a non-spreading fire. The results demonstrated that such a model was capable of predicting the observed trends in temperature distribution throughout the aircraft.

Using the B-737 geometry and the previous fire specification, several other hypothetical fire scenarios were simulated representing various 'what if' cases. These consisted of investigating the effect of various fuselage openings [8] and cabin partitioning [3] on the temperature distribution within an empty burning cabin. Another series of fire simulations involved the B-737 fuselage fitted with seats and overhead stowage bins and considered the effect of the cabin ventilation system on the developing fire atmosphere [3,8].

The cabin openings consisted of combinations of external doors and ceiling apertures. The ceiling aperture was intended to simulate a rupture in the fuselage. Four cases were examined, these consisted of Case A: aft left door open, Case B: forward and aft left doors open, Case C: left and right aft doors open and Case D: left aft door and the ceiling above the door are open.

The model results clearly showed that the temperature distribution within the fuselage was strongly affected by the nature of the fuselage openings. The highest temperatures were found in Case A (single side opening). This was consistently true, throughout the length of the fuselage and in the vicinity of both the floor and ceiling. In this case temperatures near the floor were typically 80°C while temperatures in the ceiling region were about 115°C.

Of the four cases examined, Case D (ceiling and side opening) generated minimum temperatures in the ceiling region. In the aft section of the cabin temperatures were typically 80°C while in the forward section temperatures were in the vicinity of 95°C. In this case, the model predicted that relatively cool air was entrained into the cabin throughout the open area of the left door while hot ceiling gases were vented out through the ceiling opening. In the lower regions of the cabin, Case B (forward and aft left doors), generated the lowest temperatures. In this scenario, temperatures near the floor were typically 60°C.

These simulations suggest that fuselage openings have a profound affect on the developing cabin atmosphere and hence on the threat to human life. Simulations of this type may be useful in suggesting fire fighting strategies and in investigating aircraft accidents involving fire [8].

The effects of cabin compartmentation on the temperature distribution throughout the cabin were also investigated [3]. The cabin partition consisted of a bulkhead containing an open doorway. The bulkhead was located in the forward section of the cabin 1.5m from the fire source. This effectively partitions the cabin into two sections, the forward section containing the fire and the aft section.

Model results suggested that the cabin partition offered some degree of protection to the aft section of the cabin. With the bulkhead present, temperatures in the aft section of the cabin were on average 15% lower than the corresponding temperatures in situations without the cabin partition. Conversely, temperatures on the fire side of the cabin are somewhat higher in the partitioned cabin compared with temperatures in the unpartitioned cabin.

A detailed investigation of the model predictions revealed that the ceiling soffit intercepts the hot ceiling jet. Some of the hot gases are diverted under the soffit, while immediately behind on the fire side of the cabin, a small region of recirculating air develops. These results suggest that cabin compartmentation may offer passengers some protection from elevated temperatures in the event of fire. There is some experimental evidence [10] to support these claims.

With the B-737 cabin fitted with seats, ceiling panels and overhead stowage bins the effect of the aircraft's air-conditioning system on the temperature distribution within the burning fuselage was examined [3,8]. Three venting scenarios were investigated. The first case, case A, involved no forced ventilation. In the second case, case B, fresh air is injected from the ceiling vents while hot air is sucked out from the floor vents. Case B is intended to simulate the operation of the environmental control systems found in most commercial aircraft. In the final case, Case C, the venting in case B was reversed so that air was injected at the floor vents and exhausted through the ceiling vents.

The usefulness of reverse venting in reducing temperatures and smoke concentrations near the floor in building fire scenarios is well known and has been observed in full-scale experimental room fires [11]. The purpose of these simulations was to investigate if similar benefits could be expected in aircraft fire applications.

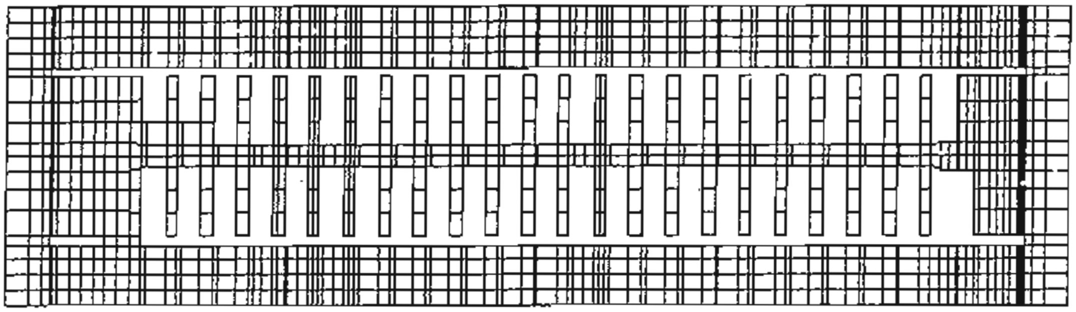
The results suggested that a reverse flow venting situation greatly reduces the temperature throughout the fuselage in the vicinity of the floor and ceiling. The use of this venting strategy could lead to the control of the rate of spread of fire and smoke within the cabin and hence prolong habitable conditions within the cabin. Such control is particularly relevant to the in-flight fire scenario. These model predictions have subsequently been reinforced by a series of FAA sponsored experiments [12].

5. CURRENT RESEARCH IN FIRE MODELLING

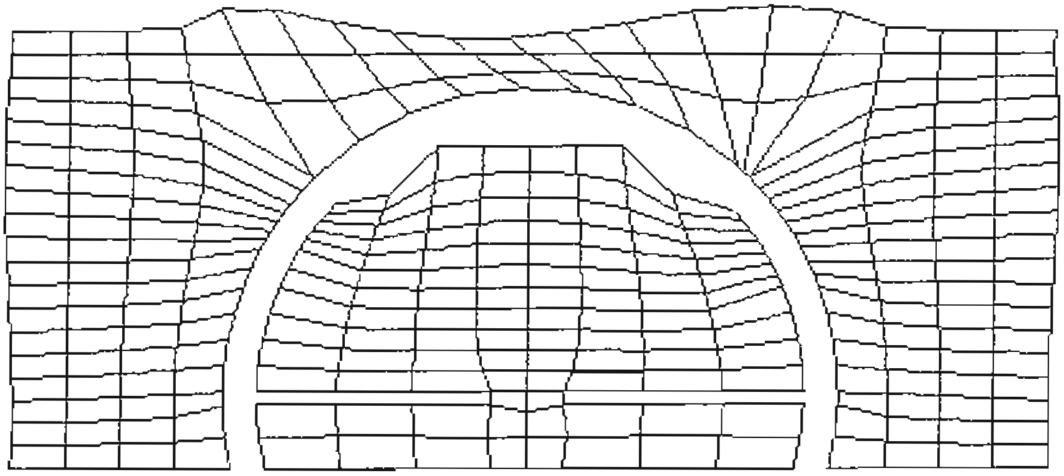
The primary application of current fire field modelling technology concerns the prediction of smoke and heat movement within fire enclosures. While the capabilities of current fire field models are considerable much research is required to widen their scope of application. Here we will examine the future development of these models in the areas of large scale applications through parallel computing, combustion modelling and extinguishment modelling.

5.1 Parallel Computing.

Field modelling requires an enormous number of calculations to be performed, thereby necessitating the need for considerable computer power. Hundreds of hours of computer time may be required to perform even the simplest of aircraft fire simulations using current generation workstations. The high computational cost associated with fire field models is being tackled through advances in parallel computing hardware and software thereby making these models more affordable and practical. The FSEG have adopted this approach and developed a parallel fire field model [4,13].



(A) HORIZONTAL VIEW



(B) CROSS-SECTIONAL VIEW.

FIGURE 1: Computational Mesh in the Boeing 737 fire simulation. Depicted are the (a) plan view and (b) cylindrical section view

The ability of fire field models to exploit parallel computing techniques will enable these models to be accurately and efficiently employed in large geometries such as B777 and A340 aircraft and their successors. Without this capability, compromises in mesh density and model complexity would be necessary in order to make simulations practical.

To demonstrate the capabilities of this system the parallel fire model has been used in two aircraft applications. The first concerns a fire scenario involving a reconstruction of the Manchester B737 aircraft fire and the second concerns a hyperthetical UHCA.

5.1.1 B737 Fire.

The first example is intended to simulate a fire similar to that which destroyed the B737 at Manchester Airport on the 22 August 1985. These simulations are fully three dimensional and transient, including a detailed description of the B737 geometry. While the aircraft was preparing for take-off at Manchester airport, an external fuel fire started and eventually gained access to the cabin interior [14]. This disaster (which claimed the lives of 55 people) posed several questions concerning the spread and behaviour of fire gases within aircraft cabins. Through the application of fire models we hope to gain insight into how fatal conditions develop within the aircraft.

As sufficient fire details are not available to accurately define the Manchester fire scenario, this simulation is not intended to reproduce the actual disaster. The simulation may be considered "Manchester like" in so far as the aircraft geometry, door opening sequence, external wind conditions

(7 knots approaching from 250°) and initial fire location are similar to the reported situation.

The B737 is a single aisle aircraft that is approximately 20.3 m in length with a maximum width and height of 3.216 m and 2.105 m respectively. It has four floor level doors, two front (L1 and R1) and two aft (L2 and R2) along with two additional over-wing (ROW and LOW) exits. Of the six exits available, four were opened and only three were used by escaping passengers. The reported opening times were, R1 70 seconds, L1 25 seconds, ROW 45 seconds, R2 0 seconds. All times refer to time after aircraft came to a stop.

Analysis of the wreckage suggests that the fire initially penetrated the outer skin of the aircraft on the left side in the vicinity of seat rows 17 to 19, below the level of the cabin floor. Having breached the outer skin the fire would have quickly gained access to the cabin.

A body-fitted co-ordinate system was used to fit the computational grid to the actual specifications of the fuselage, whereby the seats, galleys and overhead lockers were all included. A relatively coarse grid of 18x20x104 (37,440) cells was used to describe the aircraft including an extended domain outside the fuselage (see figure 1). During the transient simulation the exits were opened as reported.

There is uncertainty concerning the precise location and spread of the internal fire and its heat release rate. For the simulations presented here the fire was simulated as a volumetric heat source with a time varying heat release rate and fire area. The INTERNAL fire grows to a maximum heat release rate of 720 kW in the first 30 seconds and is initially located under seat rows 17 to 19 on the left hand side. For the first 25 seconds the fire area was 1.0 m^2 growing to 3.39 m^2 after 30 seconds

In order to investigate the impact of the external cross-wind on the internal fire environment two different scenarios were simulated; the first without and the second with a cross-wind. The simulations were performed using a parallel computer consisting of four i860 processors, resulting in the case without wind requiring 97 hours of CPU time to complete 60 seconds of simulation. The nature of the flow generated by these models is highly transient and complex. Computer generated video animation is used to aid in the interpretation of the results. This animation can be viewed over internet the [15].

For the first 25 seconds, whilst the forward exit was closed, the cross-wind did not have a significant effect on the internal flow and temperature conditions. After this time the L1 exit, which was facing the oncoming wind, was opened. This allowed the wind to enter and hence influence the interior flow and temperature conditions.

After 30 seconds, the results indicate that the wind generated internal flow opposes the flow of the hot combustion gases forward of the fire. However, aft of the fire the two flows are in similar directions creating a more prominent outward flow through the open aft exit (see figure 2) . As a result of this flow, temperatures are reduced in the front part of the cabin and increased in the aft part of the cabin (see figure 3).

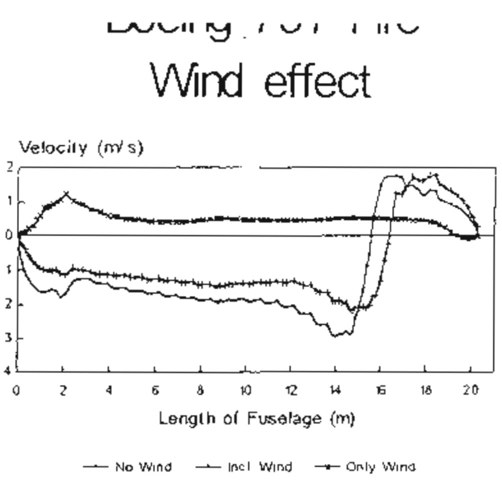


FIGURE 2: Velocity distribution along the centre of the aircraft from forward to aft 1.9m above the floor.

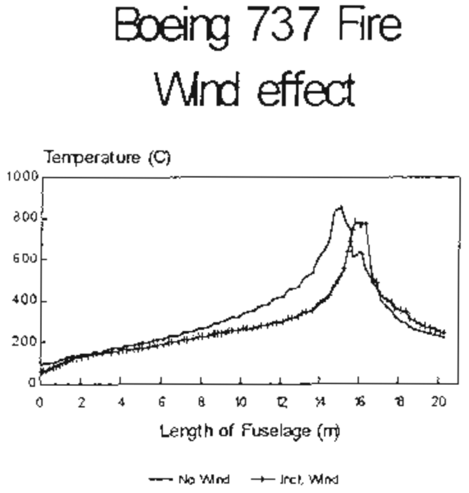


FIGURE 3: Temperature distribution along the centre of the aircraft from forward to aft 1.9m above the floor.

The peak temperature measured in the aisle is also reduced in the presence of wind. Note that at ceiling height, temperatures are in excess of 800°C in a small localised region where the plume impacts the ceiling. Pure aluminium has a melting temperature of approximately 650°C and experiments conducted for the CAA [16] suggest that a 1mm thick aluminium sheet placed in a furnace at 950°C will burn through after 80 seconds. Burnthough by the interior fire can therefore be expected at some time after 30 seconds

A well-defined temperature stratification is also observed within the cabin. While temperatures are high at head height near to the floor they are quite small. Temperature stratifications of this type have also been observed in full-scale aircraft cabin fire experiments [17,18].

The temperature stratification has a major impact on survivable conditions within the cabin. According to the Fractional Effective Dose (FED) model developed by the FAA [19], dry air temperatures of 240°C can be withstood by clothed individuals for one minute before incapacitation occurs. This figure simply relates to the temperature component of the fire atmosphere and does not take into consideration toxic gases or smoke. From figure 3 it is clear that in the case of fire without wind, this critical temperature is exceeded at head height from seat row 7 back. However, in the case involving fire and wind, the critical temperature is not exceeded until seat row 10 is reached. Close to the floor the situation is much improved. At a height of 0.5 m above the floor, the maximum aisle temperature is less than 140°C. According to the FED model, occupants can withstand this temperature for 6 minutes.

The above scenario represents the first 30 seconds of a 'Manchester type' fire situation. It is intended to continue these simulations to the point where the R1 and overwing exits are opened. It is hoped by investigating the flow that develops under these conditions we will better understand the complex interaction between aircraft, fire and wind. Further details may be found in reference 4.

5.1.2 UHCA Application

UHCA have been proposed which consist of two decks stretching along the entire length of the aircraft. In such aircraft multiple staircases linking the decks may be necessary. Aircraft designs of this magnitude arguably have more in common with hotels than aircraft. Fire models offer aircraft engineers a possible means of exploring the ramifications of fire in these unusual structures and in assessing the likely impact these fires may have on evacuation. For instance, the role of staircases in propagating smoke, heat and fire gases to regions otherwise clear of fire could be examined using fire models. The ramifications of burnthrough to other decks, cabin compartmentation and forced ventilation strategies on the associated spread of fire hazards could also be examined. Possibly of greater importance, means of preventing the spread of fire hazards may be explored using these models.

To demonstrate the application of fire models to UHCA, FSEG have performed a simulation of a fire on board a hypothetical UHCA. The portion of the aircraft simulated measured 28m in length and consisted of two decks and 260 seats arranged in a twin aisle configuration. The aircraft section contained 3 staircases with 3 pairs of doors on each deck. The fire was located on the lower deck in the vicinity of the central stair, and the doors were open during the simulation (see figure 4). Note the spread of the fire hazards to the upper deck in figure 4. The model consisted of 100,000 cells and required 14 hours of computer time on an 8 processor parallel computer.

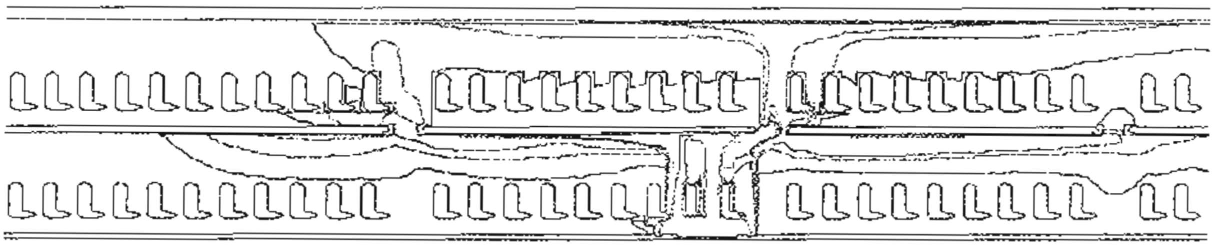


FIGURE 4: UHCA with fire on lower deck. Temperature contours predicted using FSEG fire model.

5.2 Combustion Modelling.

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, in most fire models combustion is assumed to follow a global, one-step chemical reaction mechanism [20], 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 process, does give satisfactory results for relatively simple gaseous fuels.

The prediction of flame spread over complex solid surfaces such as aircraft seats, cabin walls and floor linings is currently beyond the scope of field modelling technology and is receiving considerable interest from research groups throughout the world.

As a first approach to this problem, the FSEG is developing a simple solid fuel combustion model to be incorporated within a fire field model. The model is intended for use in engineering applications of fire field modelling and represents an extension of this technique to situations involving the combustion of solid cellulosic fuels. The model consists of a thermal pyrolysis model, a radiation model and an eddy-dissipation model for gaseous combustion. Within the model the flame spread is governed by a set of partial differential equations which express gas phase behaviour, solid phase behaviour and their interaction.

During pyrolysis, solid fuel may undergo melting, shrinking/expanding and charring. The thermal properties of the material will also vary with temperature. Several models have been developed to represent this complicated process [21,22,23]. The most complex of these models makes use of kinetic rate laws [22,23] and a large number of material properties [21]. Compared with these more sophisticated models, the thermal pyrolysis model [21] uses the relatively simple concept of the pyrolysis temperature as a first approximation. The relative simplicity of this model makes it an attractive proposition for engineering applications. This simple pyrolysis approach has been adopted here. The pyrolysis mechanism is simply described as a process in which combustible gases are given off the surface of the solid fuel at the pyrolysis temperature. While the concept of the pyrolysis temperature is questionable and scenario specific, both physical experiments [24] and theoretical analysis [25] have demonstrated that it provides a fair approximation to the pyrolysis process for various materials. In this model, combustible gases are released from the surface of the solid fuel when it is heated to its pyrolysis temperature T_p. In fires, the energy to sustain this endothermic gasification process is generally supplied by the thermal radiation emitted from the fire and hot combustion products. In addition to radiation, flame spread over the solid fuel is influenced by conduction within the fuel and so this mechanism is included within the model. Conduction allows virgin fuel not directly exposed to radiation to be preheated

The solid fuel combustion model has been demonstrated through two-dimensional simulations of flaming combustion in a room fire scenario involving a plywood ceiling. The target fuel lining the ceiling is discretised into a number of layers running parallel to the floor. Once a layer of the solid fuel is heated to the pyrolysis temperature T_p, it begins to be gasified while its temperature remains fixed at T_p. As it is being gasified, the solid fuel is consumed one layer at time. Once the lining materials have been gasified, the gaseous combustion model (eddy dissipation model) is activated to simulate the flame spread process. The radiative heat flux from the fire and hot gases and reradiated heat losses from the solid surface are calculated using the radiation model where the scattering coefficient assumes the value $0.01 m^{-1}$ [26].

The first case considers an open room scenario in which a flashover type phenomenon is predicted to occur. The second case considers a sealed room fire scenario in which the closed door is opened after some time. In this simulation a backdraft type phenomenon is predicted to occur.

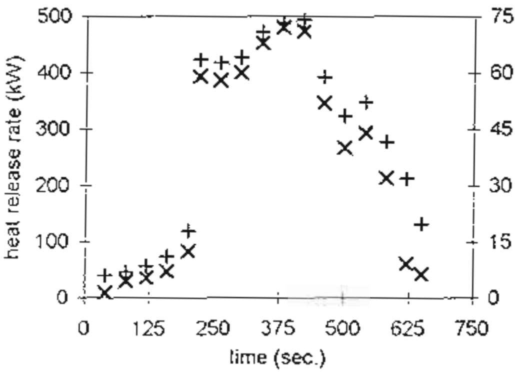


FIGURE 5: Heat release rate of gaseous combustion and burning rate of solid material in case 1. +: the heat release rate of gaseous combustion (kW); x: the burning rate of solid material (g/s).

5.2.1 Flashover

The fire development within the compartment appears to undergo a three stage development. This can most clearly be seen in figure 1 which depicts the heat release rate due to gaseous combustion and the burning rate of solid fuel within the compartment.

The curves are clearly divided into three regions representing three phases of fire development. In the first phase, which lasts for the first 200 seconds, the heat release rate of gaseous combustion in the compartment increases at a slow and fairly constant rate (see figure 6).

At about 220 seconds (see figure 5) a critical point is reached where the fire rapidly passes into the second phase of fire development and the heat release rate of gaseous combustion in the compartment undergoes a sharp increase. This rapid increase is a result of the entire combustible ceiling becoming involved in the fire.

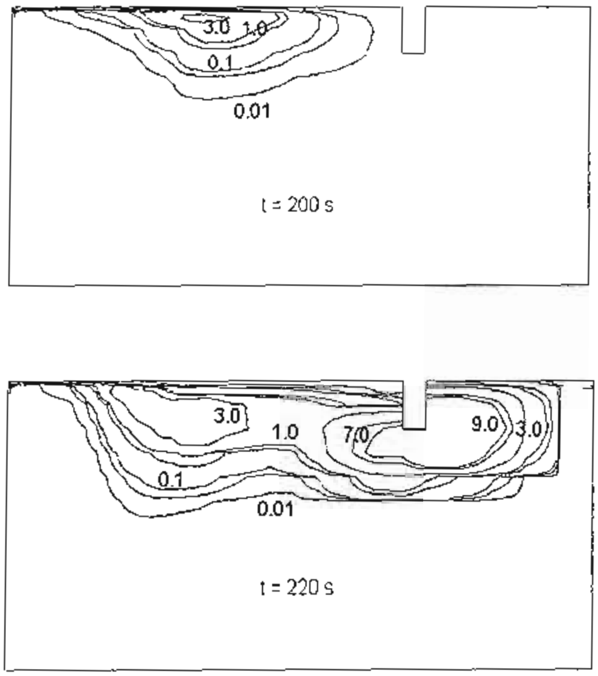


FIGURE 6: Contours of heat release rate of gaseous combustion for case 1 preflashover (t = 200s) and during flashover (t = 220s). Unit: kW.

As a result, the flame erupts out of the compartment (see figure 6), a phenomena often observed in experiments in which flashover occurs. During flashover, combustion within the gaseous phase is more pronounced and involves a greater proportion of the compartment than is observed during the preflashover stage. Over the next period of about 200 seconds the heat release rate of gaseous combustion and the burning rate reach a maximum and maintain a reasonably stable state. The fire is fully developed in this period. During this phase the gas temperature beneath the ceiling reaches a peak of about 1100K. The third phase of fire development occurs approximately after 460 seconds, where the heat release rate begins to rapidly decrease (see figure 5). During this phase all of the remaining solid fuel is consumed.

5.2.2 Backdraft

When the door to the fire compartment is closed, the initial increase in room temperature and ceiling fire spread are more rapid than those noted in the previous case. As the door is closed, there is no source of fresh air - and hence oxygen - to replenish the oxygen consumed by the combustion. In this case, while the pyrolysis process continues, combustion is incomplete and more and more unburned fuel gases accumulate within the room (see figure 7).

Compared with the previous case, instead of a sharp increase in heat release rate as the fire spreads, the heat release rate increases slowly, however there is a rapid increase in the amount of fuel accumulating within the compartment (see figure 7). After approximately 45 seconds, the heat release rate due to flaming combustion begins to decrease due to the reduction in oxygen concentration. Figure 7 suggests that even as the fire dies down, the amount of fuel accumulating in the compartment continues to increase. This suggests that the pyrolysis process continues as the hot gas mixture provides sufficient energy for the endothermic process to continue.

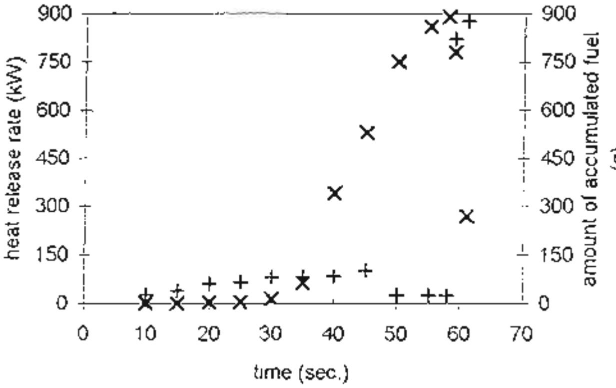


Figure 7: Heat release rate of gaseous combustion and fuel accumulation within the compartment for case 2. +: heat release rate of gaseous combustion (kW); x: amount of fuel accumulating within the compartment (g).

After 59 seconds, the door of the compartment is opened suddenly. Oxygen rich air is entrained into the room through the lower reaches of the door while the hot fuel rich gas mixture flows out the room through the upper reaches of the door, under the soffit. Almost immediately, this motion of hot fuel rich gases and cool oxygen rich air reignites the combustion process. Initially (i.e. one second after the door is opened), gaseous combustion primarily takes place in the upper layer outside the room, in the doorway and in the lower layer just inside the room. This is due to the nature of the mixing process between the hot fuel rich gases leaving the room and the fresh oxygen rich air entering the room. As a great amount of fuel has accumulated within the compartment (see figure 7), the gaseous combustion is tremendously intense. Furthermore, considerable amounts of combustible gases spill out from the top region of the doorway in a very short space of time generating a large combustion region outside the compartment i.e. the flame is seen to protrude from the compartment.

As oxygen rich air mixes with the fuel rich combustion products within the room, flaming combustion erupts through a greater proportion of the compartment in a matter of seconds. These processes result in the marked rapid drop in combustion gases noted in figure 7. This type of behaviour is similar in nature to the hazardous phenomenon known as backdraft. Further details of this work may be found in reference 26.

5.3 Extinguishment Modelling

Another area of interest is the modelling of fire suppressant systems. Such scenarios have obvious application to the development of aircraft water mist systems for use either in cabins or as a replacement for existing halon based systems in cargo holds [27,28]. Using the field modelling approach it is possible to simulate the action of water sprays in a fire compartment.

In this case there are now two interacting physical phases, the gas phase involving the general fluid circulation of the hot combustion products and the liquid phase, representing the evaporating water droplets. The numerical procedure of the fire model must be adjusted to take into account these interacting phases. This set of equations now includes the interphase processes of drag, heat and mass transfer between the liquid and gaseous phases.

One approach to the simulation of these interacting phases is the Euler-Lagrange methodology [29]. In this approach the gas phase is modelled using standard CFD techniques while the discrete phase (water droplets) are modelled using a Lagrangian particle tracking scheme. The motion and properties of individual droplets or packets of droplets are tracked either until they evaporate or come into contact with a surface. Finally, the two phases are coupled using the PSI-Cell method. In this method the particles mass, enthalpy etc are noted as it enters and leaves each cell in the computational domain. Any changes in the values of these quantities are due to gas/droplet exchange and are calculated and added to the appropriate cell in the gas phase as sources. In this manner the temperature and gas flow will effect the trajectory and evaporation rate of the water particles and the particles will react back onto the temperature and velocity field of the gases.

FSEG have developed a water spray model for use in aircraft fire applications. To demonstrate this model a conventional sprinkler system located in a long corridor was simulated. The corridor was 12m in length, 3m in height and 3m in depth. It was meshed using a uniform Cartesian mesh of 100 cells in the horizontal direction, 15 cells in the vertical direction and 15 cells in depth. A fire was arbitrarily situated in the centre of the corridor. The interaction was modelled for 120 seconds in 1 second time steps. The spray was located 5m to the right of the fire, 1m from the end of the corridor. It had a spread of 90° pointing symmetrically downwards, a flow rate of $2.4ls^{-1}$, 1000μm diameter droplets, an initial velocity of $5ms^{-1}$ and a temperature of 20°C. It was modelled using 48 trajectories in 4 rings of 12, equally spaced by angle in both the horizontal and vertical planes. Using a SUN supersparc 10, 40 MHz server, the 120 second simulation required approximately 22 hours of CPU time.

The results of the simulation have been animated and are available for study over the internet [15]. Prior to sprinkler activation, a plume of hot fire gases rises from the fire source and spreads out under the ceiling in a symmetrical manner. Prior to sprinkler activation a symmetrical flow is established and a hot ceiling layer forms which gradually deepens with time. After 10 seconds the sprinkler is activated

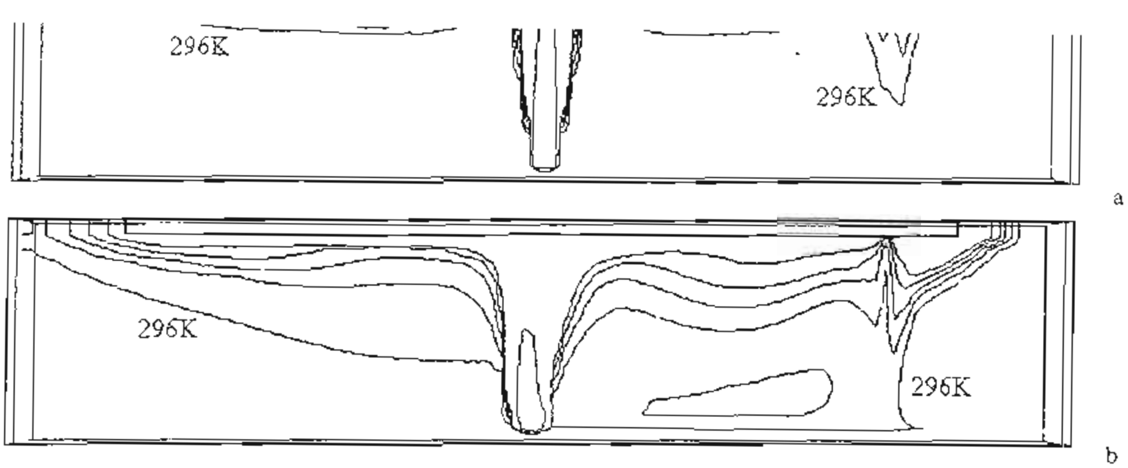


FIGURE 8: Temperature contours (K) along symmetry plane, 1 and 50 seconds after sprinkler activation. Contours start at 296 K and step in 3 K intervals.

The symmetry in the hot layer is disrupted by the sprinkler spray which induces a down draught at its location, resulting in a column of hot air being dragged to the floor (figure 8). One second after sprinkler activation, the beginnings of a descending column of hot air, dragged down by the spray, can be seen.

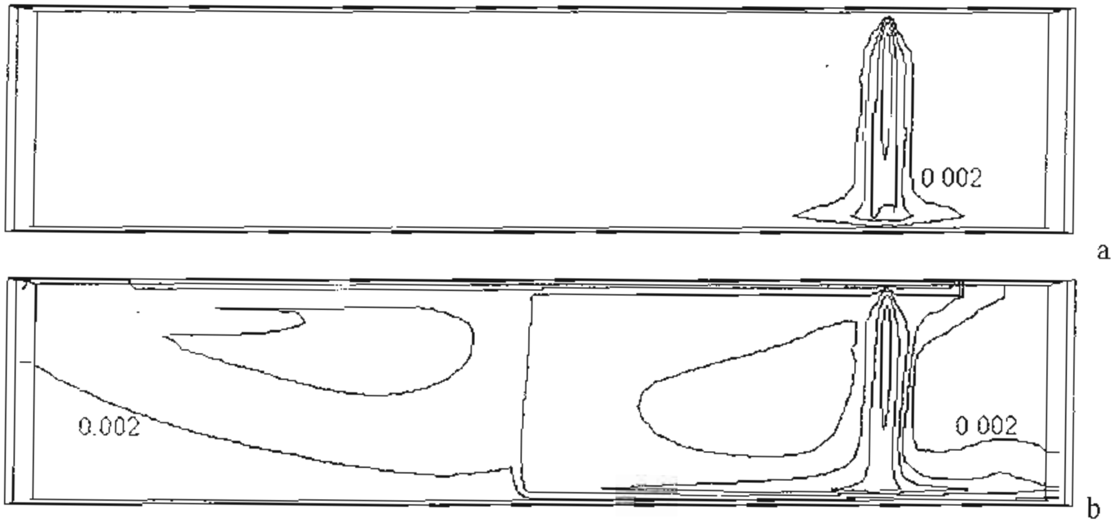


FIGURE 9: Water vapour concentration (kg water/kg of mixture) along symmetry plane, 1 and 50 seconds after sprinkler activation. Contours start at 0.002 and step in intervals of 0.002.

Figure 8 suggests that the gases in the core of the spray cone are cooled relative to the gases on the outside of the cone envelope. This suggestion is substantiated by figure 9 which depicts the water vapour concentration on the symmetry plane 1 and 50 seconds after sprinkler activation. Figure 9 reveals that there is a higher concentration of water vapour in the core of the spray cone. This higher concentration of water vapour is generated through the evaporation of the water droplets. This process requires a large quantity of heat which is extracted from the surrounding hot air. Another important observation to emerge from figure 9b is the manner in which water vapour is distributed throughout the corridor. Water vapour is present on the left side of the fire, remote from the source.

This approach has been adopted by the FSEG and forms the basis of a spray model for use in aircraft fire applications. The model includes parameters such as flow rates, droplet size, throw angle, orifice size etc. The model is being used as the basis of a European Union funded research project under framework IV known as FIREDASS. The primary purpose of the FIREDASS project is to optimise a water misting replacement for halon extinguishment systems currently used in aviation (aircraft cargo holds) and shipping (machine rooms) applications.

6. CONCLUSIONS

While still requiring further development, fire field modelling has an impressive range of capabilities to offer the aerospace industry. While existing aircraft fire field models rely on imposed fire descriptions, they can be used to describe the spread of fire hazards such as heat and smoke within the aircraft and thus reveal how potentially hazardous conditions develop.

The ability to predict flame spread over solid surfaces and the onset of flashover are two important areas in fire modelling. The model described in this paper was used to simulate the fire development within a compartment in which the ceiling - lined with plywood - was the only source of combustible fuel. In the case of the open compartment fire scenario, the model was able to qualitatively predict behaviour similar to the three stages of fire development - growth and flashover, fully developed and decay. The model was also able to predict the occurrence of a backdraft type phenomenon within a compartment which was originally closed. In this case the model predicted the initial fire growth period, the throttling back of the combustion process and the resulting deflagration when a new opening was suddenly created. The pyrolysis model adopted here appears to provide a promising approach to the prediction of fire spread within enclosures. However, the models must be further developed to include physical behaviour such as charring and downward flame spread and its suitability for other fuels must be established.

The demonstrated ability of fire field models to exploit parallel computing techniques enable these models to be accurately and efficiently employed in large geometries such as B747 and A340 aircraft and their successors. Without this capability, compromises in mesh density and model complexity would be necessary in order to make simulations practical. The linking of aircraft fire models to other predictive models such as water spray models will also be of great benefit to the aviation industry as it strives to find a replacement for Halon extinguishment systems.

7. ACKNOWLEDGEMENTS

Professor E R Galea is indebted to the CAA for their financial support of his personal chair in Mathematical Modelling at the University of Greenwich. The research work described in this paper and originating from the University of Greenwich could not have been achieved without the dedicated efforts of the staff of the Fire Safety Engineering Group and the financial support of our main sponsors the UK CAA and UK EPSRC

8. REFERENCES

1. Galea, E R. "On the Field Modelling Approach to the Simulation of Enclosure Fires, *J of FireProt.Eng* , Vol 1(1), pp.11-22, 1989.

2. Simcox, S., Wilkes N. S., and Jones I. P., 1988, "Fire at Kings Cross Underground Station, 18th November 1987: Numerical simulation of the buoyant flow and heat transfer," Technical report, U.K. Atomic Energy Authority Harwell Report AERE-G 4677.

3 Galea, E. R., and Markatos, N. C , "The Modelling and Computer Simulation of fire development in Aircraft," *Int J of Heat and Mass Transfer*, Vol. 34 No 1 pp 181-197, 1991.

4 Galea, E. R., and Hoffmann, N. A ,"The Numerical Simulation of Aircraft Cabin Fires," Technical Report, CAA Paper 95008, CAA London, 1995.

5 Galea, E R., Hoffmann, N., and Berhane, D., "Large-Scale Fire Field Modelling - the Route to General Use via Parallel Processing," *Proceedings of Interflam '93*, Oxford, England, 1993.

6 Galea E R and Markatos N C. A Review of Mathematical Modelling of Aircraft Cabin Fires App Math Mod, Vol 11, 161, 1987.

7. Galea E R and Markatos N C. Modelling of Aircraft Cabin Fires. Fire Safety Science Proc of the 2nd Int Symp Tokyo, Hemisphere Pub Corp pp801-810, Editors Takao Wakamatsu et al, June 1988.

8 Galea E R and Markatos N C. Forced and Natural venting of aircraft cabin fires: A numerical simulation. Proc 73rd Symp on Aircraft Fire Safety, NATO AGARD Portugal 22-26 May 1989

9. Kuminecz, J.F and Brickner, R.W, Full Scale Flammability Data for Validation of, Aircraft Fire Mathematical Models, NASA Tech Memo No 58244, 1982.

10. Hill, R, Boris, P.N and Johnson, G.R, Aircraft Cabin Compartmentation Concepts for Improving Post Crash Fire Safety, FAA-NA-76-12.

11. Meland, O, New Technology to reduce fire losses and costs, Ed Grayson, S.J and Smith, D.A, "The influence of the ventilation system upon smoke-filled enclosures and the detection of smoke", Elsever App Sci, 1986, p 316.

12 McCaffrey B.J , King-Mon Tu, Rinkinen W.J. and Eklund T.I. A model study of the aircraft cabin environment resulting from in-flight fires. DOT/FAA/CT-90/22 November 1992

13. Galea, E. R., and Ierotheou, C., "Fire-Field Modelling on Parallel Computers," *Fire Safety Journal*, 19,pp 251-266, 1992.

14. Air Accident Investigation Branch. Report on the accident to Boeing 737-236 Series 1, G-BGJL at Manchester International Airport on 22 August 1985. Department of Transport, Aircraft Accident Report 8/88, HMSO, ISBN 0 11 5508929, 1989.

15. http://www.gre.ac.uk/research/cms/fire/

16. Dodd D.C., Hall C.T.M., Pollard J. and Snell M.A ; "Burnthrough resistance of fuselages: Initial findings", CAA Paper 94002, ISBN 0860395936; 1994

17. Sarkos C P and Hill R. G.; "Characteristics of Transport Aircraft Fires Measured by Full-Scale Tests", Proc 73rd Symp on Aircraft Fire Safety, NATO AGARD. AGARD-CP-467, pp 11-1 - 11-19, Portugal 22-26 May 1989

18. Dussa K., Fiala R., Wagner R. and Zenses B., "Full Scale Study of a Cabin Fire in an A300 Fuselage Section"; Proc 73rd Symp on Aircraft Fire Safety, NATO AGARD. AGARD-CP-467, pp 12-1 - 12-16. Portugal 22-26 May 1989.

19. Speital L , " Fractional Effective Dose Model for Postcrash Aircraft Survivability"; Proc Int Colloquium on Advances in Combustion Toxicology, FAA CAMI, Oklahoma City, USA, 11-13 April 1995 To appear in Toxicology, 1996.

20 Magnussen, B., F., and Hjertager, B., H.,"On Mathematical Modelling of Turbulent Combustion with special emphasis on soot formation and combustion," *16th Int Symp on Combustion*, pp 719-729, 1979

21. Chen, Y., Delichatsios, M.A and Motevalli, V., "Material pyrolysis properties, part I: an integral model for one dimensional transient pyrolysis of charring and non-charring materials", Combust. Sci. Tech. 88, 309--328, 1993.

22. Parker, W.J., "Prediction of the heat release rate of wood", Proceedings Fire Safety Science First International Symposium, 1985, pp 207-216.

23 Staggs, J.E.J and Whiteley, R.H., "Modelling the combustion of solid-phase fuels in cone calorimeter experiments", Proceedings Interflam'96, Compiled by C.Franks and S Grayson, ISBN 0 9516320 9 4, 1996, pp. 103-112.

24 Delichatsios, M.A., Panagiotou, Th., and Kiley, F., "Use of time to ignition data for characterizing the thermal intertia and the minimum(critical) heat flux for ignition or pyrolysis", Combust. Flame 84, 323-332, 1991.

25. Kindelan, M. and Williams, F.A., "Theory for endothermic gasification of a solid by a constant energy flux", Combust Sci Technol. 10, 1--19, 1975.

26. Jia, F., Galea, E. R., and Patel, M K., The Prediction of Fire Propagation in Enclosure Fires. Paper No 96/IM/10, ISBN 189999 10 9 3, CMS Press, 1996

27. CAA, "Cabin Water Sprays for Fire Suppression: An Experimental Evaluation," Technical Report, CAAPaper 93009, 1993.

28 Hill, R., Sarkos, C , and Marker, T , "Development and Evaluation of an Onboard Aircraft Cabin Water Spray System for Postcrash Fire Protection," *SAE Technical Paper*, Series 912224, Aerospace Tech Conf and Exposition Long Beach Calif., 1991.

29 Mawhinney, R , N., Galea, E , R , and Patel, M , K , "Fire-Sprinkler spray interaction modelling using an Euler-Lagrange approach," Proceedings Interflam'96, Compiled by C Franks and S Grayson, ISBN 0 9516320 9 4, pp841-846, Interscience comm Ltd, 1996.