THE FIREDASS (FIRE Detection And Suppression Simulation) MODEL

A.J. Grandison[*], R.N. Mawhinney[*], E.R. Galea[*], M.K. Patel[*], E.P. Keramida[+], A.G. Boudouvis[+] and N.C. Markatos[+].

Fire Safety Engineering Group[*]
University of Greenwich
Wellington Street
London SE18 6PF, UK
http://fseg.gre.ac.uk

Department of Chemical Engineering[+]
National Technical University of Athens
Zografou Campus
Athens 15780, Greece

ABSTRACT

The FIREDASS (FIRE Detection And Suppression Simulation) project is concerned with the development of fine water mist systems as a possible replacement for the halon fire suppression system currently used in aircraft cargo holds. The project is funded by the European Commission under the BRITE EURAM programme. The FIREDASS consortium is made up of a combination of Industrial, Academic, Research and Regulatory partners. As part of this programme of work, a computational model has been developed to help engineers optimise the design of the water mist suppression system. This computational model is based on Computational Fluid Dynamics (CFD) and is composed of the following components: fire model; mist model; two-phase radiation model; suppression model and detector/activation model. The fire model – developed by the University of Greenwich - uses prescribed release rates for heat and gaseous combustion products to represent the fire load. Typical release rates have been determined through experimentation conducted by SINTEF. The mist model – developed by the University of Greenwich - is a Lagrangian particle tracking procedure that is fully coupled to both the gas phase and the radiation field. The radiation model – developed by the National Technical University of Athens - is described using a six-flux radiation model. The suppression model – developed by SINTEF and the University of Greenwich - is based on an extinguishment criterion that relies on oxygen concentration and temperature. The detector/activation model – developed by Cerberus - allows the configuration of many different detector and mist configurations to be tested within the computational model. These sub-models have been integrated by the University of Greenwich into the FIREDASS software package. The model has been validated using data from the SINTEF/GEC test campaigns and it has been found that the computational model gives good agreement with these experimental results. The best agreement is obtained at the ceiling which is where the detectors and misting nozzles would be located in a real system. In this paper the model is briefly described and some results from the validation of the fire and mist model are presented.

1. BACKGROUND

At present the fixed halon system is the most commonly used type of fire suppression and extinguishment system. In particular, its light weight and proven effectiveness has led to its exclusive use in aircraft cargo holds. The advent of the Montreal protocols (1986) banning the production of Chlorofluorocarbons (CFCs), however, means that aircraft manufacturers and operators can no longer use this system. Industry has therefore been investigating alternatives, including powders, other gases and water mists. GEC-Marconi Avionics (GMAv, UK) is a potential supplier of a water mist system and is seeking to optimise its system for aircraft applications. For example, in an aircraft a primary consideration is the weight of water required to suppress the fire as this will affect the aircraft running costs.

Another problem with aircraft fire safety is the fire detection systems which are installed in the cargo holds. Currently approximately 95% of all reported smoke warnings are false alarms. New fire detection technology is required in order to improve reliability and to activate any new technology fire suppression systems. Cerberus Guinard (CG, France) is a manufacturer of fire detection systems and is seeking to develop new systems for aircraft cargo holds that have a greater reliability. It has a need to optimise sensor type and location in order to achieve maximum selection effectiveness.

Both companies found that they had a strong need for tools which could reduce the number of tests they had to perform during the development, optimisation and certification of their fire detection and fire suppression/extinguishment systems. Research showed that computer modelling techniques, in particular Computational Fluid Dynamics (CFD) based fire field models[1], could be used to create these tools. This led to the formation of the FIREDASS Consortium with the dual aim of (1) developing the computer modelling techniques required to develop and optimise the detection and suppression systems and of (2) using the tools developed to do said optimisation.

In order to validate the models developed and to aid in the optimisation of the systems under development the program included an experimental component designed to provide appropriate data. An additional aim of the project is to use this data to assist in the formulation of a European requirement for water mist fire extinguishment/suppression systems.

The consortium consists of GMAv (UK), CG (France), the University of Greenwich (UoG, UK), the National Technical University of Athens (NTUA, Greece), SINTEF NBL (Norway), the UK Civil Aviation Authority (CAA, UK) and DLR (Germany). The FIREDASS programme is sponsored by the European Commission under BRITE/EuRam Framework IV (contract no. BRPR-CT95-0040) and is scheduled to run from 2/96 - 2/99.

The need to improve fire detection systems is immediate. With the increasing cost of halon, and with depleting halon stocks, the demand for a halon replacement suppression medium is increasing. By the end of the programme, the Consortium will be in a position to provide an improved fire detection system and an acceptable halon replacement suppressant in the form of water mist. This time frame, the Consortium believes, will be commensurate with market demand within the airline industry.

2. THE COMPUTER MODEL

The computer modelling techniques needed have now been developed. This paper describes the submodels developed at UoG and the results of their testing and validation.

2.1 BENEFITS OF THE MODEL

The computer model provides a number of benefits. These include:

- the ability to model a fire detection/activation system configuration and water mist suppression/extinguishment system configuration to allow the combined systems performance (detection time, suppression efficiency and extinguishment capability) to be assessed without the need for a fire test;
- the ability to model a wide range of different fire scenarios, fire positions and physical configurations of aircraft, ships, vehicles and buildings and hence to allow the optimisation of the systems;
- a consequent reduction in the development time and number of fire tests necessary for system certification;
- the replacement of synthetic extinguishing media that damage the environment.

Note that further work would be required to model alternative agents such as powders and gases.

2.2 STRUCTURE OF THE MODEL

The model is based on Computational Fluid Dynamics (CFD) techniques that have been successfully used in the past to model a variety of fire scenarios. CFD involves the numerical solution of the partial differential equations that describe the physical properties of the system given the appropriate initial and boundary conditions[2]. It has become increasing popular due to the increasing power and decreasing cost of computers, the increased power in particular meaning that the computational time required for the solution of highly complex problems has been significantly reduced.

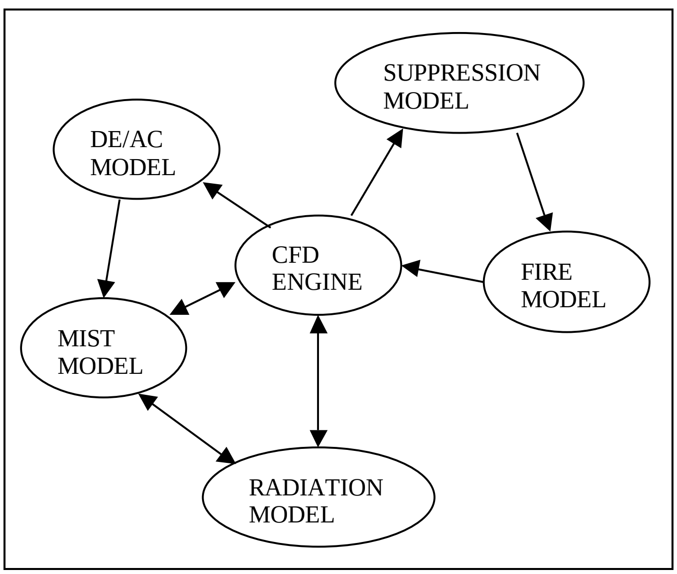

Figure 1 - Interactions between FIREDASS submodels.

The FIREDASS computer (CFD) model consists of the following submodels:

- A fire submodel (developed by UoG)
- A mist submodel (developed by UoG)
- A radiation submodel (developed by NTUA)
- A detector/activation submodel (developed by CG)

- A suppression submodel (developed by UoG and SINTEF)
- A CFD engine (CFX4.1) (developed by a 3rd party supplier, AEA Technologies)

The interactions between these submodels are shown in Figure 1.

2.2.1 The FIREDASS fire submodel

The fire submodel[3] simulates the fire. It supplies heat, smoke and gaseous combustion gases (carbon dioxide, carbon monoxide, water vapour and oxygen) production/consumption rates to the CFD engine. The rates supplied at any time come from rate tables. These are prescribed by experiment as the present fire model does not perform any combustion calculations.

The heat source is added into the enthalpy equation. The smoke is added as a concentration flux to a scalar equation. The gaseous combustion gases are modelled as simple mass fractions and an appropriate mass flux is added to these equations. In addition the combined mass flux from these equations is added to the pressure correction (mass continuity) equation as a source of mass. It is assumed that the source terms all act in the same specified volume.

2.2.2 The FIREDASS radiation submodel

The radiation submodel[4] was developed by NTUA and simulates the radiation field in the compartment and its interaction with the fire, air, smoke and water mist. It is a multiphase model based on the six flux model and generates opacities and scattering coefficients for the air, smoke and water mist in order to model their effect upon the radiation field.

The concentration of smoke is used to generate a modified combined opacity for the air/smoke system which is used to determine how this system absorbs energy from the radiation field[5]. This opacity is calculated by adding the opacity of air to an opacity for the smoke calculated using Rayleigh theory[11] to generate an average absorption coefficient from the smokes volume fraction and temperature. From this opacity the amount of radiation absorbed by the air and smoke is calculated. This quantity is then passed to the gas phase as a source.

The opacity of the mist is calculated by summing over the opacities for each particle size present[6]. These opacities are calculated from the number of droplets of that size, their projected area, their residence time in the cell and their wavelength averaged scattering efficiency calculated using the Mie formula[12]. From this "mist" opacity, for each computational cell, the amount of radiation absorbed by the mist in that cell is calculated. This is passed to the mist submodel as a source.

2.2.3 The FIREDASS mist submodel

The mist submodel[7] simulates the behaviour of the water mist injected by the nozzles and its interaction with the fire atmosphere and the radiation field. The formation of the mist by the nozzle is not modelled as this would constitute a major modelling exercise in its own right. Instead the state of the mist at this point was measured experimentally and the results are used as a boundary condition. Also, no attempt is made to model the direct interaction of the mist with the fire, i.e. neither mist interaction with flames nor fire suppression via wetting of surfaces is considered. However, the fire model does include an extinguishment criterion that is based on the average temperature and oxygen concentration in the vicinity of the fire. Thus in an indirect way the mist model does interact with the fire via mist droplets entrained into the volume of the fire causing cooling and oxygen displacement as they evaporate.

The mist submodel is a Lagrangian particle tracking model[13]. It tracks a representative sample of the droplets from the nozzle through the domain, terminating the tracking either upon contact with a surface or upon evaporation of the droplet. Momentum, heat and mass transfer are considered. Heat transfer includes absorption of heat from the radiation field. Tracking is accomplished by evaluating a closed form integral solution of the droplet transport equations. Gas phase sources to the droplets are implicit in the equations solved for the droplet histories. Droplet phase sources to the gas are applied using the PSI cell method[8]. Two way coupling is achieved by iterating the solution of the gas and droplet phases on each timestep.

The source of enthalpy from the radiation field is applied to the droplet phase by summing the surface area times residence time product for each droplet over all the droplets which pass through a computational cell and then distributing the source to the droplets proportionate to each droplets own surface area times residence time. This source is added to the source from the gas by convection and the loss from evaporation to get the total droplet enthalpy and therefore temperture change as it passes through a cell.

3. VALIDATION OF THE MODEL

3.1 VALIDATION OF THE FIRE SUBMODEL

During the early stages of the project SINTEF performed a series of fire tests specifically to provide data for use in the validation of the computer models developed[9]. These tests involved gas burners, cardboard boxes and kerosene pool fires. The tests were simulated using the FIREDASS computer model and the predicted results compared with the experimental. The results for the gas burner cases are described below.

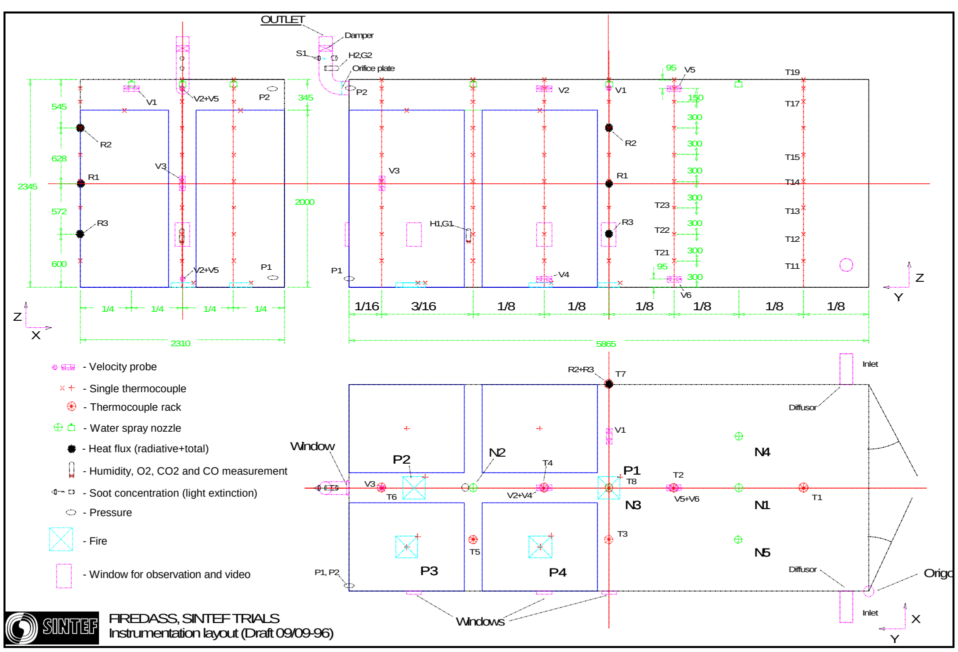

Figure 2. Diagram showing instrumentation of SINTEF test chamber.

The nature of a CFD code is such that predicted results are available anywhere within the computational domain (i.e. the chamber). In the experiments temperatures were recorded as a function of time at 72 locations, see Figure 2. Thus temperature comparisons could be performed at each experimental location. Due to limitations on space however only a few representative locations will be compared in this paper. The four representative thermocouple locations selected are:

1) T12 – Near the floor and door;
2) T18 – In the same thermocouple stack as T12 but just below the ceiling;
3) T58 – Just below the ceiling between fire positions P3 and P4;
4) T36 – Located centrally and approximately ¾ height.

Comparisons are also presented for gas and smoke concentrations in the outlet (G2, H2) and at one location in the room (G1, H1).

3.1.1 Preliminary discussion of results

There are a number of general factors applying to all fire modelling validation that lead to differences between experimental and computed results. For these results the following additional issues arise:

1) the thermocouple temperature is not necessarily the same as the gas temperature due to absorption/emission of radiation;
2) there is a large degree of uncertainty associated with the release rate data and the water vapour concentration measurements;
3) there are insufficient measurements of key parameters such as oxygen concentration and heat fluxes.

3.1.2 The 02HS case (the propane burner case)

3.1.2.1 Preliminary work

The first simulations of this case assumed that the walls were at a constant temperature of 288K and did not include radiation. As can be seen in Figure 3, when radiation was included the predicted temperatures dropped by around 50% giving temperatures much closer to the observed experimental values.

3.1.2.2 Results

The 02HS case was run, with radiation, for the experimental time range 25-800s. A combustion efficiency of 80% was assumed throughout the whole time period. Figure 4 to Figure 13 compare the predicted and measured results.

<u>Temperatures</u>

Figure 4 to Figure 7 compare predicted and measured temperatures. They are labelled internally with the appropriate thermocouple position. This position can be identified from Figure 2.

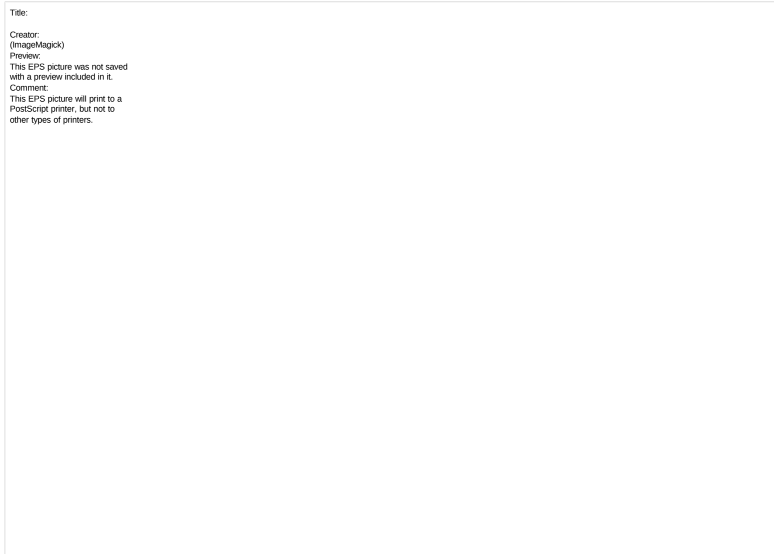

Figure 3 - Comparison of FIREDASS model with and without radiation.

Figure 4 – 02HS case. Predicted and measured temperature variation at T12 (near floor and door).

Figure 5 - 02HS case. Predicted and measured temperature variation at T18 (near ceiling and door).

Figure 6 - 02HS case. Predicted and measured temperature variation at T58 (near ceiling and fire).

Figure 7 – 02HS case. Predicted and measured temperature variation at T36 (3/4 height and central).

It can be seen that the temperature variation is well predicted by the fire model and it is clearly demonstrated that a reduction of 20% in the release rates (in order to represent an actual combustion efficiency) yields a closer agreement with experimental values. This is not surprising as it could be expected that a 100% combustion efficiency, i.e. a fully efficient fire, would never actually occur. In reality the combustion efficiency would not remain constant but would tend to decrease as the oxygen within the chamber was consumed. However it is not possible to predict this variation without a detailed combustion model which is not within the scope of the present project.

The temperature predictions (see Figure 5 and Figure 6) near the ceiling provide quite good agreement with experimental results. From the point of view of creating a temperature based detection system this is an important result.

Combustion products

Figure 8 to Figure 13 compare predicted and measured concentrations for the combustion gases.

From the figures it can be seen that the predicted values for the major combustion species are in good agreement with the experimentally determined values. Smoke and CO were not considered as the experimental measurements for smoke and CO were small for the 02HS case.

Figure 8- 02HS case. Predicted and measured CO2 variation at the outlet.

Figure 9 - 02HS case. Predicted and measured CO2 variation in the room.

Figure 10 - 02HS case. Predicted and measured O2 variation at the outlet

Figure 11 - 02HS case. Predicted and measured O2 variation in the room.

Figure 12 - 02HS case. Predicted and measured H2O variation at the outlet.

Figure 13 - 02HS case. Predicted and measured H2O variation in the room.

Common features

The predicted trough in the outlet readings for all the tests is much narrower than that measured. This can be partially attributed to the modelling of the chimney (outlet). The model assumes that any air coming through the chimney from outside has mass concentrations of 23% oxygen and 77% nitrogen. The model further assumes that the temperature of this air is 288K (ambient). In reality the air that is sucked back through the chimney will be a mixture of ambient air and the exhaust air from the chamber. This air will therefore be hotter and will contain some amount of combustion products. This results in the chamber not being cooled as quickly by the incoming air, which leads to the trough being wider than that predicted by the model.

3.2 VALIDATION OF THE MIST SUBMODEL

The mist submodel was validated by simulating a set of tests carried out by GEC and comparing the results[10]. The tests were carried out for the purposes of (1) determining the characteristics of the nozzle (such as its throw) and (2) providing experimental measurements for the validation of the mist submodel.

The experiments involved running a nozzle for one minute and then allowing the mist to settle for a further two minutes. The nozzle was placed above an array of collecting trays. There were 100 trays, each 0.3m square, covering an area 3.0m square. When settling was completed the trays were weighed to determine how much water had collected in each. In the simulations the nozzle was run for the same time and the droplets then allowed to settle. The results were post processed by using the point at which the droplet hit the floor to determine which tray it would have landed in and determining the weight in each tray by summing over all the droplets tracked. In order to compare results the weights were plotted as contour graphs of the percentage of the total water delivered which landed in each tray.

Figure 14 shows the layout of the test chamber and the following four figures compare the measured and predicted distributions for two cases. In the first case nozzle N1 only was active and in the second case nozzles N1 and N2 were active.

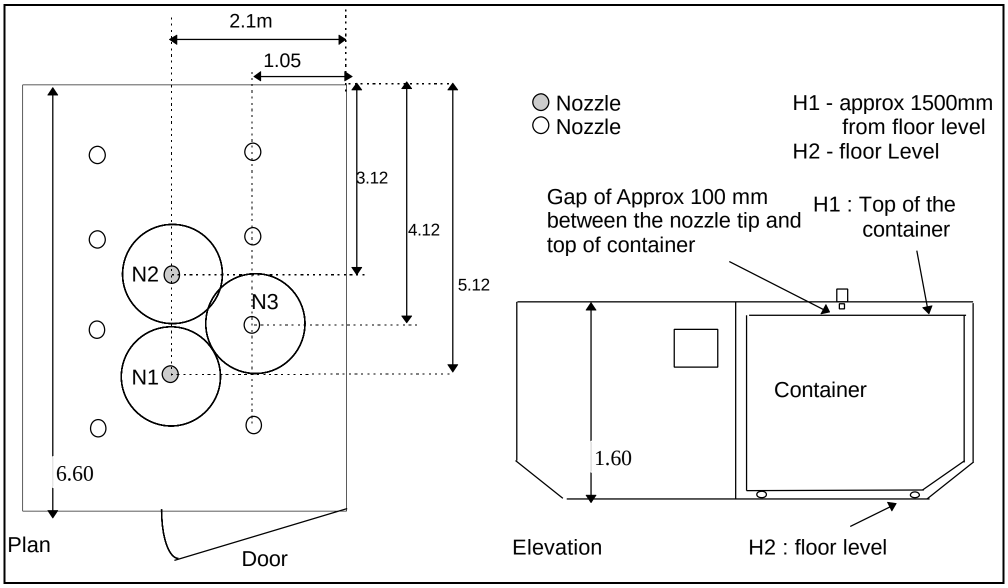

Figure 14 - Layout of test chamber used to do mist validation experiments.

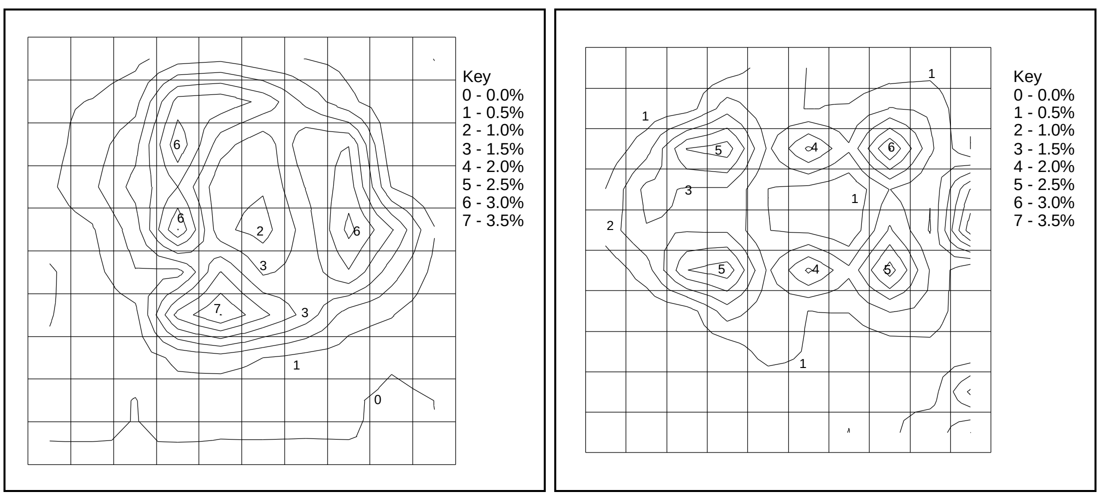

Figure 15 and Figure 16 - Measured and predicted floor water distribution for nozzle N1. Contours represent amount of water collected per tray as a percentage of discharged water.

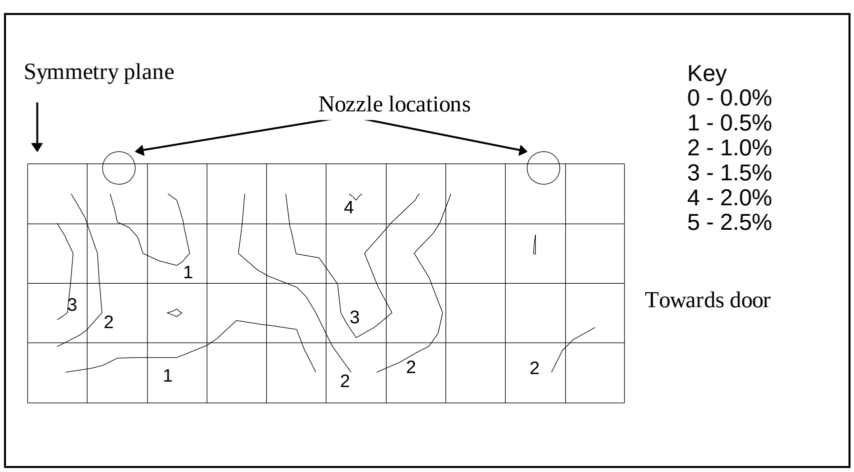

Figure 17. Measured floor water distribution for nozzles N1 and N2. Contours represent amount of water collected per tray as a percentage of discharged water.

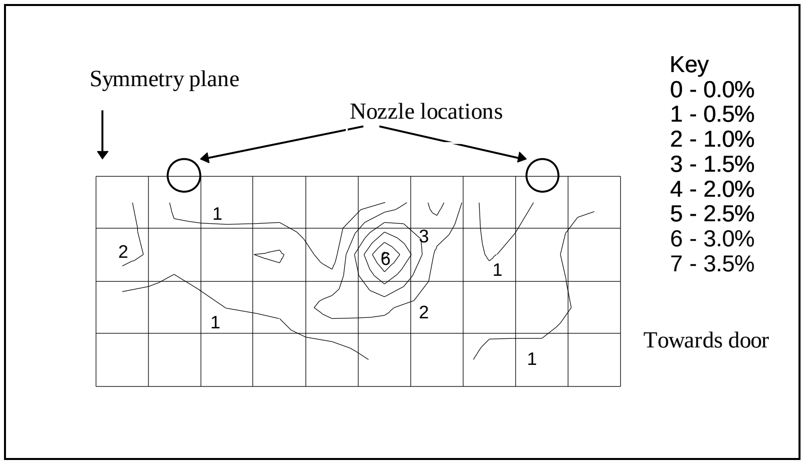

Figure 18. Predicted floor water distribution for nozzles N1 and N2. Contours represent amount of water collected per tray as a percentage of discharged water.

It should be noted that the mist is modelled using only a representative number of droplets as the actual number of droplets in the mist is very large. This inevitably means that the model produces results that are more discrete in nature, i.e. more “lumpy”, than those which actually occur. This effect is smoothed somewhat by grouping the droplets into trays but inevitably the simulated results will show a greater variation in the mass per tray than the experiment as large droplets carrying comparatively large mass arrive at random in one tray and not the next.

It should also be noted that in the experiments substantial but non-quantified amounts of the mist impacted upon the ceiling in the vicinity of the nozzle and then dropped into the trays below. This is an effect which can not be included in the model and which will therefore be responsible for variation between measured and predicted results.

Given these factors, plus the other approximations which had to be made in order to produce a workable model requiring reasonable amounts of computational time, it could be argued that too close agreement between experiment and prediction for these simulations should not be expected. Indeed it was expected that the model would only produce results that were in qualitative rather than quantitative agreement with the experimental observations.

It is argued, however, that these results are in fact much better than this and that they do show good agreement. In particular the following points about the single nozzle case should be noted:

1) the maximum values are similar being around 3.0 - 3.5% in both cases;
2) both sets of results show significant reductions in deposition directly under the nozzle;
3) the low concentration central region is surrounded by regions of high concentration (the 3% and 3.5% contour peaks in the experimental figure and the 2% and 2.5% contour peaks in the numerical predictions);
4) both show a slightly rectangular shape, though this is more pronounced in the simulated results and rotated 90^{o} to the experimental results;
5) taking the 0.5% contour line as a cut-off point, the two sets of results have similar sizes, being around 2 - 2.2m by 2 - 2.2m for the experimental results and around 1.8m by 2.5m for the numerical;
6) although it is likely to be for different reasons, both sets of results show broken rings of

deposition.

It should be noted that, still for the single nozzle case, the numerical results show a slight elongation towards the nearest wall (in this case the nearest wall being towards the right side of the plot). This may indicate that there is an interaction taking place between the flow and the wall. It is not clear if a similar process is taking place in the experiment, the results showing a deviation from the circular. The cause of this interaction may be a real effect or an artefact of the use of the Cartesian mesh favouring flow along an axis to a surface.

4. CONCLUSION

The fire submodel developed for the FIREDASS project uses specified production/ consumption rates for the main species involved, i.e. heat, smoke, CO, CO_2 and O_2. The model was not intended to predict these release rates but to transport the resultant products around the fire compartment. This was necessary due to the inherent uncertainties and computational costs associated with a computational model of gaseous and/or solid fuel combustion processes. The prescribed rates approach to modelling a fire is widely used and is considered to be a practical engineering approach to the prediction of the spread of fire products. However, in addition to the usual approximations introduced in a CFD based fire field simulation this approach introduces two further approximations. Firstly, as the combustion processes are not modelled release rates must be prescribed. Thus the details of the combustion process are lost. Secondly, the release rates are prescribed and hence the overall predictions are reliant on accurate release rates. Once the modelling approach has been selected the first approximation is accepted and introduces a fixed range of restrictions on the model capabilities. The second approximation can have a major influence on the quality of the model predictions and is within the control of the model to influence in that good representation of the release rates will produce good predictions of the chamber conditions within the constraints prescribed by the first approximation.

As such the quality of the model predictions depends very heavily on the quality of the experimentally defined release rates. While the experimental work did not produce release rate information to the level of accuracy originally anticipated, it has provided the FIREDASS model with data of sufficient accuracy to perform reasonable prediction simulation. From the work performed it can be seen that the fire model provides a good degree of agreement with the experimental work. Despite not modelling the effects of increasing ventilation control of the fire the computed results are still close to that of the experiments for heat and major combustion products. The main area of concern is the lack of good modelling of the minor combustion products (smoke and CO). Any improvement of the model will probably require additional experiments to try and get a better understanding of the combustion process. It is still possible, however, to tune the parameters with sufficient data. The most important of the two minor products is the smoke as it impacts on other aspects of the model. The model itself can represent three different fuel types at present, i.e. propane, kerosene and cardboard boxes.

The mist submodel is based upon the particle tracking methodology and is therefore sensitive to the momentum, heat and mass transfer relationships used to model the interphase transfer. There are two further main areas of uncertainty in the model. The first is the behaviour of the droplets immediately after leaving the nozzle when they are still in a stream of air also introduced by the nozzle. The way this affects the transfer processes is uncertain and cannot be modelled as this would require mesh resolution of the order of the nozzle hole size (1mm) which is not possible with current computing facilities for chambers of the size being modelled.

The second area of uncertainty is how much of the mist attaches to the ceiling and then evaporates from there. The rate at which this occurs will be significantly different due to the much smaller surface area to mass ratio. GEC noted that at the end to their tests the surfaces of the compartment were dry implying that all water impacting on the ceiling had evaporated though the rate at which this had happened could not be determined.

Despite these uncertainties, however, the results obtained both in the simulations of the footprint tests and in the simulations of the suppressed fire tests show good agreement with the experimental results.

5. ACKNOWLEDGEMENTS

The FIREDASS programme is sponsored by the European Commission under BRITE/EuRam Framework IV (contract no. BRPR-CT95-0040). The authors would also like to acknowledge the invaluable efforts of their collaborators Cerberus, GEC-Marconi Avionics, the National Technical University of Athens, SINTEF NBL, the UK Civil Aviation Authority and DLR. Finally, Prof Galea is indebted to the UK CAA for their financial support of his personal chair in Mathematical Modelling at the University of Greenwich.

6. REFERENCES

1. Galea, E. R.; "On the Field Modelling Approach to the Simulation of Enclosure Fires"; *J. of Fire Prot. Eng.*; Vol.1(1), 1989; pp.11-22.
2. Patankar, S. V.; "Numerical Heat Transfer and Fluid Flow"; *McGraw Hill*; 1980.
3. FIREDASS report; "The Fire Model Validation Report"; pro.* ref. no. 2.2-4; March 1998.
4. FIREDASS report; "Radiation Model Implementation"; pro.* ref. no. 3.1.3-4; May 1998.
5. FIREDASS report; "Radiative Properties of the Soot"; pro.* ref. no. 3.1.3-3; May 1997.
6. FIREDASS report; " Attenuation of Fire Radiation Through Water Droplets"; pro.* ref. no. 3.1.3-2; April 1997.
7. FIREDASS report; " Mist Model - Development, Implementation and Validation"; pro.* ref. no. 3.1.1-2; June1998.
8. Crowe, C.T., Sharma, M.P. and Stock, D.E.; "The Particle-Source-In Cell (PSI Cell) Model for Gas-Droplet Flows"; *Trans. of ASME - J. of Fluids Engr.*; June, 1977; pp325-332.
9. Wighus, R., Aune, P., and Drangsholt, G.; "FIREDASS – Fire Detection and Suppression Simulation"; Task 1.3.2 SINTEF Trials; SINTEF report No. STF84F97620, 1997.
10. FIREDASS report; "Spray Footprint and Fire Trials"; pro.* ref. no. 1.3.3-2; July 1997.
11. Modest, M.F.; "Radiative Heat Transfer"; *McGraw Hill International Editions*; 1993.
12. Barrett, J.; "The Optical Properties of Water Droplets in the Infrared"; J. Phys. D; Vol. 18, pp753-764, 1985.
13. Mawhinney, R.N., Galea, E.R. and Patel, M.K.; "Fire-Sprinkler Spray Interaction Modelling Using an Euler-Lagrange Approach"; *Proc. Interflam'96, the 7th Intl. Fire Sci. and Engr. Conf.*; 1996; pp841-845.

* All project reference numbers refer to Contract No. BRPR-CT95-0040, Project No. BE95-1977.