Simulating the Swissair Flight 111 In-Flight Fire Using the CFD Fire Simulation Software SMARTFIRE

F.Jia, M.K.Patel and E.R.Galea
Fire Safety Engineering Group
University of Greenwich, London SE10 9LS. U.K.
http://fseg.gre.ac.uk

ABSTRACT

AT 8.18pm on 2 September 1998, Swissair Flight 111 (SR 111), took off from New York's JFK airport bound for Geneva, Switzerland. Tragically, the MD-11 aircraft never arrived. According to the crash investigation report, published on 27 March 2003, electrical arcing in the ceiling void cabling was the most likely cause of the fire that brought down the aircraft. No one on board was aware of the disaster unfolding in the ceiling of the aircraft and, when a strange odour entered the cockpit, the pilots thought it was a problem with the air-conditioning system. Twenty minutes later, Swissair Flight 111 plunged into the Atlantic Ocean five nautical miles southwest of Peggy's Cove, Nova Scotia, with the loss of all 229 lives on board.

In this paper, the Computational Fluid Dynamics (CFD) analysis of the in-flight fire that brought down SR 111 is described. Reconstruction of the wreckage disclosed that the fire pattern was extensive and complex in nature. The fire damage created significant challenges to identify the origin of the fire and to appropriately explain the heat damage observed. The SMARTFIRE CFD software was used to predict the "possible" behaviour of airflow as well as the spread of fire and smoke within SR 111. The main aims of the CFD analysis were to develop a better understanding of the possible effects, or lack thereof, of numerous variables relating to the in-flight fire. Possible fire and smoke spread scenarios were studied to see what the associated outcomes would be. This assisted investigators at Transportation Safety Board (TSB) of Canada, Fire & Explosion Group in assessing fire dynamics for cause and origin determination.

1. INTRODUCTION

In this study, a CFD analysis of an in-flight fire in Swissair Flight 111 (SR 111), a McDonnell Douglas MD-11 aircraft, HB-IWF, is carried out. The fire caused the aircraft to crash into the ocean killing all 215 passengers and 14 crew members on board [1]. The four and a half year crash investigation that followed was one of the most challenging in aviation history.

The fire damage that was concentrated in the ceiling area of the forward fuselage of the SR111 aircraft created significant challenges to identify the origin of the fire, and to appropriately explain the heat damage observed. The wreckage indicated a highly complex pattern to the fire that required the analysis of a number of fire scenarios. Various techniques were required to assess the circumstances of the fire and its possible impact. A sound, systematic, scientific approach was necessary to evaluate hypotheses and substantiate findings through testing. To this end two primary approaches were available to provide further insight into the events leading to the accident, which could also be used to answer "what-if" type questions on possible events.

(1) The first option was to conduct a series of actual fire burn tests. This would prove to be very difficult as large scale burn tests would be necessary. The high costs associated with this approach and the difficulty in developing an appropriate test facility effectively ruled this approach out.

(2) The second option would be to carry out investigative fire tests using computer modelling, in particular the use of Computational Fluid Dynamics (CFD). Here virtual full scale experiments of the events that "may" have occurred could be investigated without the need to construct (and destroy) a large scale test facility. This approach would utilise data from smaller scale burn tests in order to predict larger scale events.

It is worth noting that the use of CFD techniques are not error free; indeed the route to prediction of possible scenarios requires data that can only be obtained with extensive laboratory based experiments. These in essence provide information relating to the heat release rate of materials, their material properties, the rate of fire spread depending on various conditions, behaviour of these materials when ignited – e.g. dripping effects, etc. Thus it is important to understand the limitations of such simulations.

It is clear that each approach has its advantages and disadvantage; however the key goal of the study undertaken was to validate and/or determine the probability that a scenario is "likely" and to quantify the heat released in the simulation. The rigours of such a complex simulation are quite demanding and only a few CFD based fire models are capable of handling the complex curved geometry of an aircraft fuselage, and the maze of cabling, ducting and other utilities in the above-ceiling voids. The TSB chose the specialist CFD-based fire simulation software, SMARTFIRE [2,3,9], developed by the Fire Safety Engineering Group at the University of Greenwich. The SMARTFIRE Computational Fluid Dynamics (CFD) software was used to predict the "possible" behaviour of airflow as well as the spread of fire and smoke within SR 111. The main aims of the CFD analysis were to develop a better understanding of the possible effects, or lack thereof, of numerous variables relating to the in-flight fire. This assisted investigators in assessing fire dynamics for cause and origin determination.

The pre-fire airflow CFD analysis was based to a large extent on data collected by investigators at Transportation Safety Board (TSB) of Canada, Fire & Explosion Group during MD-11 airflow flight tests. Using the SMARTFIRE software, CFD fire field modelling techniques were then applied to the pre-fire airflow patterns to initiate a fire at a pre-selected location. Possible fire and smoke spread scenarios were studied to see what the associated outcomes would be [9]. TSB investigators took the CFD results into consideration as part of their fire investigation. SMARTFIRE thus became the first CFD fire model to be used in an air crash investigation. The official TSB report acknowledged the technology's successful debut in 'helping to develop better insight into and understanding of the fire' [1].

2. SOFTWARE AND MODEL DESCRIPTION

A research version of the SMARTFIRE V3.0 [2,3] software was used to perform the fire simulations in this study. The fire field model, SMARTFIRE, is an open architecture CFD environment written in C++. It has four major components: a CFD numerical engine, Graphical User Interfaces, an automated meshing tool and an Intelligent Control System. As a specialist fire model, its CFD engine has many additional physics features, including an six-flux radiation mode, a multiple ray radiation model, provision for heat transfer through walls, a volumetric heat release model or gaseous combustion model (using the eddy dissipation model) to represent fires, smoke modelling and turbulence (using a two equation K-Epsilon closure with buoyancy modifications). Within SMARTFIRE the user can define a range of scalar variables, which can be used to represent the transport of products such as toxic gases and smoke.

SMARTFIRE uses three-dimensional unstructured meshes, enabling complex irregular geometries to be meshed without the recourse of cruder methods such as the stepped regular meshes or body-fitted meshes. The code uses the SIMPLE algorithm and can solve turbulent or laminar flow problems under transient or steady state conditions. As part of the SMARTFIRE development, the software has undergone considerable validation.

The fire simulations in this study were performed using a special version of the SMARTFIRE V3.0 software with features developed particularly for this project. These included a) new face patch types to facilitate new boundary condition types and surface burning conditions; b) new output file formats for monitoring locations, fire spread tracking information, etc.; c) FEMGV [4] mesh import filter for SMARTFIRE. The following software was also utilised during the analysis: a) RHINO V2.2 [5] to extract the aircraft geometry from the supplied CAD files; b) FEMGV V6.1 [4] to create the computational mesh of the aircraft geometry used in the SMARTFIRE simulations; c) and the free-ware package MayaVi [6] to create the three-dimensional visualisations of the data produced by SMARTFIRE.

The SMARTFIRE CFD field model was constructed from a simplified version of a detailed, three-dimensional CAD aircraft model provided by the TSB. The area selected for both the airflow, fire and smoke spread comprised of the cockpit interior and cockpit attic airspace, as well as the attic airspace above the forward cabin drop-ceiling and first-class ceiling areas. The aircraft coordinate system was preserved in the model. CAD elements were simplified to reduce the complexity and size of the original electronic files, while still retaining important three-dimensional features (i.e., curves and shapes were reduced to more basic surfaces and line segments). Due to the highly complex nature of the geometry in the aircraft, some of "small" features and components had to be either approximated or removed from the geometry to make the SMARTFIRE computer files manageable. The approach to simplify complex geometry is a common CFD practice. A snapshot of the components included is depicted in Figure 1. A further simplification of some of larger components was also necessary due to the complex nature of some of the shapes involved. In any case, the inclusion and/or omission of such "small" features and components and these simplifications of larger components were discussed with the TSB to try and ensure that any modifications that were made to the simulation space would be of little consequence to the airflow and fire. Finally, some components such as the cockpit crew seats that may not have played a role in the fire were represented with "hollow" interiors within the modelling space considered.

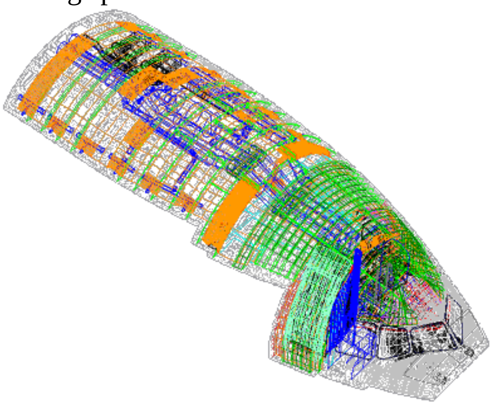


Figure 1: A snapshot of the CAD components used in the SMARTFIRE CFD model.

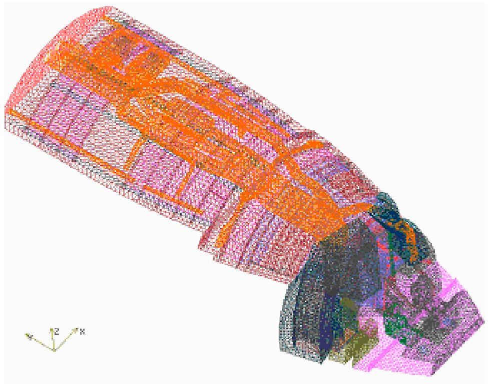


Figure 2: SMARTFIRE CFD model mesh used in the simulations.

The meshing task was undertaken with the aid of Femgv 6.1. The mesh consisted typically in excess of 280,000 tetrahedral elements. A typical SMARTFIRE mesh for this geometry is depicted in Figure 2. The size of each mesh cell in the cockpit was in the order of 1-6 cm; however care was taken to ensure that the mesh cells reduced in dimension as they approached solid surfaces. In the aft section of the aircraft, mesh cells were typically of the order of 10-12 cm. Each cell was assigned various properties, for example material properties, criteria for ignition, etc.

Key boundary conditions to generate the initial airflow patterns within the cockpit and ceiling, ambient external temperature, etc, were provided by the TSB. Several simulations were performed to ensure that the airflow pattern observed in the model, with the aid of minor changes to the inlet conditions, was similar to that observed in actual aircraft during airflow flight tests.

The TSB provided estimated MD-11 cockpit airflow values (e.g., conditioned air supply rates into and out of the cockpit) that were obtained from the Boeing Company for the flight condition that best represented the occurrences. These airflow values were used to set the initial airflow conditions at the various flow components in the CFD model. The model was then run and the velocity vectors and flow rates were studied at different locations within the virtual aircraft. Some adjustments were made to the flow rates until the model's velocity vector values closely matched those calculated from the flight test video recordings. The airflow patterns in the computer model were then compared to the smoke release flow patterns recorded on the flight test video recordings. The airflow patterns in the computer model were determined to be consistent with those observed experimentally.

Other parameters, not explicitly provided, and required to perform the simulations were derived. Where possible, the data was derived from data provided by TSB.

The TSB provided a series of cone calorimeter test data for the key combustible materials present within the aircraft. This data was used to derive the heat release rates required for the fire simulation together with an indication of when materials would ignite (not easily measured). The heat release rates used for the simulations were derived from the use of (i) raw cone calorimeter and (ii) averaged cone calorimeter data. This was necessary, as the raw data was at certain instances negative.

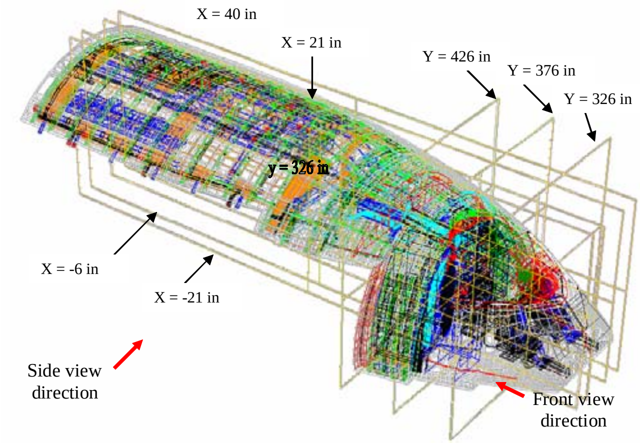
3. FIRE SCENARIOS

The fire ignition site, as provided by TSB, was prescribed with the aid of a volumetric heat source. The source was adjusted so as to sustain the combustion of the material over a short duration. This was necessary to allow the material to reach a critical temperature to allow flame propagation. In the simulation, the fire spread after initial ignition, was only allowed to spread once the neighbouring cell reached the ignition criterion. This allowed the heat release to be predicted by the amount of surface that was ignited as a result of the flame spread, if the ignition criteria were met. This approach eliminates use of the more complex route of predicting flame propagation via chemical reaction.

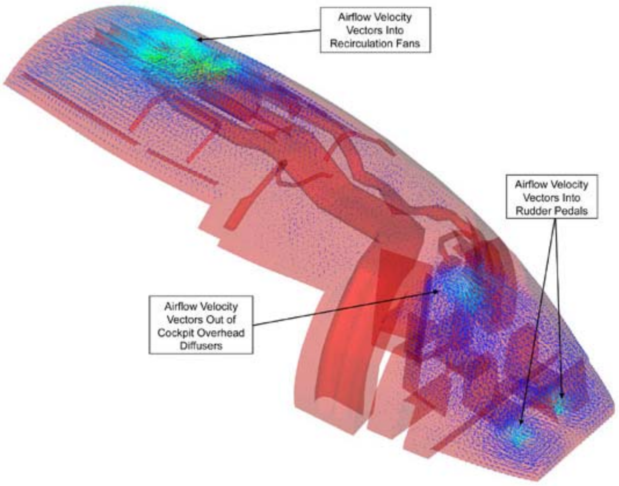
Similarly, fire spread to adjacent materials is addressed in the same way. However, in this case, one of two criteria needs to be met for the fire to propagate. The two criteria used to propagate the fire are (i) critical radiative heat flux from the surrounding fire that would ignite nearby material and (ii) critical surface temperature at which the material(s) would ignite. The ignition of the material is initiated when one of the of the two ignition criteria conditions are met. A combustible surface cell is allowed to ignite if it is in the vicinity of a region of a flame/flame tip for two seconds or more. Furthermore, the surface cell is also allowed to ignite based on experimental data available in literature [7] for similar thin-films considered here. Data available related to the vertical spread of flames were utilised here [7].

The main scenario considered for simulation was that the fire started within the confines of a relatively small area above the right rear cockpit ceiling, just forward of the cockpit rear wall near STA 383. The TSB determined that it was most likely that the fire started from a wire arcing event that ignited the nearby MPET-covered insulation blankets. These MPET-covered insulation blankets are easily ignited and were prevalent in the area. Of all the wires and cables that were located in the area of interest, the only arc-damaged wire that could be positioned in that area with relative accuracy were a pair of In-Flight Entertainment Network (IFEN) Power Supply Unit (PSU) cables. Although it is possible that other wires from this localised area that were not recovered might also have arced, the only arcing event that is known to have occurred within that area is the forward arc on TSB Exhibit 1-3791, located just forward of STA 383. The TSB requested that this location be used as the fire initiation site.

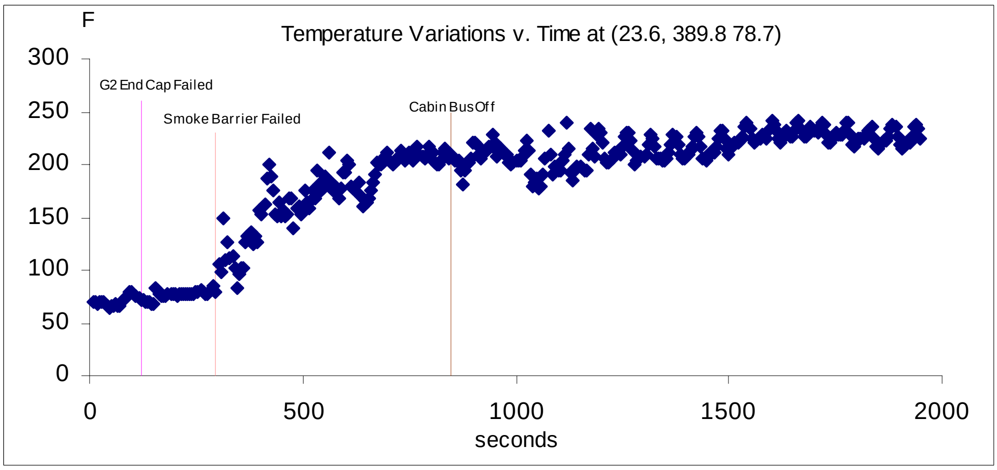
A heat release rate, to sustain the fire at the fire ignition site, was prescribed, as CFD simulations cannot predict arcing type ignition sources. To achieve this, it was necessary to ascertain what fire load was required to sufficiently heat the MPET in a small-localised area, so as to create a small self-propagating fire at the initiation site. This method of igniting the fire was agreed to by the TSB.



(a) Planes of importance for data presentation



(b) Three-dimensional VR type data presentation showing a pre-fire airflow example



(c) Temperature variations versus time at monitor point 2 (23.6in, 389.8in, 78.7in)
Figure 3: Data Presentation methods used for CFD results

Two types of scenarios were considered. These are defined as:

(1) Pre-fire Airflow Modelling (airflow only) – this allows the initial steady-state flow conditions, at the time of the fire ignition, to be predicted. This phase allows one to understand the effects of the various airflow boundaries and its effects on the flow pattern.

Furthermore, it also allows the comparison of the flow pattern with those measured and observed in the flight tests.

(2) Flow and Heat Modelling (with fire) – this allows the initial flow conditions predicted in (1) above to be used as the starting conditions for the scenario when the fire is assumed to have started.

4. SMARTFIRE GENERATED OUTPUT

Output of the results from the SMARTFIRE CFD simulation can be in one of the following three forms:

1. Graphical representation at prescribed times during the simulation in the form of still images. The data depicted with the images can be one of the following parameters: Flow pattern (velocity vectors), maps of temperature and smoke (line of filled contours) at planes depicted in Figure 3a. These key planes were provided by TSB.

2. 3-dimensional data files for MayaVi data visualizer are also provided, Figure 3b.

3. Transient data at various prescribed locations, Figure 3c. A total of 32 monitoring data locations have been activated, as requested by TSB.

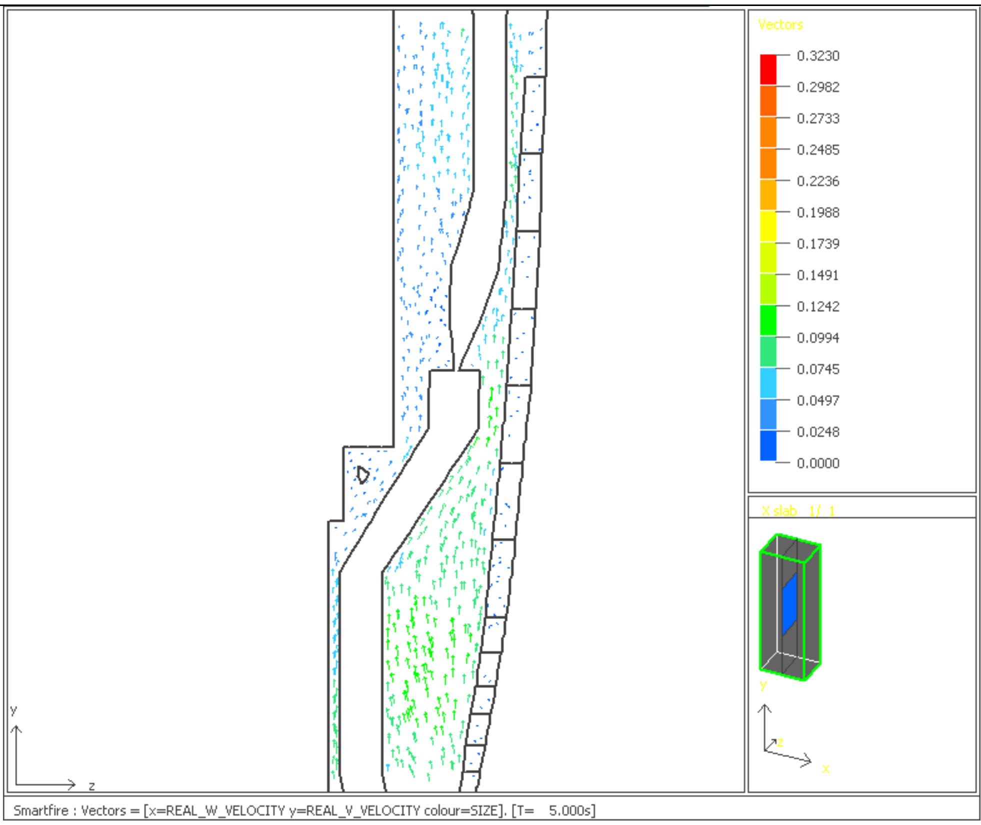
5. SIMULATION RESULTS

Results for two specific types of events were performed, i.e. pre-fire and post-fire. The pre-fire simulations were performed to ascertain the conditions in the cockpit and ceiling areas with respect to the air flow conditions only. Initial results, four flow rates, have been delivered to TSB. Furthermore, three post-fire simulations have also been delivered to TSB. Results relating to these simulations and the final set of simulations performed for the pre- and post- fire scenario are presented here.

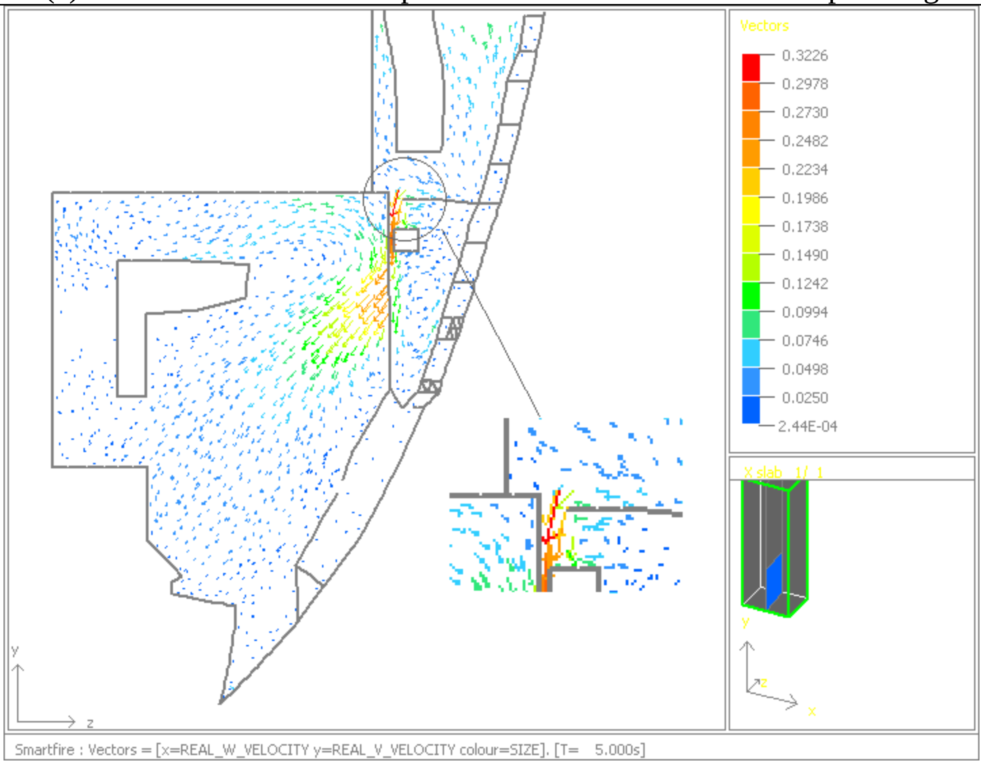
5.1 Pre-fire Flow Simulations

Four flow only simulations with different initial airflow conditions were performed in steady-state mode. Each simulation was converged within a maximum of 2000 iterations. To reduce simulation time and storage requirements, the heat-transfer and radiation options were not activated, as these were not required for flow only simulations.

The TSB provided estimated MD-11 cockpit airflow values (e.g., conditioned air supply rates into and out of the cockpit) that were obtained from the Boeing Company for the flight condition that best represented the occurrences. These airflow values were used to set the initial airflow conditions at the various flow components in the CFD model. The model was then run and the velocity vectors and flow rates were studied at different locations within the virtual aircraft. Some adjustments were made to the flow rates until the model's velocity vector values closely matched those calculated from the flight test video recordings. The airflow patterns in the computer model were then compared to the smoke release flow patterns recorded on the flight test video recordings. The airflow patterns in the computer model were determined to be consistent with those observed experimentally.



(a) airflow within the attic space above the forward cabin drop ceiling



(b) airflow around the smoke barrier

Figure 4: airflow (a) in the attic above the forward cabin drop-ceiling and (b) at around the smoke barrier

Findings of Airflow Flight Tests

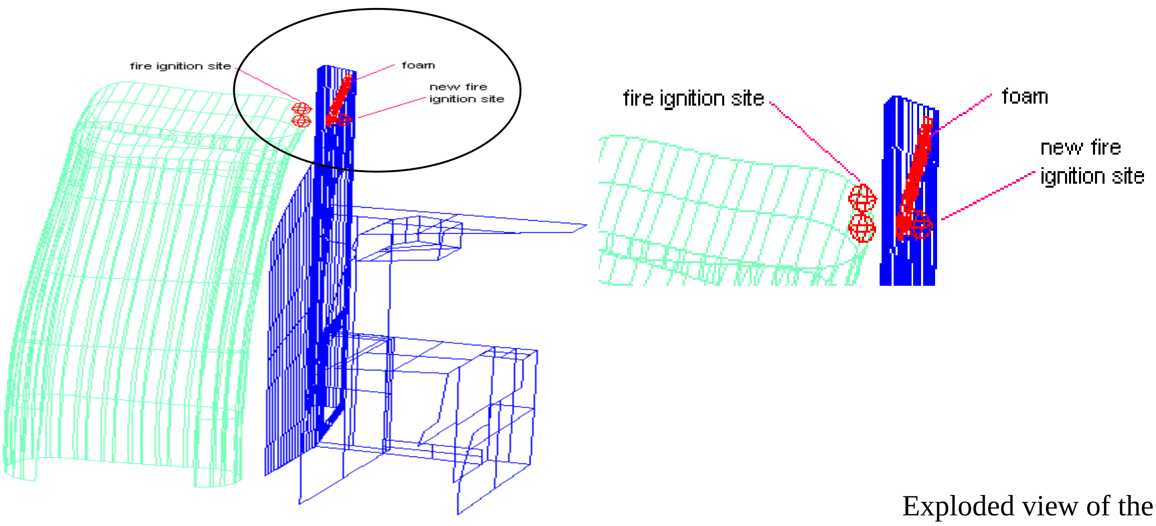
Airflow flight tests were conducted by TSB, Swissair and Boeing to assess airflow patterns in the cockpit, the attic space above the cockpit ceiling and the forward cabin ceiling. Due to the limitation on the paper length, only one of the findings of the airflow flight tests and the investigation[1] are presented below for comparison purpose with pre-fire simulations.

Observation: With the ECON switch selected to the ON position, airflow in the attic space above the forward cabin drop-ceiling is mainly drawn after to the recirculation fans. However, airflow near the cockpit rear wall could be drawn forward into the cockpit attic or down the cable drop and into the cockpit.

Error! Reference source not found.4a depicts the flow pattern aft of the cockpit in the attic space above the forward cabin drop-ceiling. The effect of the recirculation fans is clearly evident, where the airflow is predominantly aft due to the intake strength of the two recirculation fans, however this effect diminishes fairly quickly as one goes forward in the aircraft. Figure 4b depicts the airflow pattern into the attic space above the cockpit from the rear wall of the cockpit. It clearly shows the airflow through the small holes in the smoke barrier into the cockpit attic. These airflow patterns illustrated by the two figures of Figure 4 match well with **Observation**.

5.2 Post-fire Simulations

Six fire simulations were carried out in transient mode. The initial flow conditions for these fire simulations were obtained from the pre-fire simulations performed as part of the fire simulation. These cases consisted of various modifications leading to the final scenario of interest. Only the final simulation results with all the required revisions implemented are reported here.



Exploded view of the area of interest

Figure 4: The fire ignition sites are shown as red balls

The revisions include the new fire location, revised geometry (geometry near the central pedestal was updated; panels were added to seal off some space within the cockpit according to the updated CAD file provided by TSB). The G2 end cap was assumed to fail 120 seconds after the fire started.

In this simulation, the fire was ignited above the right rear cockpit ceiling, just forward of the cockpit rear wall. The location corresponds to the forward arc on TSB Exhibit 1-3791, which is located just forward of STA 383 (labelled as new fire ignition site in Figure 4).

Figure 6 depicts the total heat release rates vs. time, predicted from the simulation as the initial small creeping fire propagates over MPET and MPVF film surfaces.

Figure 7 depicts the flow rate vs. time at the cockpit ceiling crack. It clearly depicts that the peak flow at the crack, out of the cockpit, occurred when the Cabin Bus was switched OFF, whereas the peak flow, into the cockpit, occurs around 100 seconds after the fire starts and before the failure of the G2 End Cap. Past the Cabin Bus OFF condition, the flow at the crack diminishes gradually.

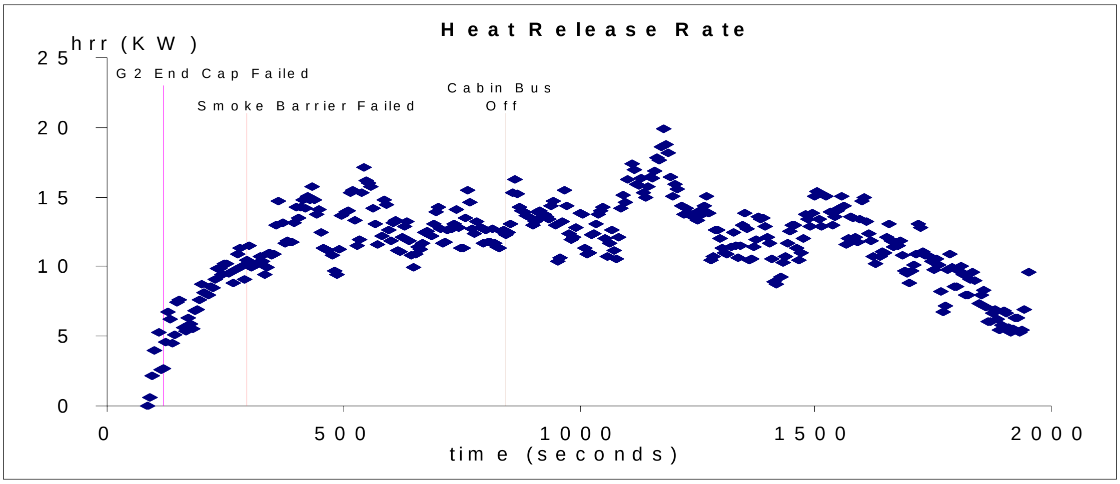


Figure 6: Heat release rates of burning of MPET and MPVF.

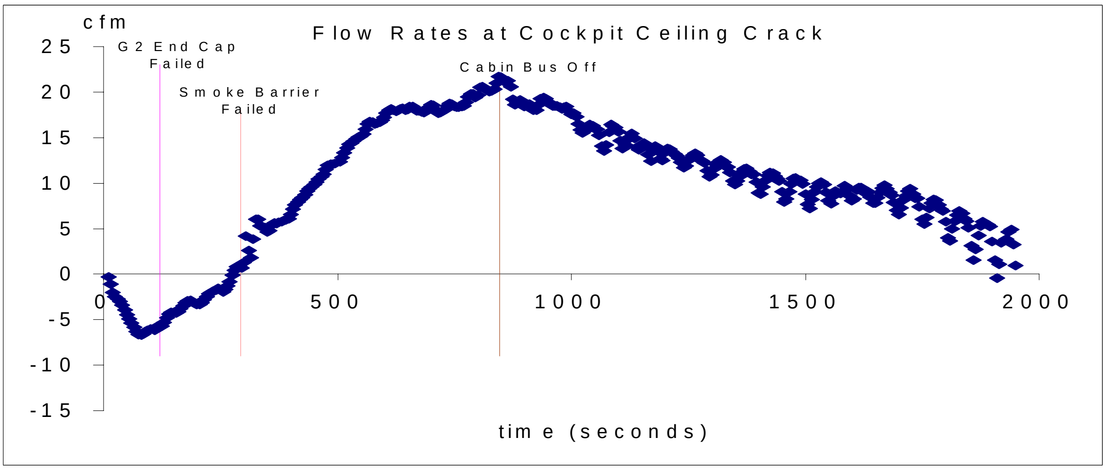


Figure 7: Flow Rate at Cockpit Ceiling Crack at Avionics CB Panel.

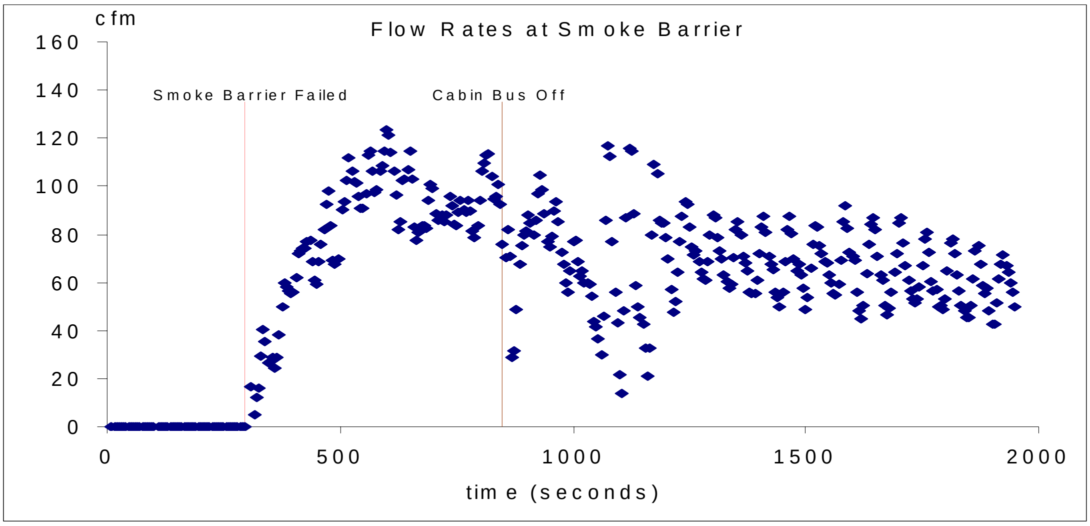


Figure 8: Flow rates at the smoke barrier.

Figure 8 depicts the flow rates vs. time at the smoke barrier. In this simulation, at approximately 5 minutes after the fire started, part of the smoke barrier collapsed due to high temperatures. The positive value indicates that air was being drawn aft, out of the cockpit through the newly created opening.

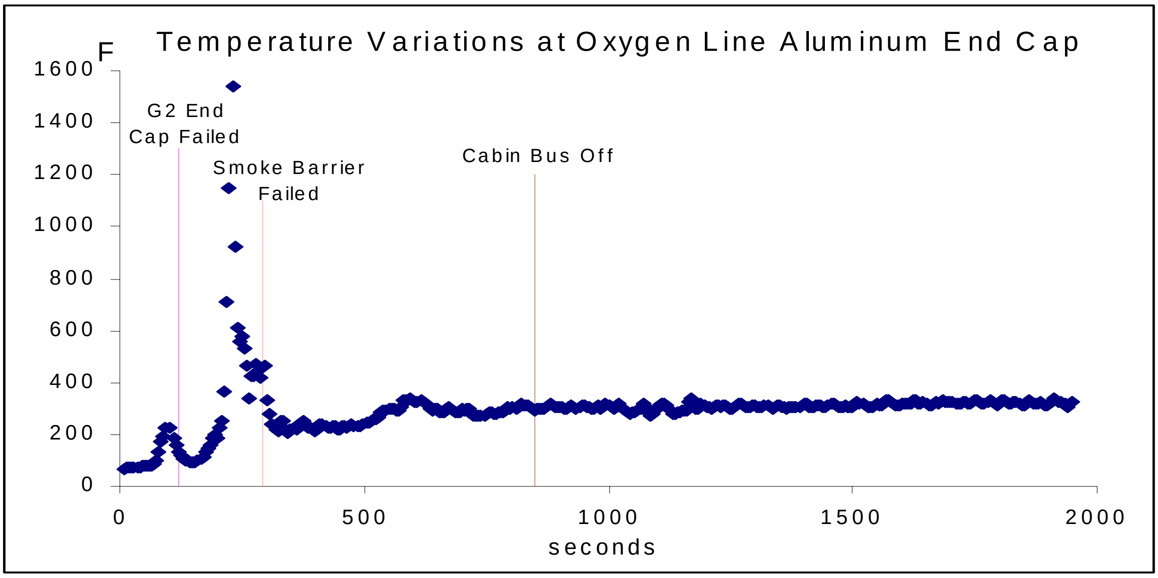


Figure 9: Temperature variations at Oxygen Line Aluminium End Cap.

Figure 9 depicts the time vs. temperature history of the oxygen line aluminium end cap. It is clearly evident that at approximately 300 seconds, there is a sudden increase in temperature for a very short duration. This is due to the propagation of the fire in the vicinity of the cap. The temperature of the cap stabilises between 350-400F after some 600 seconds. This is also the case when the Cabin Bus is switched OFF.

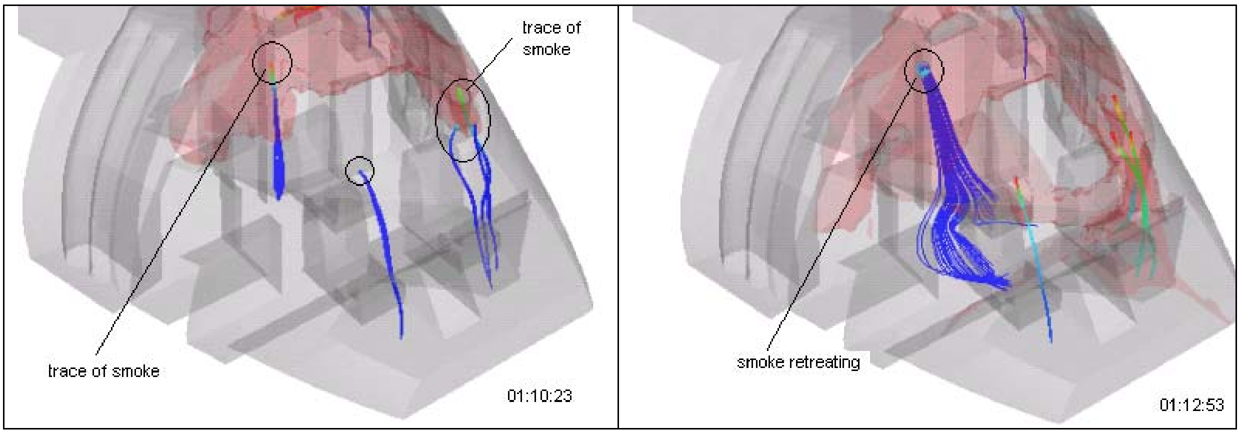
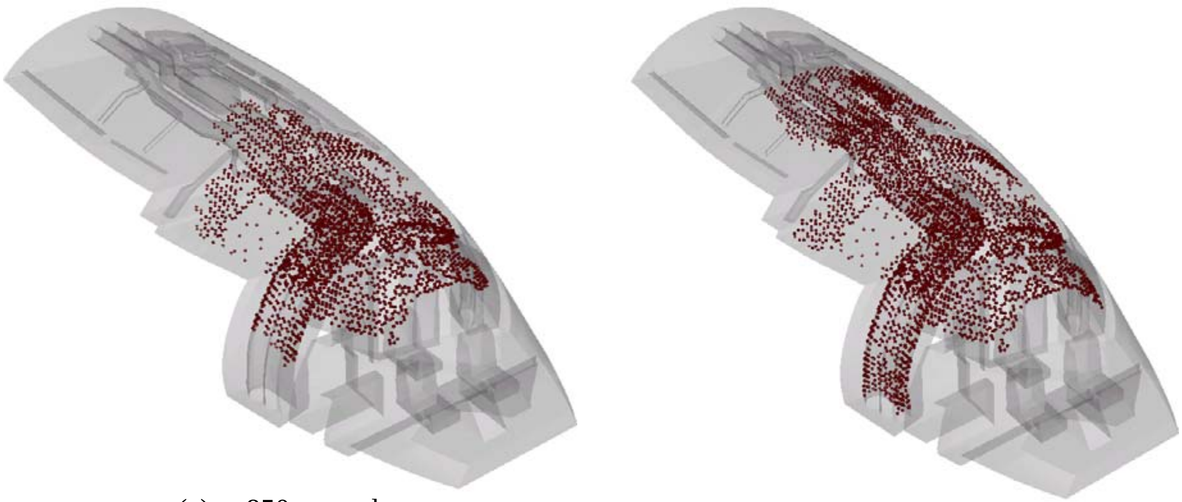


Figure 10: Flow at cockpit ceiling cracks at 80 seconds.

In the fire simulation, one of the main concerns is to reproduce the unusual odour and the smoke inside the cockpit at the early stage of the fire development within the aircraft. The picture on the left in Figure 10 shows the moment of smoke starting to enter the cockpit through the cracks at the ceiling at (i) the crack adjacent to the Avionics CB Panel, close to the right overhead diffuser, and (ii) through the small opening in the cockpit ceiling liner located above and to the left of the Captain’s seat. The blue lines are stream lines started at the cracks on the cockpit ceiling. They indicate the flow directions at the starting point of each stream line. The colour at the starting point indicates the smoke concentration. From blue to red means the increase of smoke concentration. The simulation predicted the event started at 01:10:23 while the investigation revealed that the first indication of the presence of odour and smoke within the cockpit is at 01:10:38. The picture on the right indicates that the smoke starts retreating from the cockpit. Note the colour at the top of the stream line changes from red to green. This indicates the smoke concentration becomes less at the crack and more importantly, the flow direction at the crack starts to change. Actually the flow at the crack is leaving the cockpit. The simulation predicted that this event started at 01:12:53 while the investigation revealed that it appeared that at 01:12:24 the smoke was no longer visible in the cockpit.



(a) at 850 seconds (b) at 1252 seconds

Figure 11: Fire spread at (a) 850s and (b) 1252s

Figure11 shows the fire damage on the insulation blankets and the riser ducts at 850 seconds just after the Cabin Bus was switched OFF. The predicted damage both aft and forward of the cockpit wall was 3.8 and 1.4 metres respectively at 850 seconds and 5.6 and 1.6 metres at 1252 seconds respectively. This compares well with data available from the SR 111 investigation [1], where extensive fire damage in the area above the ceiling in the front section of the aircraft was concentrated about 1.5 metres forward and 5 metres aft of the cockpit wall.

6. CONCLUSIONS

The work undertaken on behalf of TSB and presented in this paper has contributed to the formal investigation of the Post-Fire events associated with the SR 111 accident. The work focused on the innovative use of CFD fire modelling techniques to explore numerous "what-if" scenarios associated with the forensic fire investigation. In particular, the fire modelling software SMARTFIRE was used to gain insight into possible initial fire locations and the impact the resulting fire initiation had on the airflow circulation and the spread of smoke and fire. This demanding task was made more challenging due to the need to take into consideration the nature of the materials involved and the details of the highly complex geometry and the other interacting unit processes within the aircraft.

The geometrical data defining the aircraft geometry and components was specified by CAD drawings and imported into meshing software. The complex computational mesh used to define the volume of the aircraft to be investigated in the fire analysis was then generated over a period of time using a "trial and error" approach. This proved necessary due to the vastly varying length scales that were needed to represent the various important components present within the enclosure. Using this approach, it was possible to achieve stable and converged predictions for all the scenarios investigated. This analysis has highlighted that great care must be taken to ensure that key components of the geometry, i.e. those that will exert great influence on the nature of the predictions, must be adequately represented, no matter what their size or shape.

The pre-fire simulation predicted flow patterns matched what were observed in actual aircraft during airflow flight tests. The post-fire simulation predicted area of burning was close to the burning marks in the wreckage and captured the critical events of a small amount of smoke leaking into the cockpit and then quickly disappeared from the area.

The success of the approach adopted in this analysis is highly dependent on access to relevant material data. This includes both fundamental material properties and the behaviour of the material when ignited. Furthermore, this data must be readily available in a form suitable for use in computer modelling. For some of the materials found in aircraft, this data is not available and so well targeted experimental programmes will be necessary to support the future use of this approach.

The nature of the problems posed by this investigation also meant that the SMARTFIRE fire simulation software could not simply be used in an "off the shelf" manner. It was necessary to enhance the capabilities of the software to more accurately represent some of the complex boundary conditions that needed to be specified in order to represent some of the phenomena. The CFD analysis of the various scenarios was also undertaken in an iterative fashion. This approach provided information regarding the relative importance of various components and most importantly revealed their relative impact on the complex flow structures present within the cockpit and upper ceiling area.

Finally, with the successful completion of this project, SMARTFIRE has become the first fire model to be used in an air crash investigation. This study represents a significant milestone in the use of Computational Fire Engineering (CFE) tools, such as SMARTFIRE in forensic analysis. The official TSB report acknowledged the technology's successful debut in 'helping to develop better insight into and understanding of the fire'. This study has demonstrated that CFD based fire analysis is a cost effective approach to investigating complex flow/fire scenarios and that coupled with well targeted controlled experiments generating quality experimental data, CFD fire simulation can be a powerful tool in aircraft accident investigation. Combined with other CFE tools such as the aircraft evacuation model, airEXODUS [8], linking human behaviour and evacuation with both fire simulation and human response to toxic fire gases, these tools offer formidable capability to probe 'what-if' scenarios which would otherwise be the subject of informed speculation.

ACKNOWLEDGEMENTS

The authors are indebted to the Transportation Safety Board of Canada for their financial support in this project. Prof. Galea is also indebted to the UK CAA for their financial support of his chair in Mathematical Modelling.

REFERENCE:

1. Transportation Safety Board of Canada, In-Flight Fire Leading to Collision with Water, Report Number A98H0003
2. Wang, Z., Jia, F., Galea, E.R., Patel, M.K. and Ewer, J., Simulating one of the CIB W14 round robin test cases using the SMARTFIRE fire field models, Fire Safety J., vol. 36, 2001, pp. 661-677.
3. Ewer, J., Jia, F., Grandison, A., Galea, E.R., and Patel, M.K., SMARTFIRE V3.0 User Guide and Technical Manual, Fire Safety Engineering Group, University of Greenwich, UK, 2002.
4. Femsys: http://www.femsys.co.uk/
5. Rhino: http://www.rhino3d.com/
6. Mayavi visualizer: http://mayavi.sourceforge.net/
7. Quintiere, J. G., The effects of angular orientation on flame spread over thin materials, DOT/FAA/AR-99/86, U.S. Department of Transportation, Federal Aviation Administration, 1999.
8. Galea E.R., Blake S., Gwynne S. and Lawrence P., The use of evacuation modelling techniques in the design of very large transport aircraft and blended wing body aircraft. The Aeronautical Journal, Vol 107, Number 1070, pp 207-218, 2003.
9. Jia, F., Patel, M.K., Galea, E.R.."CFD Fire Simulation of the Swissair Flight 111 In-Flight Fire". Proceedings of the 10th International Interflam Conference, Edinburgh, 5-7 July 2004, Vol. 1, pp 1195-1206; ISBN 0 9541216 4 3.