Engineering Notes

ENGINEERING NOTES are short manuscripts describing new developments or important results of a preliminary nature. These Notes cannot exceed 6 manuscript pages and 3 figures; a page of text may be substituted for a figure and vice versa. After informal review by the editors, they may be published within a few months of the date of receipt. Style requirements are the same as for regular contributions (see inside back cover).

Pool Fires in a Simulated Aircraft Cabin Interior with Ventilation

C.P. Bankston,* L.H. Back,† Y.I. Cho,‡
and P. Shakkottai§
Jet Propulsion Laboratory
California Institute of Technology
Pasadena, California

Introduction

THE penetration and spread of fire into an aircraft cabin interior under postcrash conditions greatly increases the hazards to surviving passengers. In recent years, NASA and the FAA have supported investigations[1-3] that are leading to a better understanding of the postcrash fire scenario.[4] This paper describes one aspect of this work carried out at the Jet Propulsion Laboratory. Specifically, experiments have been conducted in a one-third scale simulated aircraft cabin geometry to study pool fire and ventilation flow interactions. The experiments represent the situation where an external fuel fire has penetrated the fuselage through a door or a crack and spread into the cabin. A ventilation flow is imposed on the fire to represent wind-induced flow through the fuselage due to open doors.

Test Setup

The experiment (Fig. 1) has been described in detail in Ref. 3. The test section, which is 0.76 m high by 1.52 m wide (approximately one-third scale of a wide-body transport interior) consists of four modular, interchangeable steel subsections. The total test section length is 3.4 m, and its exterior is covered with fiberboard insulation. Ventilating airflow is provided by a centrifugal blower, which supplies a maximum airflow of approximately 48 m^3/min (1700 CFM). A water-cooled fuel pan measuring 30 cm in the streamwise direction by 150 cm in the spanwise direction is located 1.11 m downstream of the test section inlet. It is mounted flush with the floor and is centered on the test section centerline. Turbojet A is used as the fuel in all tests, and a 2.5-cm-thick ceramic fiber wick placed in the fuel pan produces a relatively uniform burning surface.

Gas phase temperature data are provided by 60 type-K thermocouples located on probes in the flow. Two water-cooled radiometer and calorimeter pairs were mounted in the ceiling along the centerline of the test section 25 cm and 71 cm downstream of the back edge of the fuel pan. Oxygen concentration measurements made in the exhaust stack were utilized to calculate the energy release rates during tests by oxygen consumption calorimetry. Twelve viewports are provided for visual and photographic observations along the test section. In particular, these observations include data from five flow direction indicators spaced along the length of the test section.[3]

Experimental Conditions

Previously, we have reported results for experiments at three burning and ventilation conditions.[3] It was found that for flow velocities in the range expected for wind-induced flow, the visible fire plume was blown over significantly, except at the lowest velocity tested (~0.2 m/s). On the other hand, a significant reverse flow ceiling jet was formed under all conditions, leading to the transport of smoke and toxic gases *upstream* of the ventilating flow. In the work reported here, flow characteristics have been studied in more detail, velocity measurements were made in the reverse flow region upstream of the fire, and heat flux measurements in the ceiling were analyzed for the previous test conditions and for additional fire/flow combinations.

The test conditions and a summary of the results are shown in Table 1. The nominal flow velocities were determined by hot film anemometer traverses upstream of the fuel pan under cold flow conditions.[3] For tests 1–4, the fuel pan extended the full width of the test section. The heat release rates in tests 1 and 2 were varied by changing the fuel loading in the wick and water coolant flow through the fuel pan. In tests 5 and 6, the pan was shortened to 75 cm and was centered on the test section centerline. In test 6, a sheet metal barrier 38 cm high by 91 cm in the spanwise direction was centered on the floor 15 cm downstream of the fuel pan. This barrier could represent the presence of a row of seats adjacent to the fire. Tests were typically 3 min long, and the data presented are from a 2–2½-min quasisteady period beginning 30 s after ignition.

Flow Characteristics

Figure 1 shows the flow patterns in the test section for the highest crossflow velocity at 86 kW (test 1). The illustration shows the flame blown over significantly with a flame length of 0.5–1.0 m. This was also typical for the higher heat release rates up to 140 kW. Note the reverse flow ceiling jet that extends at least 0.5 m upstream of the fuel pan. It causes smoke to be spread to the test section inlet in the ceiling region. Another interesting feature is the apparent existence of a large eddy just downstream of the flame tip. This structure is the probable mechanism for the transport of hot combustion products to the ceiling while the net flow continues downstream. We believe that such an eddy must occur in all situations where the fuel pan (and fire) extends across the full width of the test section. For comparison, we have also plotted the extent of the reverse flow ceiling jet for a much higher heat release rate (140 kW, test 2). For the increased fire intensity, the reverse flow region extended upstream of the test section inlet. Typical temperature isotherms from test 1 are given in Fig. 2. Also, as would be expected, the visible fire plume tended to be more vertically oriented as the incident ventilation velocity decreased in tests 3 and 4.

Received March 10, 1987.

*Technical Group Supervisor. Member AIAA.

†Technical Group Supervisor. Associate Fellow AIAA.

‡Member of Technical Staff; presently, Drexel University, Philadelphia, PA. Member AIAA.

§Member of Technical Staff. Member AIAA.

Figure 3 shows the flow patterns for the short fuel pan without the downstream barrier at the lowest crossflow velocity (test 5). The shorter fuel pan allows part of the core flow to be directed around the fire plume, resulting in a more vertical orientation to the fire plume. Heat fluxes at the ceiling were relatively high, even though the heat release rate was one-half of most other tests. The strong reverse flow ceiling jet transports large quantities of smoke into the plenum region, causing smoke to be recirculated back into the incoming flow. For these conditions, the smoke level drops almost to floor level within 30 s of ignition. In test 6, an even more vertical plume orientation was observed, and this was the only test where flames impinged on the ceiling. This result is presumably due to the downstream barrier, which simulates, for example, a row of seats in the cabin. The conditions of test 6 were utilized to study the response of ceiling panel materials to cabin fires, and more details can be found in Ref. 5.

Vertical hot film anemometer traverses were taken on the test section centerline 30 cm upstream of the fuel pan for three of the tests. A custom-designed, temperature-compensated probe capable of operating at gas temperatures up to approximately 600 K was used. Figure 4 provides a direct readout of the bridge output vs the vertical height while traversing at approximately 0.8 cm/s for test 2 at the highest ventilation flow rate. For comparison, the pretest cold flow profile is also plotted. Note the velocity increase in the lower two-thirds of the test section. This is due to the incoming ventilating flow being diverted into a smaller cross-sectional area and to the effect of the increased temperature and reduced gas density while the ventilation mass flow rate remains fixed. The velocity drops off sharply as the flow reversal point (as determined separately by flow direction indicators) is approached. In the region near the ceiling, there was a reverse flow ceiling jet where velocities were in the upstream direction. Since the hot film is insensitive to flow direction, the bridge output in the reverse flow region was reversed to coincide with the actual flow direction. Of note is that considerable soot collected on the probe, but because of film overheat there was no soot on the hot film itself.

Heat Transfer

The heat flux measurements at the ceiling are summarized in Table 1 for all test conditions at locations 1 and 2 (Fig. 2). In general, heat fluxes were highest for fires that were observed to have the most vertically oriented plumes. Also, the ratio of radiative to convective heat fluxes varied. At the highest crossflow velocities, the gases near the ceiling were cooler due to dilution and radiation dominated. As ventilation decreased and the fire plume became more vertically orientated, convective heat fluxes increased significantly.

Where significant convective heat transfer occurred, heat-transfer coefficients were in the range of 10–40 $W/m^2 \cdot K$ (Table 2). These were measured at locations downstream of the direct plume impingement regions or when the plume was significantly diluted by the ventilating flow. These values are near the range of corridor ceiling heat-transfer coefficients (20–60 $W/m^2 \cdot K$) measured by Quintiere et al.[6] just outside the door of a burning room. In test 6, the fire plume was nearly vertical, and it impinged directly on the ceiling, giving heat-transfer coefficients of 40–50 $W/m^2 \cdot K$. For this case, we have found that measured heat fluxes are approximately 50% below the fluxes predicted by the stagnation-point heat flux correlation used by You and Faeth.[7] Also, the ceiling heat fluxes in test 6 are in the range of the full-scale tests of Ref. 1.

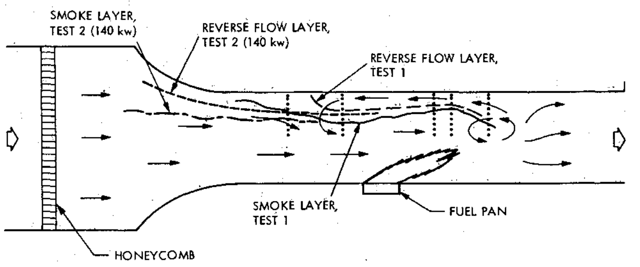

Fig. 1 Flowfield and smoke layer characteristics, test 1, 86 kW, $\bar{u}=0.74$ m/s.

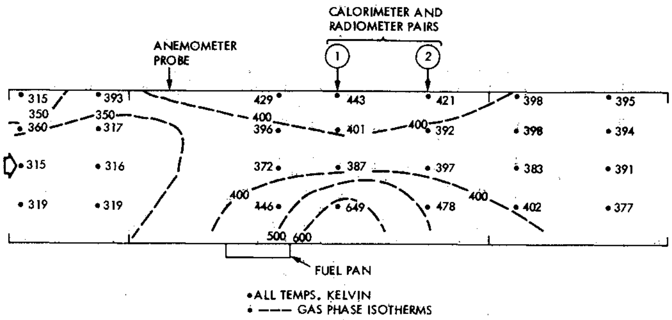

Fig. 2 Average gas phase temperatures for test 1, 86 kW.

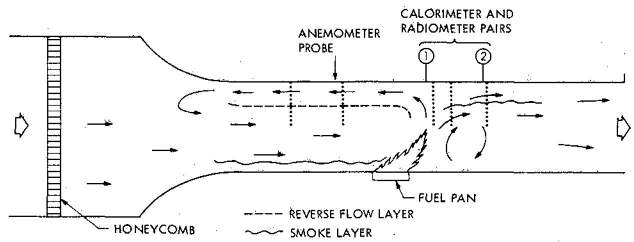

Fig. 3 Centerline flowfield and smoke layer characteristics, test 5, 53 kW, $\bar{u}=0.21$ m/s.

Table 1 Summary of test conditions

Test	Ventilation velocity, m/s	Fuel pan, cm	Heat release (kW)	Peak temp, K	Total heat flux, W/cm^2		Radiative heat flux, W/cm^2		Reverse flow layer depth, cm	Upstream smoke layer depth, cm
					1	2	1	2		
1	0.74	150	86	721	1.0	0.5	1.0	0.5	0	15
2	0.76	150	140	810	1.4	0.6	1.4	0.6	15	23
3	0.43	150	125	880	1.2	0.6	1.0	0.5	25	50
4	0.16	150	91	810	2.9	1.9	2.7	0.9	25	70
5	0.21	75	53	916	2.2	0.6	1.8	0.3	23	70
6	0.2	75	120	917	4.8	1.5	2.6	0.3	19	70

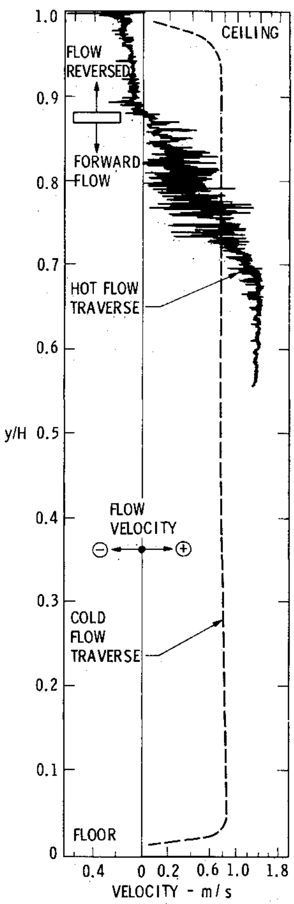

Fig. 4 Hot film anemometer traverse for test 2 conditions 30 cm upstream of fuel pan.

Table 2 Measured convective heat-transfer coefficients

Test conditions	Location 1	Location 2
Near vertical plume:		
test 6 (with fence)	40–50 $W/m^2 \cdot K$	10–20 $W/m^2 \cdot K$
Slight blowing:		
tests 4 and 5	5–12 $W/m^2 \cdot K$	12–40 $W/m^2 \cdot K$
Moderate blowing:		
test 3	10–12 $W/m^2 \cdot K$	~ 5 $W/m^2 \cdot K$
Highly blown:		
tests 1 and 2	Negligible	< 5 $W/m^2 \cdot K$

Conclusions

These results describe the detailed fire and flow characteristics in a one-third-scale aircraft cabin fire simulation with application to the evaluation of aircraft postcrash fire hazards. While reduced in scale, the experiment possesses key features that can be expected in the postcrash fire scenario (heat fluxes, fuel, and ventilation conditions). Assuming the same heat release rate per unit area of fuel, a fire of approximately 5 times the width (1.5 m) and heat release would possess a similar flame geometry in a full-scale cabin dimension. Flame height scales with pool width as $\sim W^{0.7}$ in this regime.[8] Of course, since momentum fluxes in the plume scale on flame height, the relative momentum fluxes of the ventilation and plume flow would be different in reduced and full-scale conditions. Nevertheless, these results show that wind-induced ventilation may significantly affect fire plume orientation, smoke transport, and heat fluxes and thus will affect subsequent fire spread and the immediate survivability of the passengers.

Acknowledgments

This work presents the results of research carried out by the Jet Propulsion Laboratory at the California Institute of Technology under contract with the National Aeronautics and Space Administration. The work was supported by the Transport Aircraft Aviation Safety Program at the Office of Aeronautics and Space Technology (OAST) of NASA. The authors acknowledge Dr. K.G. Harstad and Messrs. D.C. Schneider and J.J. Godley for their assistance in conducting the experiments.

References

[1]Sarkos, C.P., Hill, R.G., and Howell, W.D., "The Development and Application of a Full-Scale Wide-Body Test Article to Study the Behavior of Interior Materials During a Post Crash Fuel Fire," *Journal of Fire and Flammability*, Vol. 13, 1982, pp. 172–202.

[2]Parker, J.A. and Kourtides, D.A., "Fireworthiness of Transport Aircraft Interior Systems," AIAA Paper 81-1142, presented at the AIAA 16th Thermophysics Conference, Palo Alto, CA, June 23–25, 1981.

[3]Bankston, C.P. and Back, L.H., "Pool Fire-Ventilation Crossflow Experiments in a Simulated Aircraft Cabin Interior," *Journal of Aircraft*, Vol. 22, 1985, pp. 861–868.

[4]"Special Aviation Fire and Explosion Reduction (SAFER) Advisory Committee, Final Report, Vol. 1," FAA-ASF-80-4, Federal Aviation Administration, Office of Aviation Safety, Washington, DC, June 26, 1980.

[5]Bankston, C.P. and Back, L.H., "Measurements of the Response of Transport Aircraft Ceiling Panels to Fuel Fires," *Journal of Aircraft*, Vol. 23, 1986, pp. 207–212.

[6]Quintiere, J., McCaffrey, B.J., and Kashiwagi, T., "A Scaling Study of a Corridor Subject to a Room Fire," *Combustion Science and Technology*, Vol. 18, 1978, pp. 1–19.

[7]You, H.Z. and Faeth, G.M., "Ceiling Heat Transfer During Fire Plume and Fire Impingement," *Fire and Materials*, Vol. 3, 1979, pp. 140–147.

[8]Thomas, P.H., "The Size of Flames From Natural Fires," *Proceedings of the Ninth Symposium (International) on Combustion*, The Combustion Institute, Academic Press, New York, 1963, pp. 844–859.

Effect of a Round Airfoil Nose on Leading-Edge Suction

C. Edward Lan* and Ingchung Su†
University of Kansas, Lawrence, Kansas

Introduction

IT is well known that highly swept slender wings can develop a significant amount of vortex lift and drag due to lift. This vortex lift effect is, in turn, influenced by the leading-edge radius. To calculate the vortex lift effect through the suction analogy, it is necessary to predict the leading-edge suction. Most recent methods of the suction analogy to estimate the vortex force incorporate either the method of Carlson and Mack[1] or Kulfan[2,3] to determine the effect of rounded leading edges. Carlson and Mack developed an empirical method for estimation of attainable thrust and then equated the vortex force to the undeveloped thrust. On the other hand, Kulfan assumed that the vortex lift started to develop at an angle of attack (α_s) at which the nose drag is equal to the leading-edge suction.

Received Oct. 22, 1986; revision received Feb. 12, 1987.

*Professor, Department of Aerospace Engineering. Associate Fellow AIAA.

†Graduate Student.