SPREADING OF FIRES IN SHIPS' CABINS UNDER FORCED VENTILATION

A. K. AHMED and M. E. HORSLEY,
Department of Mechanical Engineering,
University of Portsmouth,
Portsmouth PO1 3DJ,
United Kingdom.

ABSTRACT

The investigation of fires spreading in ships' cabins by use of practically validated computer modelling is part of a major research programme and work is reported regularly [2 - 4]. The model here reported deals with the spread of fires in cabins under forced ventilation conditions. The essential problem is that of the movement and temperature histories of combustion products as buoyant, turbulent, circulating flows in a three dimensional enclosure. For the present work, the source fire was located on a cabin bed adjacent to the rear wall of the cabin.

A three dimensional steady state field model was developed, comparable to a fire in the near-steady operational plateau of its life. The related computational fluid dynamics (CFD) work being based on the FLUENT code. The above references give details of the CFD code, the experimental arrangements and the layout of the full-scale cabin used for the progressive validation. The present investigation included models of forced ventilation with differing extracting fan flow rates, equivalent to a fan speed range of 3 to 25 m/s.

In general, with an extraction speed up to 10 m/s, there was a modest effect compared to natural ventilation with the fan enhancing the entrained air flow condition around the fire plume. The ensuing deflection of the fire plume towards the rear wall had an almost negligible effect in changing the plume behaviour. As the fan flow rate increased up to 25 m/s, there was clearer evidence of asymmetric flow due to the increase in air entrained towards the fire. The deflection of the fire plume towards the rear wall was increased significantly as a result of the net pressure difference between the entrained air and the fire plume. This behaviour increased the effect of the rear wall in enhancing the fire plume, the spread over the wall giving a larger area over which to entrain air to burn the residual fuel volatiles.

INTRODUCTION

Fires in cabins often start small, such as with a cigarette in a waste paper bin or a cabin bed. Once started, the fire may spread to other combustibles depending upon their proximity to the ignition source. As the fire grows in size, a hot gas layer containing smoke particles (soot) and other gaseous products of combustion may then flash over to other compartments. Several studies have been conducted to determine the temperature and smoke concentration of thermal plumes produced from different fire scenarios.

Under natural flow movements, fires in small compartments such as rooms or cabins have been examined in enclosures with an open door or window [1, 2]. The temperatures throughout an enclosure in which a fire is burning are affected by the amount and the point of entry of air in the compartment. One of the most difficult decisions during fire fighting is when and when not to ventilate, as forced ventilation in the fire compartment can have a number of effects. It can increase the amount of smoke extracted from the compartment through the ventilation system, hence improving visibility and thus aiding escape from the fire area, fire location and fire fighting. [3, 5] However, enhancement of the fire may result if the fire is ventilation controlled, hence burning more violently and possibly spreading to adjacent areas.

In the present forced ventilation work, fire spreading from a cabin bed adjacent to the rear wall and its effect over that rear wall of an open-door cabin were investigated theoretically using the FLUENT (Fluent Europe Ltd) Computational Fluid Dynamics code. The computational models were validated and refined progressively against real fire tests conducted in a full-size part model of a steel ship structure. The chosen site for the forced ventilation fan was at the centre of the cabin's ceiling, Fig 1, for removing most of the smoke and combustion products. Fan speeds were adjustable in the range up to 25 m/s.

MODELLING TECHNIQUE

In fire events, movements of the combustion products of hot gases and smoke released from the fire plume are dominated by buoyancy effects. This gives rise to a large scale turbulent motion creating a non-uniform buoyancy force which in turn controls the rate of diffusion of mass and momentum and thus the mixing process.

The modelling of a real spreading fire which is likely to be unsteady in its behaviour has not been attempted in the present stage of the work. However, analysing the characteristics of a steady fire affecting the flow movements within the cabin should yield useful information, as many fires have a near-steady operational plateau for much of their life.

The experimental arrangement was as described in Ref 2. In essence, a full-size steel cabin 3.7 m long, 2.05 m high and 2.6 m wide was used. A bunk bed, in steel plate for durability, fitted across the rear wall and a petrol pool fire simulated a bunk fire for repeatability. Primary temperature measurements were carried out using thermocouples along the centre plane of both the cabin door and the rear wall. Adjacent to the thermocouples, McCaffrey probes were used to measure the flow velocities along the door. An array of firelighters (very combustible proprietary products for aiding domestic fire lighting, camp fire lighting and so on) was fixed across the rear wall at the height of each thermocouple, Fig 9, to indicate the flame spread clearly. This simple technique, wherein the spreading flame ignited the firelighters, proved a most effective mapping method. Photographs, e.g. Fig 10, recorded the spread.

In the Computational Fluid Mechanics (CFD) work, the investigation was carried out by designing a three dimensional steady state field model using the FLUENT (Fluent Europe Ltd) code. The field modelling involved solving the set of three dimensional partial differential equations which govern the phenomena of interest, using the finite volume method. In summary, the set of equations consists essentially of the continuity equation, the three space directions momentum equations and the equation for conservation of energy, buoyancy modified. In these equations, the independent variables used are the width, height and length of the cabin. The dependent variables for solution are the gas velocities, the pressure, the gas enthalpy and the turbulence kinetic energy and its dissipation rate. Some empirical constants [6] were involved, the solution procedure was based on the SIMPLE algorithm [7] and a first order accurate power law discretisation scheme was used.

The cabin model, based on the above-noted real cabin dimensions, was designed using a non-uniform grid of 37 x 18 x 15 cells, Fig 1. For all the cabin walls, the no-slip condition was applied to the velocity and an isothermal wall temperature of 283 K was used. The fire was simulated by heat flux to give an output of 50 kW. Whilst comment may be made about the isothermal wall choice and the level of heat flux, it is emphasised that the work is part of a major programme of the progressive building of powerful, reliable validated models. The work reported here is no therefore intended to be a report of a study of a specific incident or circumstance.

RESULTS AND DISCUSSIONS

The pattern of flow movements within a cabin fitted with an extraction fan varied with fan flow rate [3]. For the fire plume conditions as in the present case, its deflecting and spreading would be affected by the fire scenario, such as the restriction of free air entrainment from an adjacent cabin wall or ceiling or the influence of momentum of flow movement created by forced ventilation. Generally speaking, with a cross draught in an open area, flame will be deflected any air movement to an extent dependent upon wind velocity. Air movements tends to enhance the rate of entrainment of air into a fire plume, which is likely to promote combustion.

For a fire near to a compartment boundary, there will be restriction of free air entrainment due to the asymmetry of the flow towards the fire plume. Thus the temperature of the plume will decrease less rapidly with height as the rate of mixing with cold ambient air will be less than for the unbounded case. For the same reason, flame extension at a non-combustible wall will occur as the flame has to increase in size to give a large enough area through which to entrain air to burn the fuel volatiles. Clear asymmetric entrainment of the flow towards the fire plume would cause the flame to be deflected towards the restricting wall as a result of the net directional flow of the air into the plume. This effect will enhance upward flame spread on vertical (and sloping) combustible surfaces as well as encouraging fire spread to vertical surfaces from adjacent flames.

Where extension of the fire plume is confined by a ceiling, the hot gases will be deflected as a horizontal ceiling jet, thus providing the momentum by which combustion products are carried away. In the present fire tests, the fire plume was under a combination of effects from these phenomena and their effects varied according to the fan extraction rate.

For all CFD cases considered, the solution domain consisted of about 25000 cells and the number of sweeps required to achieve convergence varied between 4000 and 5000. All figures presented here for plotting results were viewed through the vertical centre plane, door to rear wall, of the cabin and the fire, where the effects of the cabin walls were minimised and where the flow can be assumed to be substantially two dimensional, Fig 1. The predicted results were validated progressively and against real fire tests.

The discrepancies between the ultimate computations and the practical measurements were within practical measurement error. In the CFD model, the reverse velocities associated with the large vortices and recirculation zones were over predicted. That may be attributed to numerical diffusion and the shortcoming of the k-e model in dealing with recirculating flows. However, considering the small magnitudes of the flow in each position, the accuracy of the predictions is satisfactory overall, especially in terms of the mean values. Thus the computational model may be claimed to provide a good description of the flow dynamics.

For a natural draught (no fan) fire scenario [2], the general pattern of flow movement within a cabin is as in Fig 2A, indicating that the highest velocity was reached in the region above the fire source due to the strong buoyancy effect. As the fire source was sited near to the rear wall, the fire plume was confined by that wall. Thus the hot gases extended horizontally towards the free sides with higher momentum than the free ceiling jet. The combustion products were carried away towards the cabin door as the only exit. The stratification of the smoke layers represented by the contours of the hot gases, Fig 2B, indicates that there was little mixing between the hot gases and the cold air beneath it.

At the fire region and from the CFD results, the momentum of the air drawn towards the fire source was enough to create almost axisymmetric entrainment around the fire, Figs 3 & 5.. That could also be confirmed by the peripheral area of the flame plotted as temperature contours, Figs 6 & 7. It had an almost circular cross section and was not attached to the rear wall. It was similar to the cross section of the fire source, located about 0.5 m from the rear wall. As the air flow towards the flame was almost axisymmetric, the rear wall had an almost negligible effect on the flame characteristics and the flame stayed straight, Fig 8.

In the experimental work, that conclusion was confirmed first by noting that the highest temperature of the rear wall was around 1 m above the bed whilst further up the wall the temperature decreased progressively, Fig 9, showing that the flame was not deflected towards the wall. Secondly, only the firelighters at positions 1 - 3, within the 1 m height, were ignited, showing that the flame height was not changes significantly due to upward spread caused by the presence of the rear wall, Fig 10.

With an extracting fan in use, as the fan flow rate increased, the region adjacent to the fan was at a progressively lower pressure compared to the rest of the ceiling area [3] and the buoyant plume moved towards that region with progressively higher momentum, Fig 5. Keeping a mass balance within the cabin, the mass of hot gases extracted by the fan was replaced by cold air entering the cabin. Thus the momentum of of air towards the fire source increased according to the mass flow rate of air entering the cabin, Fig 11 and the general pattern of flow movement in the cabin also changed.

In addition to that main mechanism, the momentum of the air flowing towards the fire source was affected by both the ceiling and the rear wall. In order to show the proportions and paths of the air flow, ten CFD streaklines were traced from the door, representing the mass flow entering or leaving the cabin.

Using the natural ventilation scenario as a reference, the proportions and paths there indicated five streaklines of air entering and five of hot gases leaving through the door, the only natural ventilation opening, Fig 4. Running the fan with speeds of 6 m/s and 10 m/s, the proportions and paths of streaklines representing air entry through the door increased to six and eight respectively. That indicated the increase in momentum which led to a slight asymmetric flow condition around the fire source, Fig 3. As a result, the net pressure difference between the cold air intersecting with the flame and the fire plume would deflect the plume towards the wall, which may be confirmed by the cross section of the plume at different positions. As an example, for a fan speed of 10 m/s compare the peripheral area of the flame at positions 0.5 and 1.3 m above the bed, Figs 6 &7. The fire plume at the base had enough strength to overcome the pressure increase of the entrained air. In the upper part of the plume, the pressure difference was significant and the plume deflected towards the wall, affecting its cross section by the spread of fire over the rear wall.

Increasing the fan speed progressively to 25 m/s, the air flow towards the plume similarly increased. As an example, at 18 m/s all the streaklines were of air entering through the door. door and the fire plume deflected further towards the rear wall. There were parallel significant changes in the plume cross section, Figs 6 & 7, confirming the deflection and spreading of the fire towards the wall. Thus the fire plume became significantly affected by the rear wall which enhanced the flame spread both upward and over the rear wall.

In the experimental work, those conclusions were confirmed. The evidence was first the increase of temperature of the rear wall facing the top part of the flame despite the continuous cooling effect occurring around the fire plume,Fig 9; second the firelighters in positions 1 -5 ignited, confirming that the flame height increased due to the upward flame spread, Fig 10.

CONCLUSIONS

Various models of forced (fan driven) ventilation fire scenarios within a steel cabin have been developed by CFD and have been validated progressively on a full scale part model of a ship. The models covered fan entry (that is, gas removal) speeds up to 25 m/s. A ventilation controlled fire may spread more violently over the cabin due to changes in its behaviour. That could be enhanced by factors such as an increase in fan flow rate or the effects of bounding surfaces.

With a moderate fan speed, say up to 10 m/s the top part of the fire plume was deflected slightly towards the rear wall and the fan had a modest effect in changing the behaviour of the the plume. Thus the main characteristics of the fire plume, including height and strength, were not significantly affected by the presence of the rear wall.

For higher fan speeds, recorded up to 25 m/s, the plume suffered significant deflection towards the rear wall. As a result, it spread over the wall and important characteristics were changed significantly. The consequences of forced ventilation during a cabin fire depend upon thus upon the rate of ventilation and fire spread over an adjacent wall may be an important if not disastrous consequence.

The good agreement between the CFD model and the experimental results of this case and the previous cases [2-4] highlights its value as a fire analysis tool. As the field model is derived from the fundamental flow equations, it is an ideal design tool in developing fire fighting strategies.

The authors are indebted to the U.K. Marine Technology Directorate Ltd and the U.K. Ministry of Defence for their support of this work.

REFERENCES

1 Broyt T.W et al; The use of a computational method to assess the safety and quality of ventilation in industrial buildings; Conf Heat Transfer and Fluid Flow; I.Mech.E., London 1983.

2 Ahmed A.K. & Horsley M.E.; Mathematical modelling of fires in ships cabins; Int Conf Materials and Design against Fire; I.Mech.E., London 1992.

3 Ahmed A.K. & Horsley M.E.; Mathematical modelling of fires in ships cabins with forced ventilation; Interflam 93; Oxford 1993.

4 Ahmed A.K. & Horsley M.E.; Window effects on fires in ships cabins; forthcoming.

5 Beyler C.; Analysis of compartment fires with overhead forced ventilation; Proc 3rd Int Symp Fire Safety Science; Edinburgh, 1991.

6 Launder B.E. & Spalding D.B.; Numerical computation of flows; Computer Methods in Applied Mechanics and Engineering; North Holland Publishing Co., 1974

7 Patankar S.V.; Numerical Heat Transfer and Fluid Flow; Hemisphere, 1980.

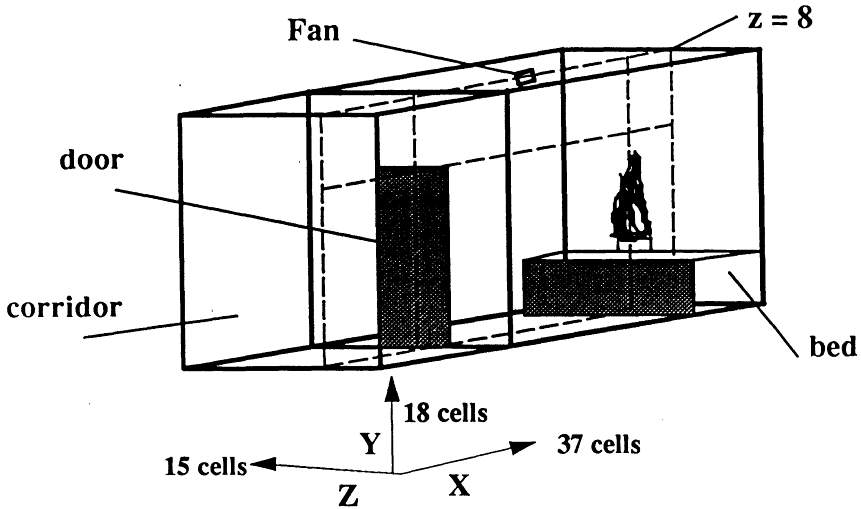


Fig 1: Layout of the cabin

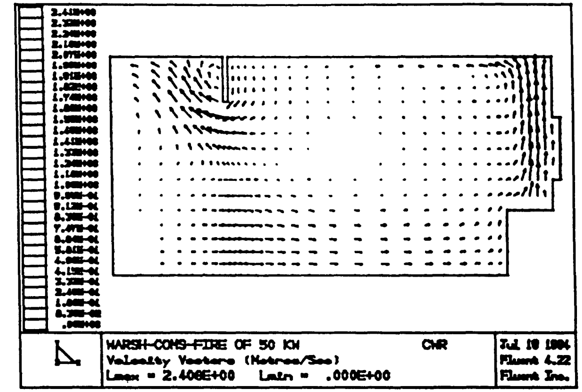


Fig 2A: Velocity vectors along the cabin without fan

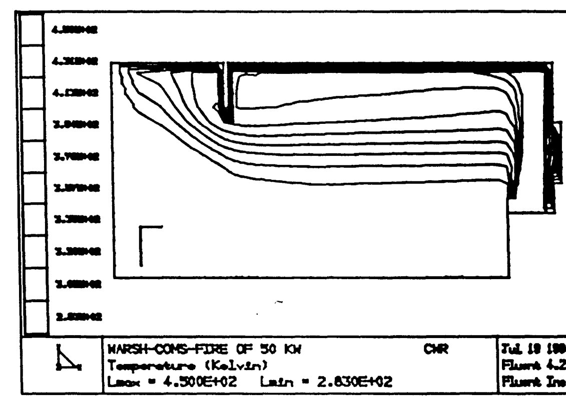


Fig 2B: Temperature contours along the cabin without fan

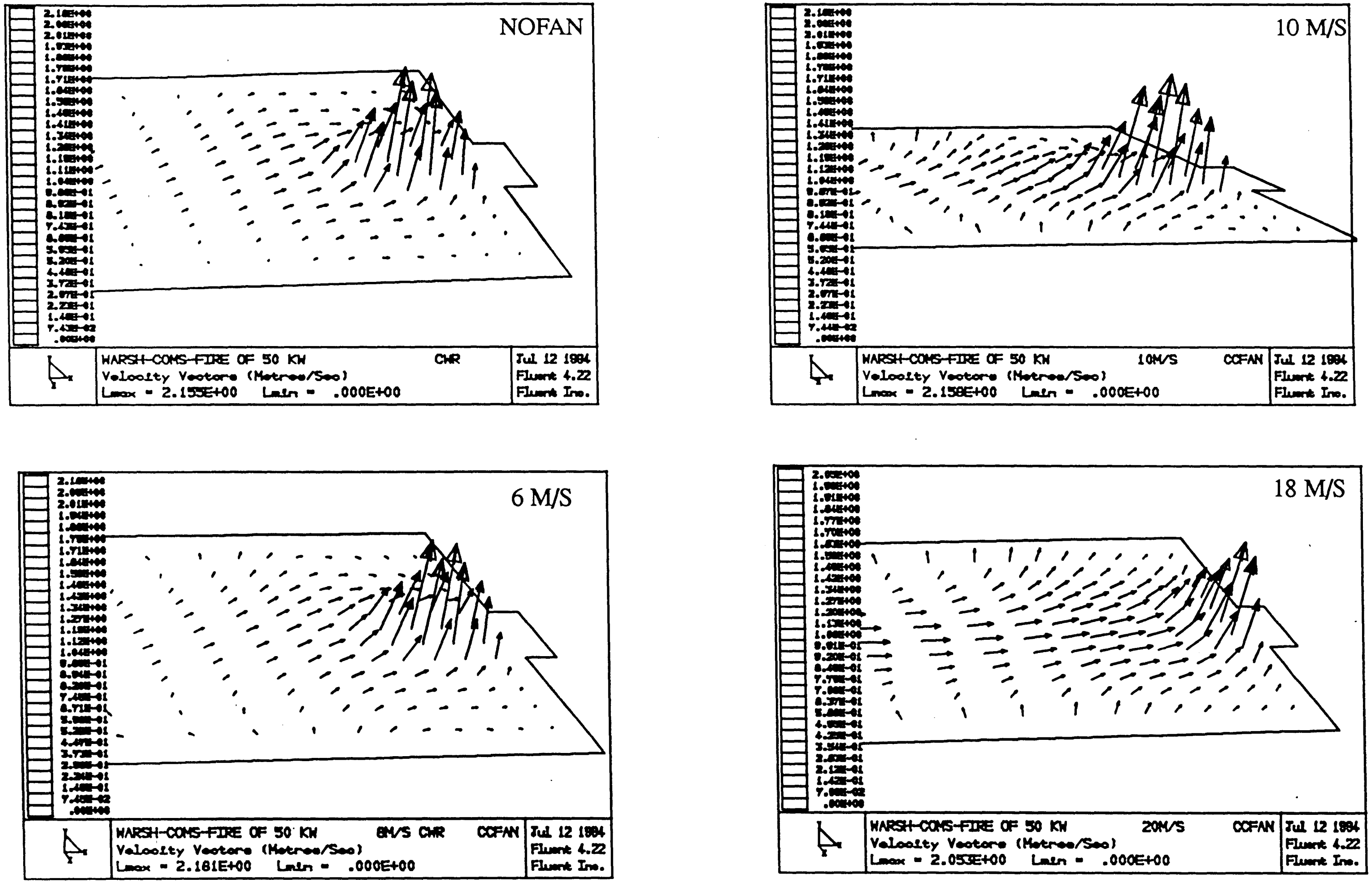


Fig 3: Velocity vectors across the cabin, different fan speeds at 0.5 m above the deb

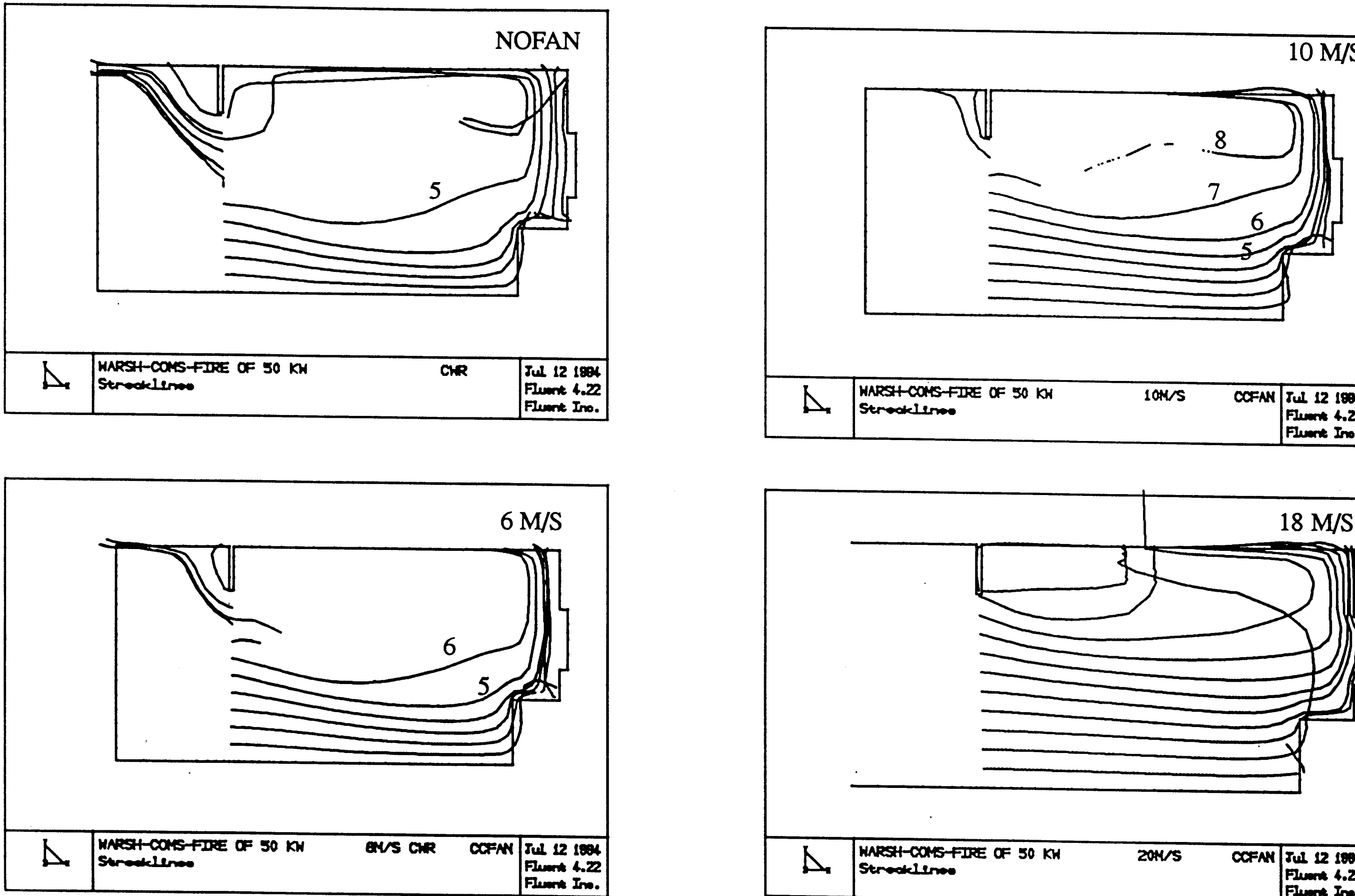


Fig 4: Streaklines along the cabin, different fan speeds plotted from the door position

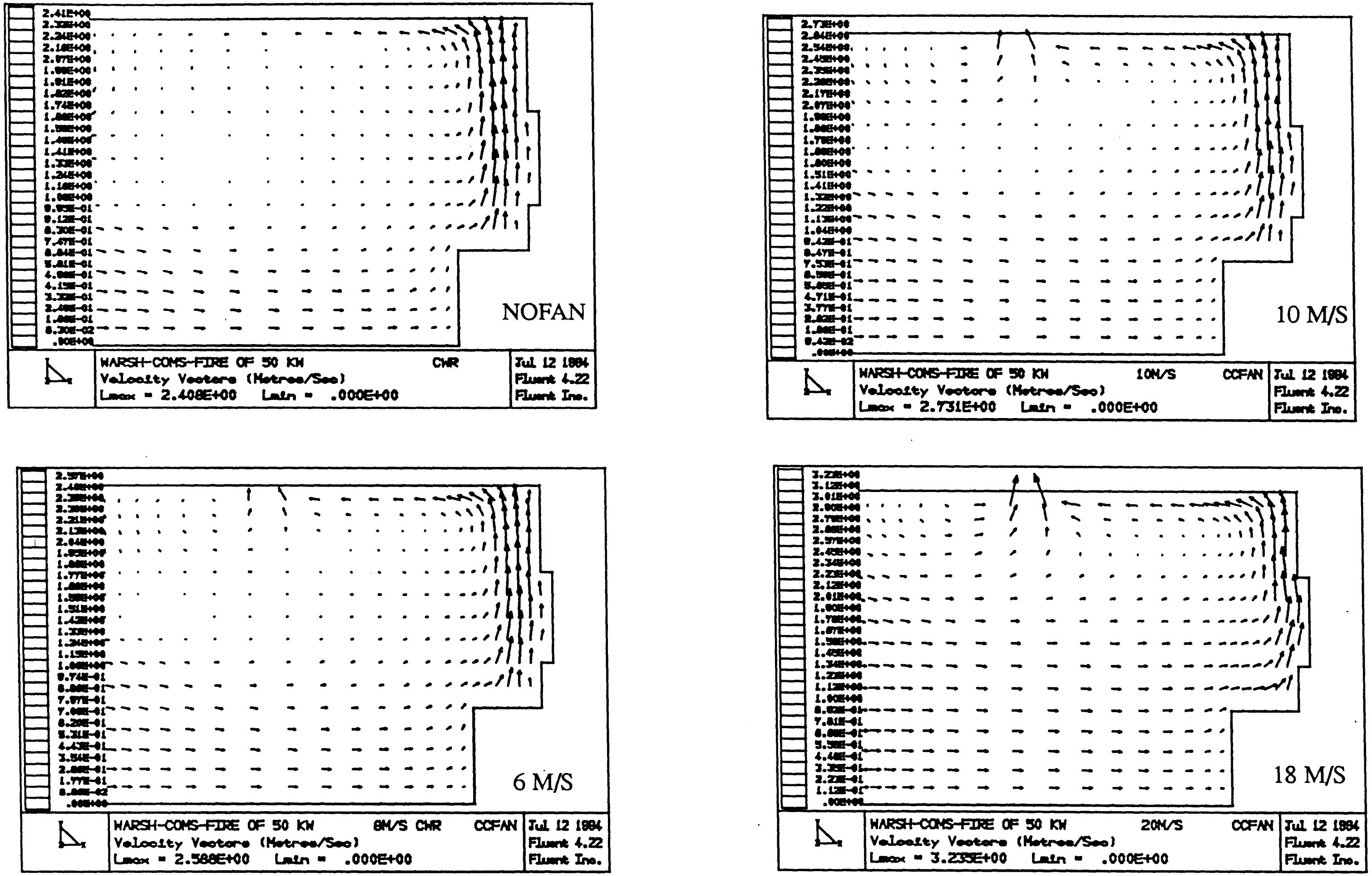


Fig 5: Velocity vectors along the cabin, different fan speeds at central plane of the cabin

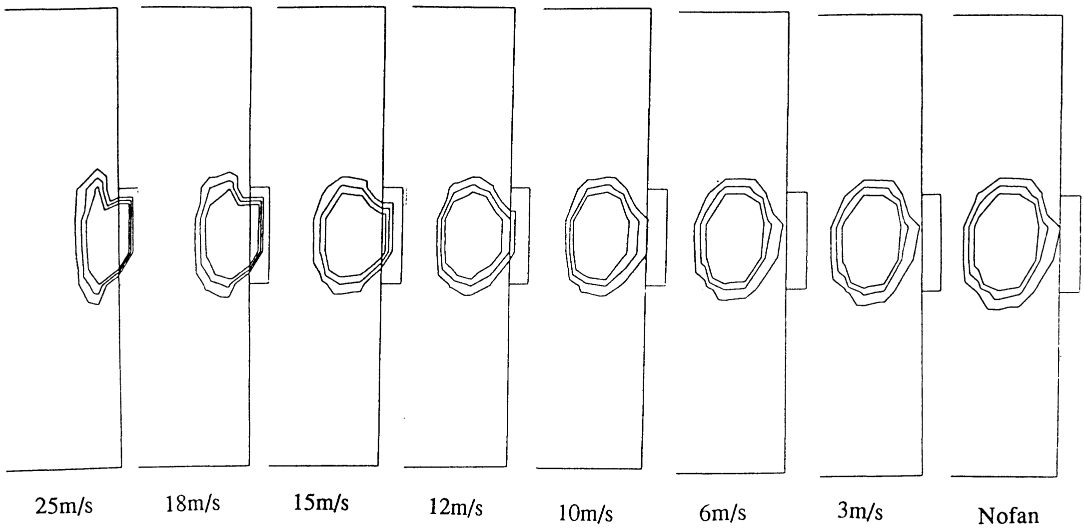


Fig 6 : **107°C - 177°C contours of the fire plume at 0.5 m above the bed**

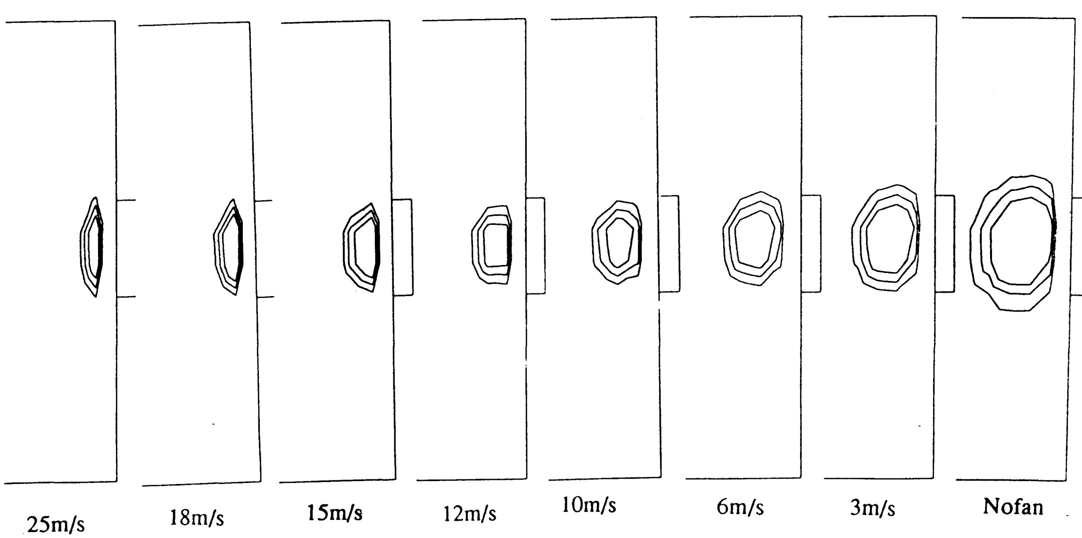


Fig 7 : **142°C - 177°C contours of the fire plume at 1.3 m above the bed**

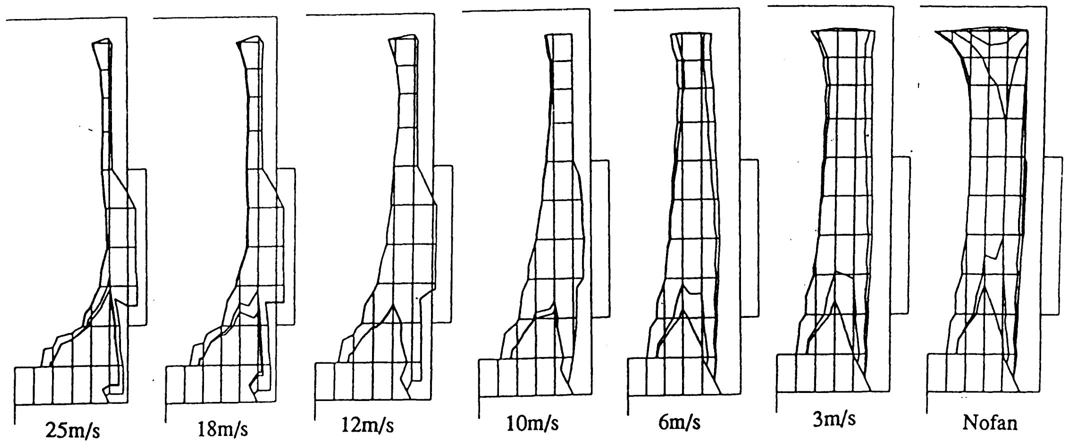


Fig 8 : 177°C contours of the fire plume with different fan speed

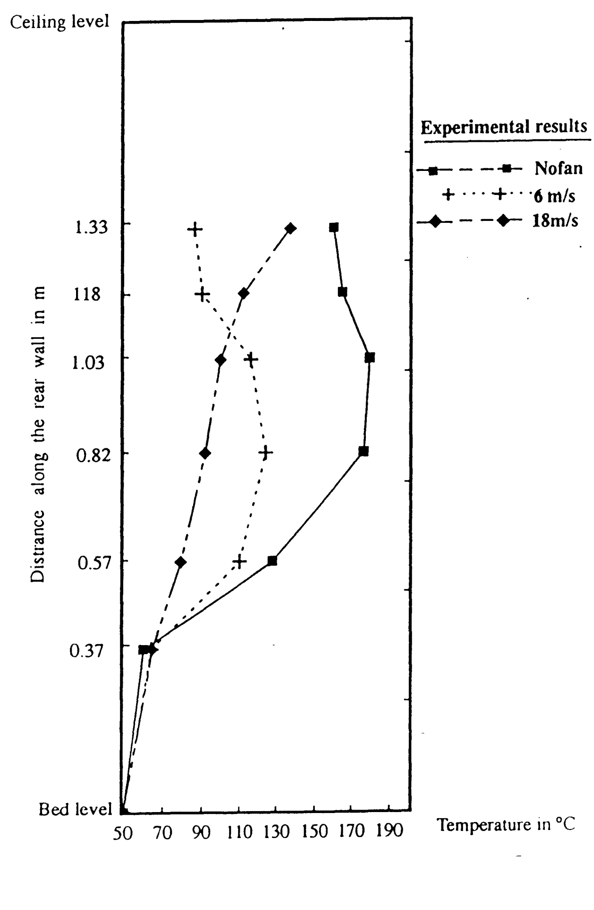
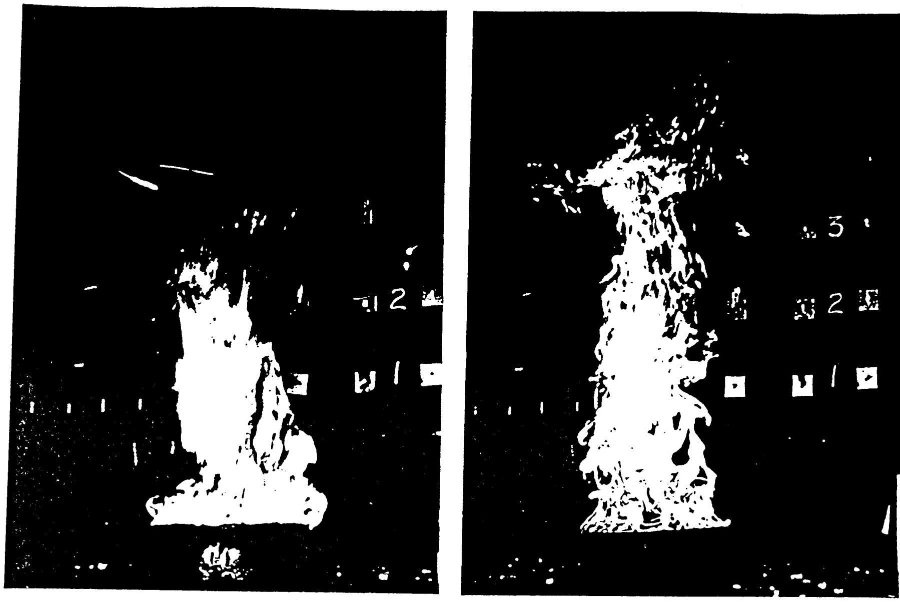


Fig 9: Temperatures along rear wall of the cabin



Fire scenario without fan

Fire scenario with fan speed of 18 m/s

Fig 10 : Position of fire lighters along the rear wall

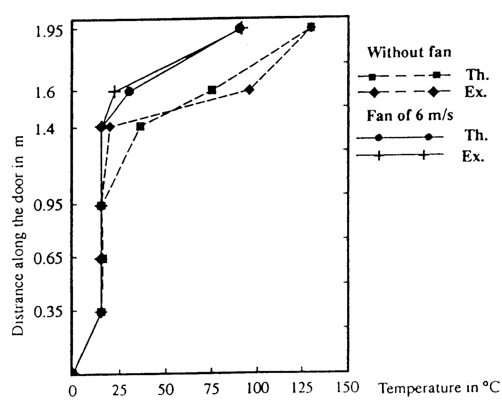


Fig 11A : Temperature distribution along the door

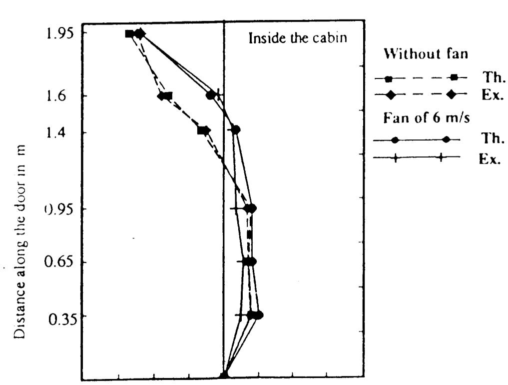


Fig 11B : Velocity distribution along the door