

The Simulation of Engine Casing Fires : Computational Modelling and Experiment

JB Moss, PA Rubini, V Sanderson, CD Stewart
School of Engineering, Cranfield University, UK
and
AJ Mullender, Rolls Royce plc.

Introduction

The fire safe design of accessory zone layout and ventilation in aircraft engines has traditionally been founded on a combination of past experience and standardised fire testing of individual components. Through design necessity the annular region between the outer wall of the engine nacelle and the fan casing on a modern high by-pass ratio gas turbine engine is often highly congested. This volume will typically contain a range of engine accessories and equipment, together with systems handling the cooling and supply of lubricating oil and fuel. The combination of rotating machinery, for example, in the gearbox and starter systems, with electrical equipment and hydrocarbon fuels therefore introduces a potential fire hazard into a relatively inaccessible domain of particular geometrical complexity. Whilst the complete zone is actively ventilated in order to reduce the build-up of any flammable vapour and to control ambient temperature levels, such flows will also exert a strong influence over the development of any fires that might arise from the loss of fuel or lubricant containment. The introduction of new materials, or re-sizing and relocation of accessories, together with enhanced use of electronic control systems and the accompanying tighter limits on the ventilation environment will tend to undermine any existing experience base, and emphasise the need to develop robust and flexible simulation tools. The combination of poorly controlled ventilation flows and complex physical obstacles in close proximity to each other makes the prediction of fire growth and of the accompanying rates of component heat transfer especially difficult. It is on these aspects of numerical simulation, and their experimental evaluation, that the present paper will focus.

Standard fire safety assessments that are based upon simplifying procedures - such as those of whole zone heating to a prescribed temperature level or burner tests of the survivability of individual components - can be used to set conservative thresholds. Such coarse design constraints are increasingly incompatible with the introduction of finer thermal tolerances on particular electronic components or the introduction of novel materials. Increased reliance on the bulk zone ventilation to provide a suitable operating environment for a range of accessories also demands greater flexibility in their positioning. Such considerations can also contribute to improvements in their ease of assembly and subsequent maintenance.

CFD is uniquely well-suited to the analysis of the interactions required by a more responsive approach to fire safe design. Although the methodology is widely employed in

the aerospace field to assist in the development and implementation of specific technologies, the simulation of accidental occurrences poses a number of different challenges, especially in relation to validation. Design point component performance provides a clearly defined focus for the application of CFD and permits a measure of model calibration when applied to a complex component like the combustion chamber. Such considerations are generally absent in the analysis of fire where, for example, neither the level of fuel flow rate nor the point of delivery can be readily prescribed.

The paper describes the initial development of a CFD-based approach to the analysis of fan casing accessory layout from a fire engineering perspective. The study focuses on a fairly generic casing configuration, incorporating geometrical simplifications that also admit parallel experimental investigation and opportunities for model validation. We describe first the layout of the experimental casing fire rig and identify some of the defining flowfield features to be reproduced in any satisfactory numerical simulations.

Experimental Rig

The basic features of the rig are illustrated in Figure 1. An annular fire zone is created between two concentric cylinders (of diameters 1.2m and 1.5m) of length 0.84m. In order to permit repeated fire experiments without damage and distortion of the zone envelope, both cylinders are water-cooled. The cooling water is pumped and recirculated, but the provision of a free surface, exposed to the atmosphere, permits operation at coolant temperatures up to levels approaching the boiling point of water.

Within the annular space created between the cylinders a range of obstacles of varying size and shape has then been fitted, representing different engine accessories, for example oil coolers, gearbox and control units. These are manufactured in stainless steel and mounted on tabs attached to the inner drum, see Figure 1. The annulus is often particularly congested towards bottom-dead-centre (bdc) where the gearbox, starter motor and air starter duct are typically located. Ventilation air is introduced into the casing through a bifurcated inlet mounted at the top of the nacelle. This produces two inlet jets, nominally of equal strength, which feed the two halves of the casing volume. The air exhausts through a rectangular opening near the base of the casing – off-set from bdc by 10° and having an open area of 350 cm^2 . The outer drum is ribbed – as would be the situation in practice with a stiffened thin-walled structure.

Although a number of different types of fire source have been examined in this rig, including kerosine sprays ignited both locally and remote from the fuel source, we shall largely focus in this paper on pool fire scenarios. In these it has been assumed that liquid kerosine has pooled at the base of the zone, following a fuel supply leak and burns as a turbulent diffusion flame. No attempt has been made in these studies to simulate any particular ignition process. The fire which then develops from the kerosine spill is fuelled by evaporative mass loss, generated by both radiative heat transfer from the highly luminous burning plume and by convective heat transfer from the combustion products that are recirculating within the zone, largely driven by the ventilation flow. Pool fires arise in many practical scenarios and have therefore attracted considerable research

attention within the fire engineering community [1]. The confined casing fire configuration is quite distinctive, however, displaying features that are not encountered in other applications where the effects of either buoyancy or forced convection predominate. The ventilation flow and that of the fire plume are, in principle, opposed but, since the internal accessory layouts are not symmetrical, plume development occurs in only half of the rig and may therefore be assisted by the ventilation flow spilling over from the opposite half. We shall discuss such issues in more detail in subsequent sections.

The temperature field within the rig has been extensively mapped using arrays of fixed and traversable thermocouple rakes. These are inserted through the rig end-plates, parallel to the axis of the cylinders creating the annular space. These end-plates are also fitted with circular windows (see Figure 1), spaced around the annulus at approximately 30° intervals. Heat flux measurements have also been made at selected locations within the rig by measuring the transient thermal response of thin metal discs mounted on the wall of the outer drum.

Fire Plume / Ventilation Flow Interaction

The introduction of high momentum air jets into the large annular space, even in the absence of fire, creates a particularly complex, maldistributed flow field. The circulating flows generated by the bifurcated inlet do not simply exit the zone after passing through 180° and reaching bdc. Substantial spillage occurs and counter-current flows become segregated with position along the annulus axis, depending upon the extent to which they initially penetrate from the front to the rear of the rig and on any obstacles they encounter in the form of the internal accessories. Figure 2 illustrates typical isothermal flow simulations of a minimally congested zone, based on observations from a water analogy experiment performed with a perspex model.

The two jets quite clearly do not separate the zone into weakly interacting left and right halves. A significant part of the clockwise directed inlet jet (identified as IP in Figure 2) is deflected to the front of the rig and completes a full circuit of the zone. Such behaviour has important implications for the plume trajectory in the event of fire and the nature of fire attack on particular zone accessories.

In general, with the distribution of accessories illustrated in Figure 2 the plume from a pool fire at the base of the zone is observed to propagate in a clockwise direction. Figure 3 illustrates the typical plume development as revealed by the measured temperature distributions in three radial planes at 60° , 120° and 150° , defined from the rig bdc. A kerosine spill of 0.1 m^2 might be expected to sustain an open pool fire with an output of approximately 150kW and a fire plume of such a size is sufficient to extend beyond the rig top-dead-centre. Even when exposed to a quite modest zone ventilation rate of 5 volume changes per minute, the plume is rapidly diluted, however. The temperature of the mixed plume, incident on the object representative of say an ECU accessory, does not then exceed 100°C . Figure 4 illustrates the distribution of measured gas temperature across the annulus, immediately above and below the unit, indicating the comparatively uniform nature of the temperature field. Buoyant plumes embedded in large volumes are

often unstable, however, and the plume trajectory is observed to periodically reverse direction. The transit time to this unit in the anti-clockwise direction is now greatly reduced. Dilution mixing is less effective and the temperature environment of the unit is now more severe. The plume temperatures incident on the lower faces of the unit approach 300⁰C as indicated in Figure 5. More active ventilation air management strategies are therefore required if such instability is to be eliminated and worst case thermal protection provision is to be avoided.

Key elements in the numerical simulation of these scenarios are evidently the plume trajectory, and its interaction with the forced ventilation, the mixed plume temperature levels and the associated heat fluxes to critical accessories.

Computational Simulation

The numerical simulations were carried out using the CFD code **SOFIE** (Simulation of Fires in Enclosures) [2], specifically developed at Cranfield for compartment fire predictions. **SOFIE** is based upon a finite volume procedure utilising an underlying general non-orthogonal coordinate system with co-located velocities, momentum smoothing and a pressure correction algorithm. Dependent variable interpolation is achieved using either a first order or a TVD based second order accurate scheme. Turbulent closure is effected through a two equation, k-ε model incorporating buoyancy modifications. Combustion is simulated using either an Eddy-breakup model or a prescribed pdf laminar flamelet based model. Thermal radiation is simulated using a deterministic ray tracing technique with combustion product optical properties described by a weighted sum of grey gases model [3].

Model Evaluation

Pool fires continue to be a challenging subject for detailed investigations using numerical techniques. They exhibit unique physical phenomena that current CFD modelling strategies struggle to represent, specifically the entrainment of fresh air into buoyancy driven plumes, with relatively large-scale temporal fluctuations superimposed upon the smaller scales of turbulence. Therefore, whilst the merits of the alternative combustion sub-models are well known for high intensity convection driven applications, typically internal combustion devices, significantly less development and validation has taken place in fires, especially for constrained buoyant fire plumes of the kind that might occur in an engine nacelle.

Given the earlier observations regarding flame development and impingement, the particular focus of the present investigation is on the relative length and local temperature of the fire plume. Once confidence has been demonstrated in predicting local temperatures then a more quantitative evaluation of radiative and convective heat fluxes to the individual components within the nacelle may be carried out.

Figure 6 presents the predicted flow streamlines within the rig in the event of a pool fire and illustrates the considerable interaction between the complex fluid dynamics of the designed ventilation system and the fire plume. Whilst the level of congestion is here quite modest and the rig represents only a generic geometrical configuration, the complexity of the flow is clearly visible. In particular, elements of the ventilation flow clearly do not pass immediately to the exhaust but rather circulate one or more times before finally escaping. Even without considering the fine detail of the fire simulation, the potential of CFD for optimising the ventilation system is evident.

Figure 7 provides a visualization of the fire plume through an iso-surface of mixture fraction (which may be considered to be proportional to the local air fuel ratio). The plume is seen to be deflected in a clockwise direction and to wrap around the first internal obstacle (representing an oil cooler). At the flow rates and heat release rate simulated the plume does not reach top dead center of the rig. Figure 8 presents a more detailed visualization of the flow streamlines in the vicinity of the plume and the fire source. Here small scale geometric details are seen to be important, specifically the proportion of unburnt fuel that 'short circuits' to the exhaust, by flowing below the obstruction of the gear box.

Figure 9 presents a quantitative representation of the local gas temperature on a number of constant radial planes through the rig. The combustion model predicts peak gas temperatures in the region of 1500K close to the fuel source, however, further downstream within the plume the temperature has been reduced by several hundred Kelvin through entrainment of air into the plume and dilution. Comparison with the experimental data, for example Figure 3, indicates that the numerical predictions are to within approximately 100K of the measured values. To place the present results in context, in an unconstrained fire plume, where the adiabatic flame temperatures must be of order 1800K, the typical quoted averaged value for the peak temperature at the tip of the flame is of order 900K.

Previous studies of free plumes [4], employing $k-\epsilon$ based modelling, have implied that the rate of entrainment into the plume is generally under-predicted. In consequence, the local gas temperature is over-predicted, regardless of the capability of the combustion model to represent the formation of dissociated combustion products such as CO. Within the geometry of the nacelle rig, two further factors must be taken into account - the effect of constraining the fire plume, which promotes local flow acceleration and restricts air entrainment, and the increased proportion of fuel not directly participating in the combustion process within the rig.

Finally, Figure 10 presents a surface plot of the local convective and radiative heat fluxes to the interior surface of the nacelle. Unsurprisingly the peak incident fluxes are in the region of the fire plume around the gearbox and oil cooler, with a maximum value of

approximately 30 kW/m². Given the uncertainty over the local gas temperature alluded to earlier, the absolute values must be interpreted cautiously, however the indicated distributions are not expected to change significantly.

Conclusions

The interaction between the ventilation flow in a representative engine nacelle compartment and the pool fire that may result from the loss of containment of fuel or lubricant has been simulated both experimentally and computationally. The trajectory of the fire plume is revealed to be highly dependent upon the layout of accessories in the zone and this, in turn, affects the local heat transfer environment. Temperature measurements on obstacles representing different accessories within the zone reveal substantial temperature differences depending on the fire plume trajectory.

Numerical simulations demonstrate the potential for CFD to simulate fire events within gas turbine engine nacelles. Key flowfield features are readily reproduced and provide opportunities for the tailored design of the overall ventilation flow and the siting of the various accessories. The general precision of the CFD simulations in respect of surface heat flux in the presence of fire remains to be determined, together with any implications regarding the development of more advanced combustion sub-models for constrained buoyant fires and different hydrocarbon fuels.

Acknowledgements

The research reported has been undertaken with financial support from Rolls Royce plc and the UK Department for Trade and Industry. Their support and interest is gratefully acknowledged. The water analogy study, underlying Figure 2, was performed by Mr D Binks.

References

1. McCaffrey, B (1988) Flame Height, in the *Handbook of Fire Protection Engineering*, SFPE.
2. Lewis, MJ, Moss, JB and Rubini, PA (1997) CFD Modelling of Combustion and Heat Transfer in Compartment Fires, Fifth Int Symp on Fire Safety Science, IAFSS, p.463-474.
3. Bressloff, NW, Moss, JB and Rubini, PA (1996) CFD Prediction of Coupled Radiation Heat Transfer and Soot Production in Turbulent Flames, Twenty-Sixth Int Symp on Combustion, The Combustion Institute, p.2379-2386.
4. J. Worthy, V. Sanderson, P.A. Rubini. "A comparison of modified k-ε turbulence models for buoyant plumes", Numerical Heat Transfer, Part B: Fundamentals, Vol. 39, No. 2, pp.151-166,2001

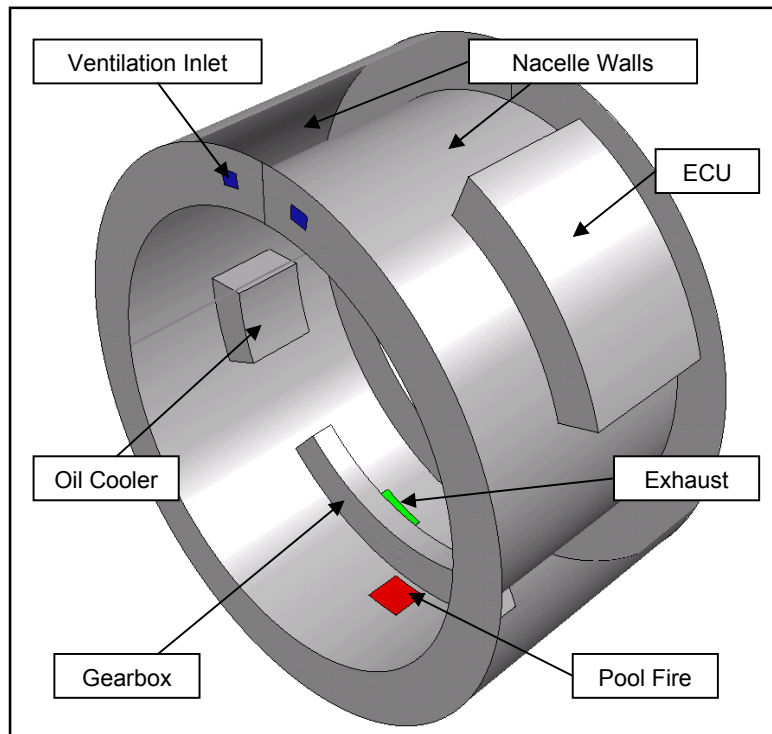
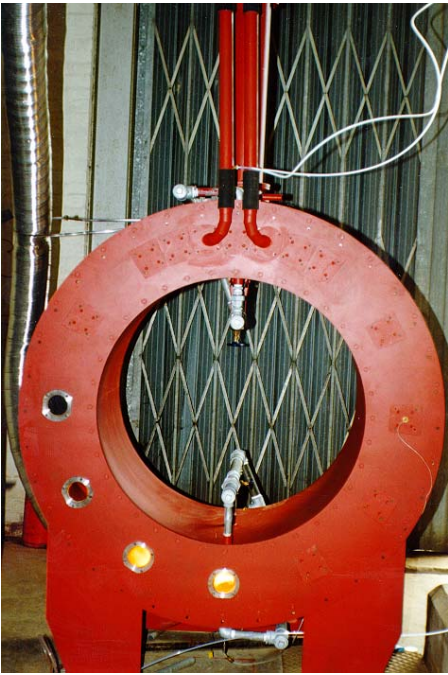


Figure 1. Experimental rig – typical configuration and accessory layout

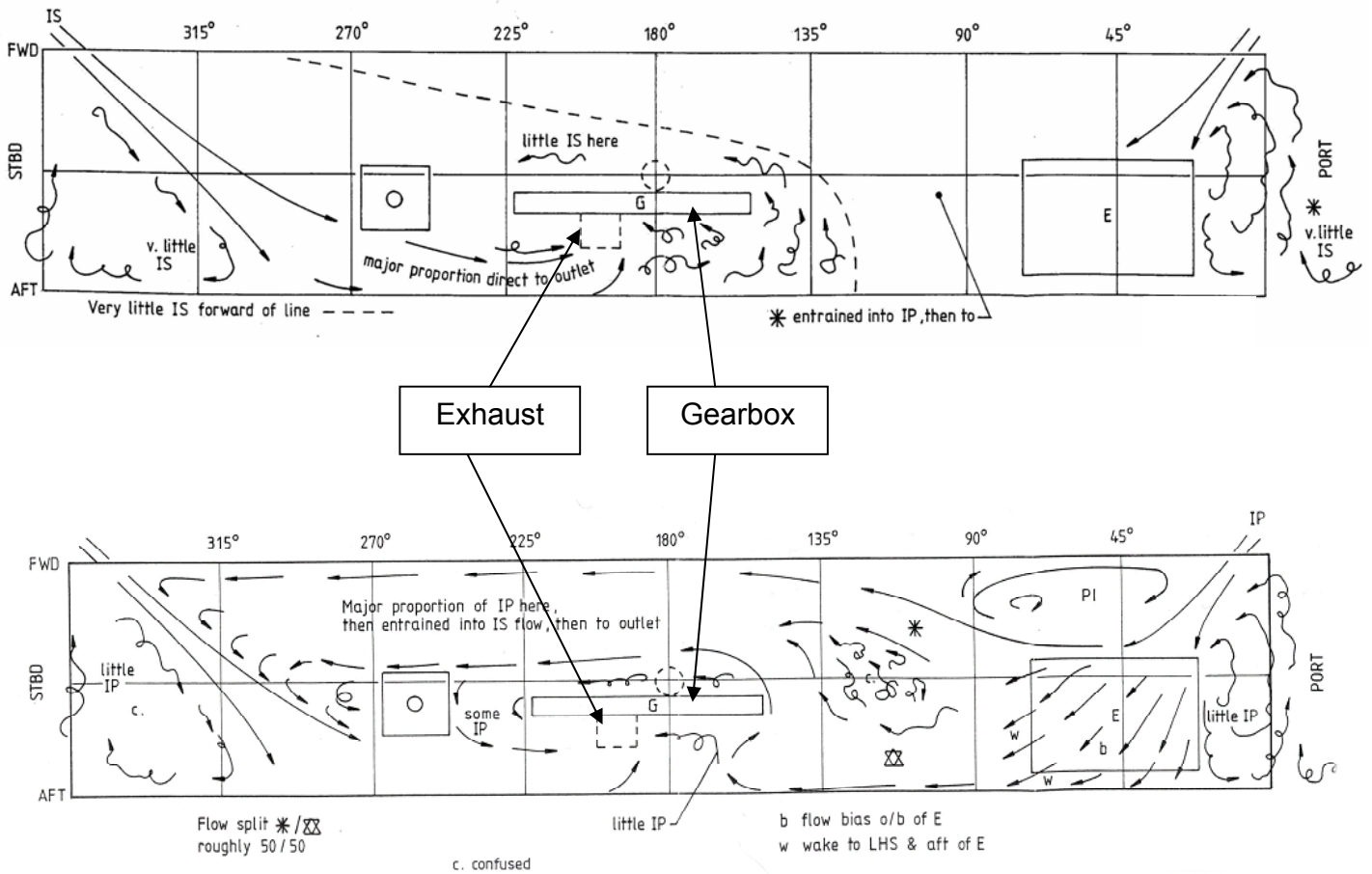
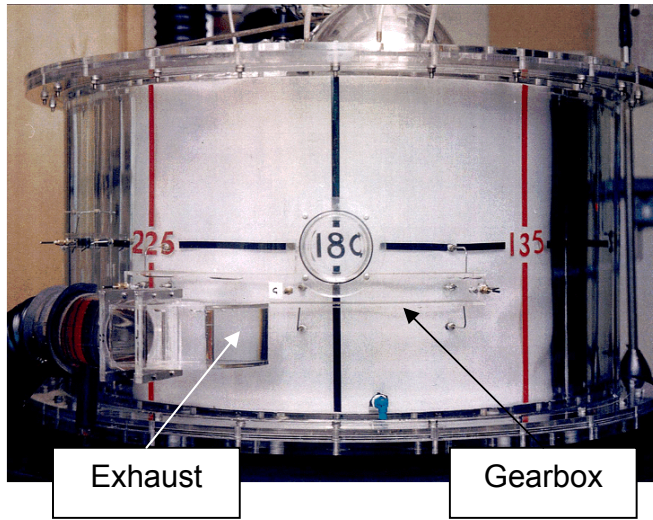


Figure 2. Ventilation flow development within the casing inferred from water flow visualisation (View from bottom dead centre)

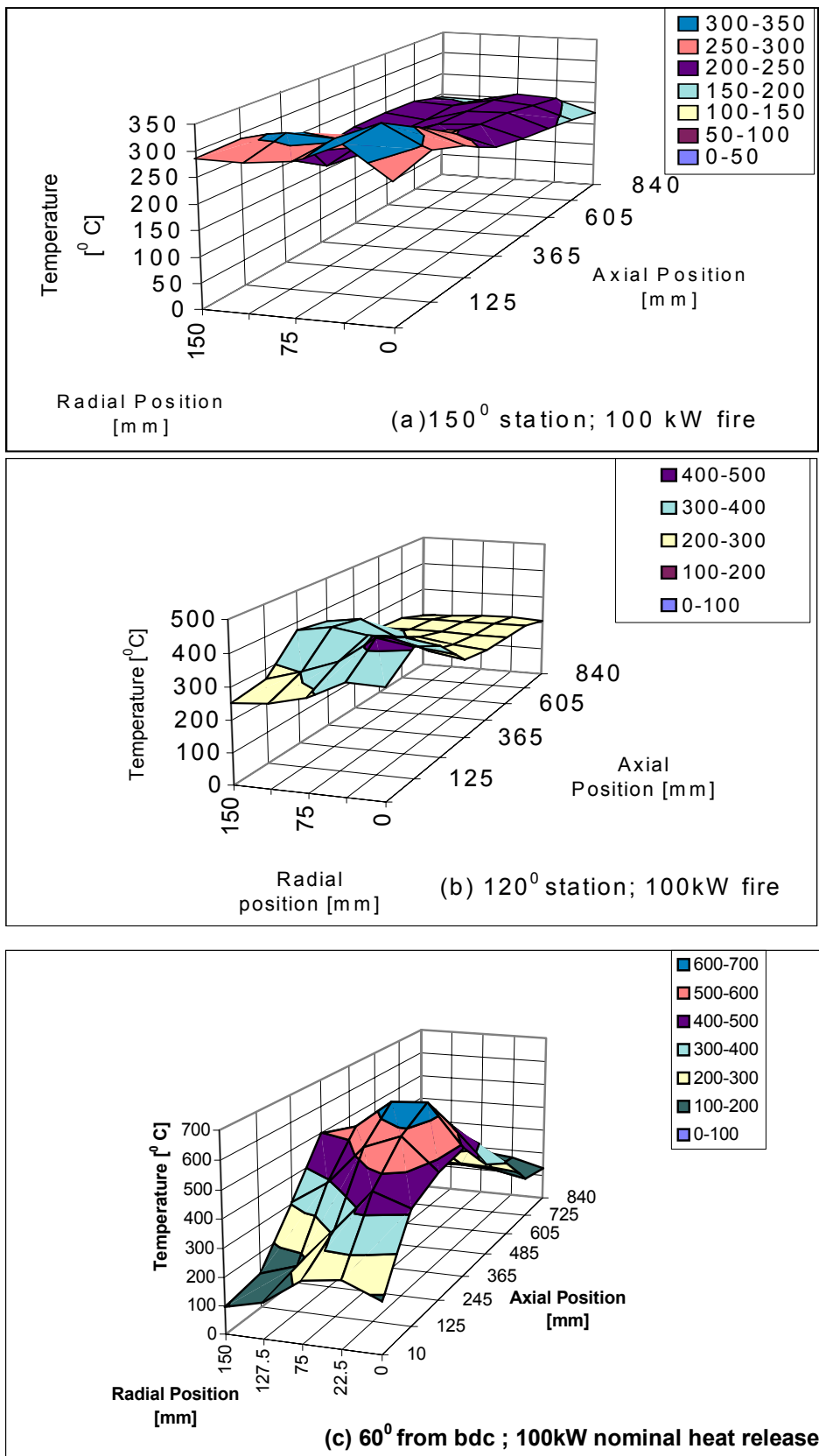


Figure 3. Illustrative fire plume development indicated by thermocouple measurements in the accessory zone at varying angular positions around the annulus

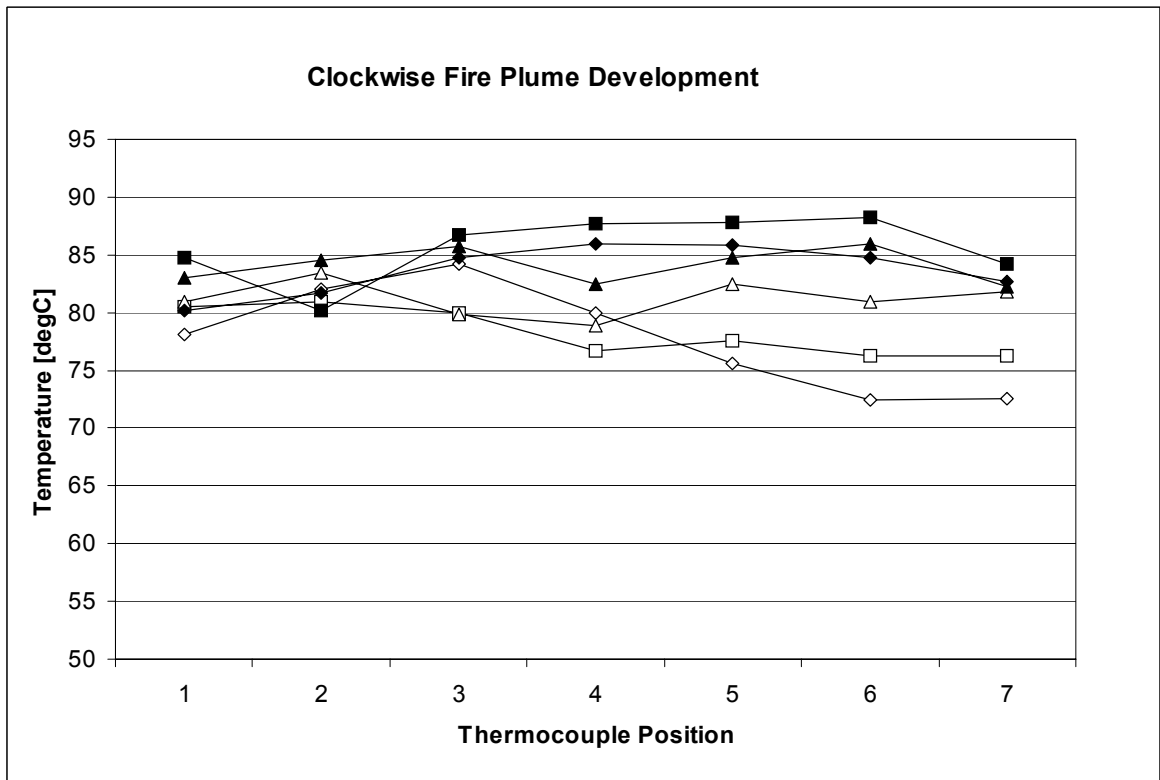


Figure 4. Temperature measurements above and below an accessory unit, located at 225° from bdc

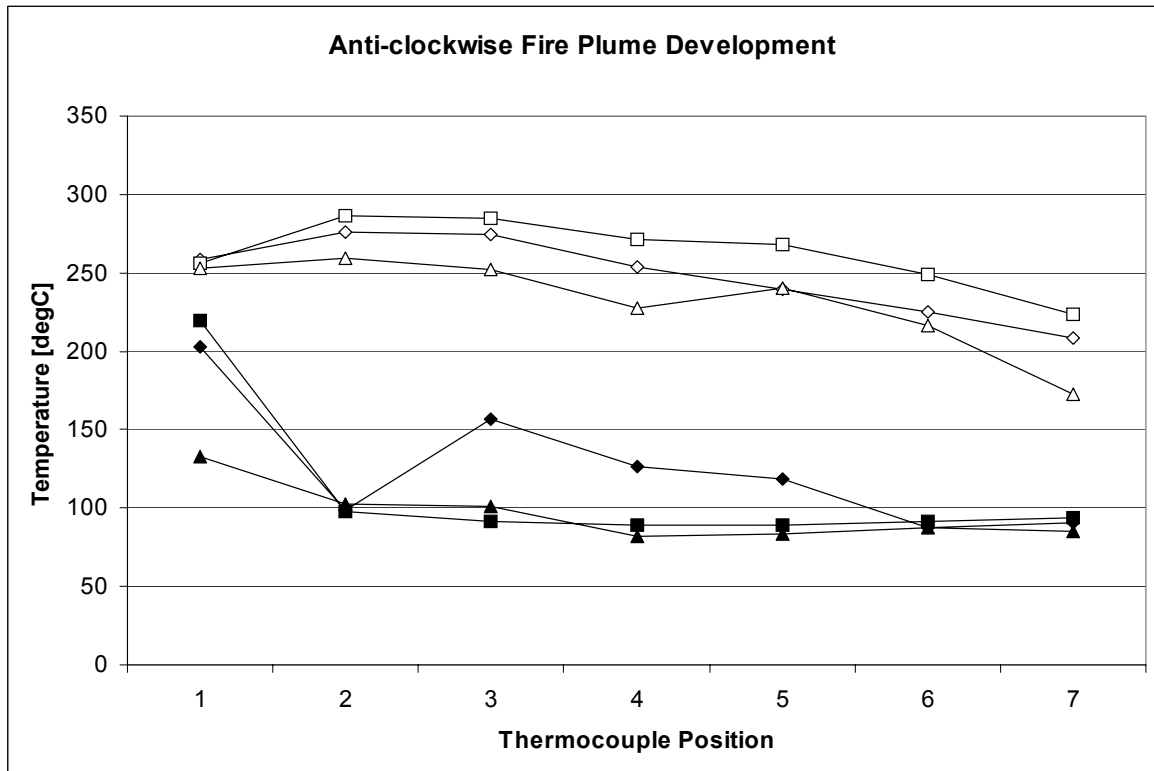


Figure 5. Temperature measurements above and below an accessory unit, located at 225° from bdc

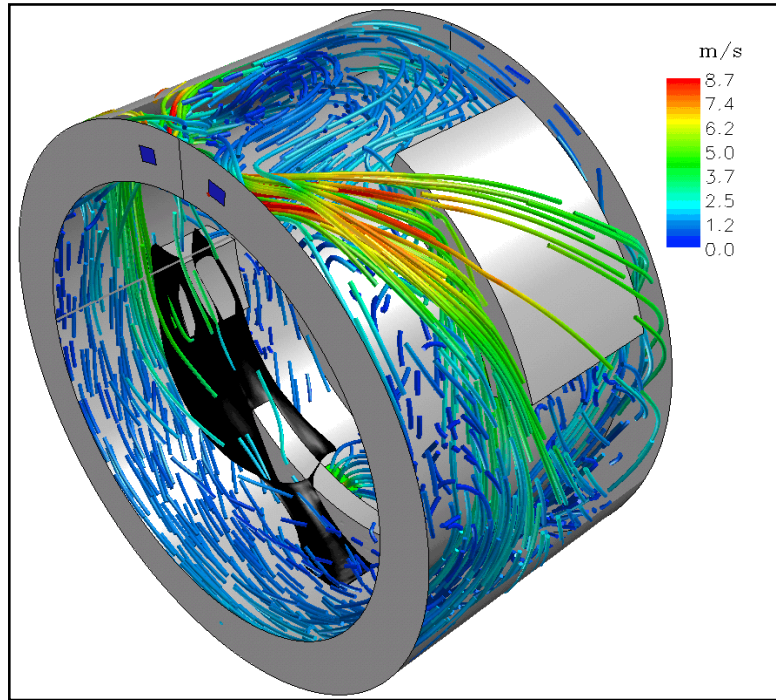


Figure 6. Visualisation of predicted flowfield in event of a pool fire (streamlines coloured by velocity magnitude)

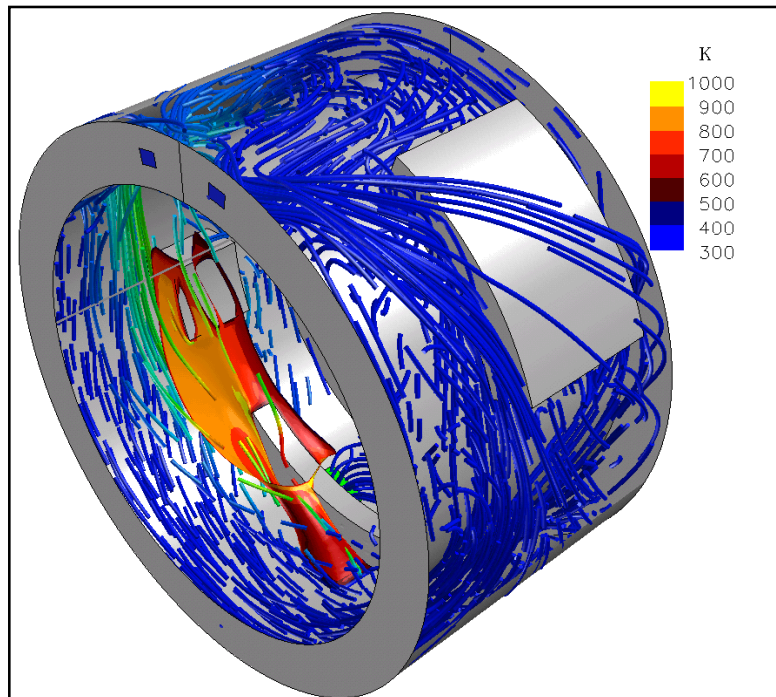


Figure 7. Visualisation of predicted fire plume (plume coloured by gas temperature)

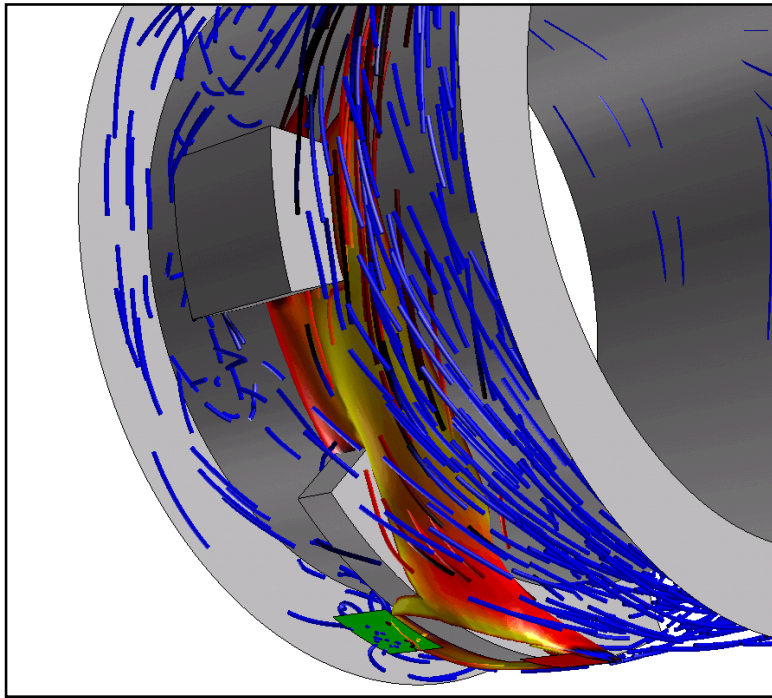


Figure 8. Near field view of flow streamlines around fire plume

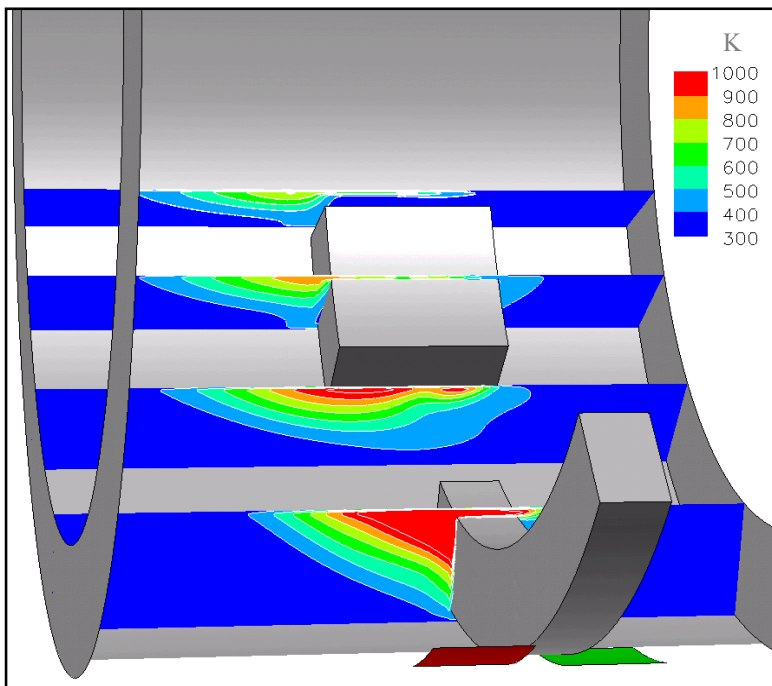


Figure 9. Local gas temperature contours around fire plume

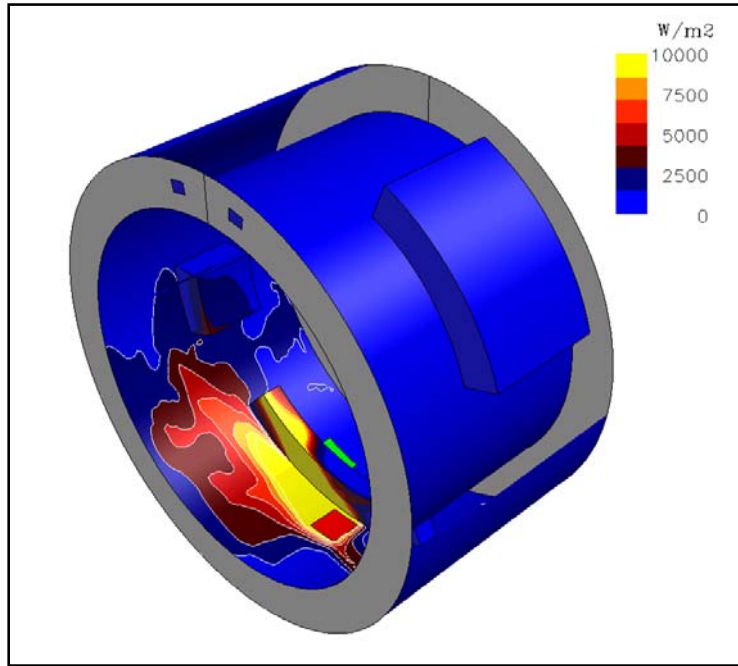


Figure 10. Predicted incident heat flux to casing walls and accessory units