STUDIES INTO THE ROLE OF VENTILATION AND THE CONSEQUENCES OF LEAKS IN GAS TURBINE POWER PLANT ACOUSTIC ENCLOSURES AND TURBINE HALLS

R C Santon, C J Lea*, M J Lewis*, D K Pritchard*, A M Thyer* and Y Sinai**
Health and Safety Executive, Quay House, Quay Street, Manchester, M3 3JB
* Health and Safety Laboratory, Harpur Hill, Buxton, Derbyshire, SK17 9JN
** AEA Technology Engineering Software, 8.19 Harwell, Didcot, Oxfordshire, OX11 0RA

The rapid growth in the installation of large gas turbines for power generation has led to the need for guidance on mitigating measures should a gas leak occur. Normal mitigation measures include ventilation to prevent the build-up of a large flammable cloud and detection to bring early attention to a gas leak. Dilution ventilation was, and remains the preferred basis of safety for gas fuelled gas turbine acoustic enclosures. Current proposals for dilution ventilation design are based on a criterion which suggests that the accumulation resulting from a gas leak which would trigger the gas detection alarm should, as an iso-surface bounding the 50% of the Lower Explosive Limit, be no more than 0.1% of the net enclosure volume. The results of a large-scale experimental study are presented which illustrate that this criterion is likely to err on the side of caution, but that there is no clear case for its relaxation. The application of dilution ventilation as a basis of safety when gas turbines are instead housed in a large turbine hall is shown, through a combined measurement and Computational Fluid Dynamics research study, to be less applicable. In this case the focus shifts towards gas detection.

Keywords: gas turbines, gas leaks, acoustic enclosures, dilution ventilation, turbine halls, CFD simulation

INTRODUCTION

BACKGROUND

In recent years there has been a rapid growth in the installation of large gas turbines for power generation[1]. Gas supply pipework to the turbine is often highly complex, with multiple joints and connections, and operated at gas pressures of up to 40 barg or even higher for some of the larger units. For example, a 250 MW unit may include over 200 flanges, 90 flexible hoses, 18 valves and 8 bellows all operating at 20 to 30 barg. Therefore there is the potential for gas leaks. Whilst it is clearly appropriate to take relevant precautions to minimise both the possibility of a gas leak and potential sources of ignition on these installations, it is not possible to completely eliminate either.

In the event that a gas leak occurs, mitigating measures are required to prevent a fire or gas explosion. Normal mitigation measures include ventilation to prevent the build-up of a large flammable cloud and detection to bring early attention to a gas leak. The effectiveness and applicability of these measures can however be expected to depend on a number of factors, in particular the presence or absence of an acoustic enclosure around the gas turbine and also the validity of criteria for dilution ventilation. This paper presents the results of two studies which build on earlier published work by HSE[1] by investigating the significance of the above factors.

DILUTION VENTILATION FOR GAS TURBINE ACOUSTIC ENCLOSURES

Dilution ventilation was, and remains, the preferred basis of safety for gas fuelled gas turbine acoustic enclosures. Enclosure ventilation air, required for turbine cooling, was originally considered to be an effective means of safely removing leaking fuel gas, whilst gas detectors would give warning of any such release so that appropriate action could be taken.

Faults in the application of this basis were exposed at a number of installations varying in scale from <2 to >200 MW. It was apparent from simple visual observations with smoke, and confirmed by Computational Fluid Dynamics (CFD) modelling, that in many cases within significant voids in the enclosures the ventilation was very poor. Despite high overall rates, air movement was very slow in some parts of the enclosures. Any gas escape from a leak into such voids could accumulate. Large volumes, in some cases amounting to over 10% of the enclosure net volume, of flammable gas/air mixture could arise from small undetected leaks. The ignition of such an accumulation would clearly give rise to an explosion with damaging overpressure consequences. It was apparent that in many such cases the distribution of the ventilation air had been poorly designed, if it had been considered at all in this context.

In some cases, of smaller simpler enclosures in particular, it has been found to be relatively straightforward to achieve more uniform and effective ventilation distribution by using baffles and/or multiple inlets/outlets. The effectiveness of such measures can be satisfactorily demonstrated using smoke, CCTV, or streamer techniques, although caution is needed if foreseeable leaks are of such a magnitude that they could significantly modify the background ventilating flow. For larger and more complex enclosures these measurement techniques are of limited value and in most cases CFD has been found to be a more useful tool, with the advantage of enabling numerous design variations to be tested theoretically. Indeed CFD is now widely used by the industry for this purpose. However, it then becomes immediately apparent that the use of CFD requires a compatible criterion to define the achievement of an acceptable and safe design. (Clearly any such theoretical design would also require subsequent practical validation.)

The principle of dilution ventilation as a basis of safety is that the ventilation should dilute and remove the gas from a leak so that no hazardous accumulation of fuel/air mixture is allowed to arise. In the case of leaks from the gas supply to a turbine, the supply pressure is invariably so high that any leak to atmosphere is sonic, or choked, and jet mixing is the dominant dilution mechanism as jet momentum slowly decays. Local ventilation flows can have little, if any, effect on this mechanism and a flammable envelope arising from and surrounding the jet is inevitable. If the jet impinges on a solid surface then mixing and dilution can be enhanced. Ventilation ultimately takes over as the mechanism to remove diluted gas, to progressively dilute it further by turbulent mixing, and to transport it to the enclosure outlet. If ventilation is not effective, and in particular if flammable mixture is directed into a poorly ventilated void and/or if re-circulation of gas/air mixture and re-entrainment into the jet arises, an accumulation of flammable mixture can arise.

Perfect dilution ventilation cannot be achieved in this case since, as noted above, jet mixing will result in a flammable envelope whilst its immediate dilution is equally impossible. To apply CFD it was seen to be necessary to suggest an upper limit for the volume of flammable mixture to be tolerated, to be predicted by the technique as resulting by these mechanisms from a leak.

It is implicit that this volume of flammable mixture is hazardous if it forms a significant proportion of the net volume of the enclosure and its ignition results in a significant overpressure. Conversely it is not hazardous if its ignition results in an insignificant overpressure in comparison with the strength of the enclosure, and no damage results.

The criterion that was proposed was that the accumulation resulting from a leak which would trigger the gas detection alarm (in a ventilated enclosure with the gas detector in the outlet) should, at 50% LEL (Lower Explosive Limit) iso-surface, be no more than 0.1% of the net volume of the enclosure. The enclosure strength should be at least 10 mbar. This criterion is similar to that proposed and validated for the admission of fuel to gas fired appliances prior to ignition[2]. A theoretical treatment of partial filling of an enclosure with flammable mixture[3], based on an adiabatic mixing model, predicts an explosion pressure of 8 mbarg for 0.1% of the net volume filled with stoichiometric methane/air mixture. It was intended that the new criterion should include a substantial margin of safety, in particular because of its uncertain basis. The various factors operating for and against it in this respect are summarised below.

The use of the 50% LEL iso-surface as a boundary represents a large overestimate of the actual volume of flammable gas, although it was found to approximate to a cloud at stoichiometric concentration of similar gas inventory. It was originally introduced to offset uncertainties in CFD modelling. Even within the LEL iso-surface there will be inhomogeniety and only a proportion of the volume will fall within the flammable range. Thus any overpressure on ignition will be substantially less than might be predicted from adiabatic stoichiometric combustion. However, turbulence induced by flow and obstacles within the accumulation may increase flame speed and overpressure and the addition of fuel is occurring over a time-scale similar to the event. Resolution of these effects is beyond the scope of most CFD techniques at this time. In addition the position and direction of the leak chosen for modelling must represent a worst case which may not be easy to identify.

The criterion is based on a single leak size defined by the ventilation rate and gas detector alarm setting. Ideally the ventilation should be checked against a range of foreseeable leak rates. However, most leaks start small. Any smaller leak than the defined value will be undetected in most cases (fortuitous additional detectors may detect such a leak) and, by definition, is not hazardous, but would be detectable if/when the leak increases and reaches the defined value.

The criterion encourages reductions in both ventilation rate and gas detector setting, since such reductions reduce the defined leaked rate and thus the size of jet and gas accumulation. They also increase the sensitivity to leak detection and thus the likelihood that small leaks will be detected and stopped early. This has found practical application.

The criterion is intended, therefore, to ensure that the ventilation will make the defined leak safe, but it must also ensure that it remains safe if it enlarges before action is taken. Even more importantly, it must have sufficient margin of safety to deal with any foreseeable initially larger leak.

It is clear from the above discussion that experimental validation of the criterion, associated with CFD, is essential.

A generic experimental study has therefore been undertaken comprising measurements of gas concentrations and explosion pressures in a large ventilated enclosure. The results are compared against the predictions of a CFD model. The findings of the study are used to assess the validity of the above criterion and give an indication as to the suitability of CFD modelling for assessing compliance with the guidance. A full report describing the study is available[4].

MITIGATION IN THE ABSENCE OF AN ACOUSTIC ENCLOSURE

Although less commonly encountered, some gas turbines are now simply sited in a very large turbine hall, providing noise reduction measures satisfy environmental and local limits. A typical volume may be an order of magnitude or more greater than that of an acoustic enclosure. Whereas the latter may have a volume of up to ~1000 m^3, a turbine hall may have a volume ten or twenty times larger.

In this case an effective strategy to deal with gas leaks may well differ from that provided by dilution ventilation. In essence, since the hall is so large, a gas cloud capable of producing significant overpressure would have also have to be large. Ventilation is again used to prevent over-heating, but the flow rates are typically small in relation to the volume of the hall and so air velocities are low, often much less than 1 m/s. It is possible that the time for gas to reach a detector could thus be significant. It would therefore appear that the main focus of mitigating measures could shift from dilution ventilation to gas detection.

A combined measurement and CFD modelling study has therefore been undertaken, examining potential gas leaks in a large turbine hall, with the main objective being to highlight differences in strategy for dealing with gas leaks in comparison to housing in an acoustic enclosure. For the purposes of this study, one of the turbine halls at the Didcot B station, owned by National Power, has been investigated, since it embodies features likely to be representative of this class of installation. Again a full report of the study is available[5].

ASSESSMENT OF THE DILUTION VENTILATION CRITERION FOR TURBINE ACOUSTIC ENCLOSURES

EXPERIMENTAL ARRANGEMENT

A rectangular steel enclosure of internal volume 93.8 m^3 (internal dimensions of 2.5 m by 2.5 m by 15 m) was used for the experimental work. Air inlets, for the ventilation air, were provided at one end of the enclosure and air outlets at the other end. The point of gas leakage was located 5m downstream of the air inlets, with the jet of discharging gas directed along the long axis of the enclosure. For safety reasons explosion relief was provided on the roof of the enclosure. A schematic diagram of the experimental arrangement is given in Figure 1.

Ventilating air was drawn through the enclosure by a variable-speed centrifugal fan, capable of providing air flows of up to 42 $m^3\ s^{-1}$. To ensure the air velocity profile in the enclosures was as uniform as possible, the air inlet and outlet arrangement consisted of 16 square holes evenly distributed over the enclosure cross-section. The edges of the openings were rounded, to minimise the disturbance to the air flow. For the tests two sizes of opening were used, one (350 mm by 350 mm) giving a nominal total open area equal to 30% of the enclosure cross-sectional area (i.e. the area normal to the flow direction) and the other (200 mm by 200 mm) a nominal total open area of 10%.

Methane gas (CP grade, 99.5% purity) was supplied from a tube trailer, or bank of manifolded cylinder pallets. The gas leak into the enclosure was simulated using a cylindrical throat type critical flow nozzle. By controlling the conditions upstream of the nozzle and using nozzles of different size (upstream pressures in the range 25 to 40 bar, nozzle bore diameters of 3, 2.1 or 1.7 mm) it was possible to produce leakage rates ranging from 0.011 to 0.052 kg s^{-1}.

Tests were conducted with the jet of methane from the critical flow nozzle unobstructed, or impinging onto an obstacle to simulate the scenario of a leak impinging onto another part of the turbine or supporting structure. The obstacles used were a square flat plate (0.5 m by 0.5 m), a round flat plate (0.5 m diameter), a pipe (0.15m diameter by 1m long) or a rectangular tray (0.5 m by 0.5 m by 0.25 m deep or 1 m by 1 m by 0.5 m deep). The tray obstacle was used to increase re-entrainment of gas into the jet and was positioned with the open side facing the jet. All obstacles were located centrally in the cross-section of the enclosure, so that jet impingement was either at right angles to the flat plate and tray obstacles or on the side of the pipe obstacle.

Electric match heads (Nobel's Explosives Vulcan electric fuse heads) were used as the ignition source. For comparison purposes tests were also carried out with small volumes of the enclosure (less than 1% of the total volume) filled with a homogenous stoichiometric methane/air mixture. This was achieved by inflating a polythene bag, suspended in the centre of the enclosure with a known volume of premixed methane/air mixture.

Measurements were made once steady state conditions had been established in the enclosure. Steady state conditions were taken to be established when the gas concentrations around the point of leakage changed by less than ± 0.1% v/v. Steady state conditions were usually established within a few minutes, but all tests were continued for at least three cycles of the gas sampling sequencer unit (about 15 minutes).

Gas concentrations at given locations within the enclosure were measured by the use of metal sampling probes. Samples were withdrawn through the probes via a gas sampling sequencer unit into a calibrated infra-red gas analyser. The analyser used was capable of measuring methane concentrations of up to 20% v/v with an accuracy of ± 0.1%. Air velocity transducers, operating on a hot-wire principle, were used to measure the flow velocities within the enclosure. They were calibrated to an accuracy of ± 2.0% of the reading. Strain gauge or piezo-resistive pressure transducers, mounted in the walls of the enclosure, were used for measuring the explosion pressures. They were capable of measuring pressures to an accuracy of ± 5% and detecting pressure rises of well below 1 mbarg.

Further details on the experimental rig, the instrumentation and the test programme can be found in the project report[4].

CFD MODELLING

The modelling work was carried by AEA Technology using their CFX-4 code, together with special explosion routines which were developed by AEA Technology under contract to HSE[6].

At the outset a deliberate decision was made to use a coarse-meshed point-source technique to represent the high-pressure gas leak. This approach is typical of the majority of applications of CFD in this field, in which the details of the high-pressure source are not explicitly resolved by the CFD grid. Instead, equivalent mass, momentum and energy sources are added in appropriate grid cells. The advantages and limitations of this approach are documented in Lea et al[7]. The point-source technique has been widely used by AEA Technology and other organisations because only one CFD grid needs to be created, irrespective of the number, position and orientation of leaks. This is an advantage when the project is subject to tight time constraints (as is common). However, other recent work by AEA Technology has resolved the leaks fully with local mesh refinement and unstructured mesh technology, instead of adopting the point-source technique.

The following sub-models and assumptions were also employed:

- Compressible, buoyant flow.
- High Reynolds number k- turbulence model with buoyancy (C_3=1)
- Adiabatic walls
- Steady stagnant conditions in the atmosphere outside the rig
- Higher-order convection discretisation for all transported flow variables

The sensitivity of results to grid resolution was investigated by a grid refinement study. The results of this study indicate that predicted gas concentrations were insensitive to grid resolution except very close to the leak source. This is unsurprising, given that a coarse-mesh approach was used to resolve the source. To obtain uniform flow in the rig, it was found important to extend the CFD grid beyond the rig inlet into the surrounding atmosphere.

Simulations were carried out for an unignited free jet, an unignited jet impinging on a square flat plate and one case in which the free jet was ignited to produce an explosion.

RESULTS AND DISCUSSION

For the unobstructed jet, that is with the jet of gas from the critical flow nozzle discharging into an empty enclosure, the effect of the area of the inlets and outlets, the methane leakage rate and the ventilation rate on the gas concentrations, were all investigated. The results for smaller inlets/outlets (10% free area) are summarised in Table 1 and the locations of the sample probes shown in Figure 2. Comparing the values in Table 1, shows that changing the ventilation rate had very little effect on gas concentrations in the enclosure and the size of the resulting flammable cloud. However, as expected, there was a marked dependency on the methane leakage rate. Very similar results were obtained with the larger air inlets/outlets (30% free area). In test CHP028, using the design conditions of 0.023kgs^{-1} leakage rate and 800 m^3 min^{-1} ventilation rate, it is estimated that the volume of the 50% LEL iso-surface was less than 0.02 m^3.

Comparisons with the predictions of the CFD model for test CHP028 are shown in Table 2. The dispersion of this free jet is reasonably well predicted, except at the edge of the jet close to the leak source. This is probably a consequence of the use of point source modelling, which is unable to resolve details near the point of leakage. It should also be pointed out that gas concentrations, both measured and simulated, are very sensitive to position at this location in the jet shear layer - with small changes in position resulting in large changes in concentration: This direct comparison of point values thus represents a severe test for the model. A slight asymmetry is also evident in the model results for this case. It is believed that this is due to the influence of the ground modelled outside the test section. Reducing the simulated ventilation rate to 400 m^3 min^{-1} made very little difference to the predicted gas concentrations, in agreement with the experimental findings.

The net effect of placing flat plate or pipe obstacles within the enclosure so that the jet of methane impinged upon them, at least for the size, type and position of obstacles used in the tests, was to reduce the size of the flammable cloud. High gas concentrations were observed close to the surface on which the jet impinged, but the concentration decreased rapidly on moving away from the surface.

Model predictions and experimental results for impingement on a square flat plate obstacle are compared in Table 3. The level of agreement is now less satisfactory, in that the model over-predicts concentrations. This does mean, however, that these predictions can be interpreted as being conservative. The reasons for the over-prediction are again probably related primarily to the use of a point source approach, together with shortcomings of the turbulence model for impinging and separated flows. These findings illustrate the need to adopt a cautious approach when assessing the results of CFD modelling.

Table 1 - Results for unobstructed jet with smaller air inlets/outlets

Test No (and conditions)	Methane concentration : % v/v									
	Probe No (for locations see Figure 2a)									
	1	2	3	4	5	6	7	8	9	10
CHP029 (0.022 kg s^{-1}, 400 m^3 min^{-1})	7.1	4	2.4	1.4	0.1	0	0.1	0.1	0.9	0.8
CHP028 (0.022 kg s^{-1}, 800 m^3 min^{-1})	6.9	3.6	2.2	1.2	0	0	0	0	0.8	0.7
CHP030 (0.022 kg s^{-1}, 1600 m^3 min^{-1})	6.6	3.1	1.8	0.9	0	0	0	0	0.6	0.5
CHP031 (0.012 kg s^{-1}, 800 m^3 min^{-1})	4.3	1.6	0.8	0.4	0.1	0	0.3	0.1	0.2	0.3
CHP032 (0.044 kg s^{-1}, 800 m^3 min^{-1})	9.9	5	2.8	1.4	0.1	0.1	0.2	0.1	1.2	0.8

Table 2 - Comparisons with predicted gas concentrations for an unobstructed jet

Test No (and conditions)	Methane concentration : % v/v									
	Probe No (for locations see Figure 2a)									
	1	2	3	4	5	6	7	8	9	10
CHP028 (0.022 kg s^{-1}, 800 m^3 min^{-1})	6.9	3.6	2.2	1.2	0	0	0	0	0.8	0.7
Predicted	5.7	3.7	2.8	2	0.4	0.8	0.9	1.4	0.7	0.9

Table 3 - Comparisons with predicted gas concentrations with flat plate obstacle

Test No (and conditions)	Methane concentration : % v/v									
	Probe No (for locations see Figure 2b)									
	1	2	3	4	5	6	7	8	9	10
CHP033 (0.022 kg s^{-1}, 800 m^3 min^{-1})	0.6	0.5	0.4	0.3	0	0	0.6	0.6	0.5	0.5
Predicted	1.3	1.1	0.9	0.7	1.2	1.2	1.6	1.5	1.1	1.1

Tray type obstacles were used with the aim of increasing the volume of the 50% LEL iso-surface by producing re-entrainment of gas into the gas jet discharging from the critical flow nozzle. For the smaller tray obstacle (0.5m by 0.5m by 0.25m deep), with a leakage rate of 0.046 kg s^{-1} and ventilation rate of 400 m^3 min^{-1}, the maximum 50% LEL iso-surface volume that was achieved is estimated at about 0.25 m^3. This estimate was obtained by interpolation of the gas concentration measurements made around the point of leakage. The maximum volume achieved in the tests with the larger tray type obstacle (1m by 1m by 0.5m deep), for the same leakage and ventilation rates, is estimated at 0.75 m^3. The latter volume is 0.8% of the enclosure volume, that is eight times the value recommended in the interim design guidance[1].

Ignition tests were carried out with the methane jet unobstructed, or with the jet impinging on an obstacle (flat plate, pipe or tray type). With the air ventilation on, methane release was continued for sufficient time to allow steady state conditions to be established within the enclosure, before the resulting gas cloud was ignited. Two electric match heads, connected in parallel, were usually used in a test and located at positions where the concentration measurements indicated the mixture was flammable.

In none of the tests was a measureable pressure rise recorded by the pressure transducers during an ignition, so any pressure rise produced was well below 1 mbarg. Even for the larger tray obstacle, where the volume of the 50% LEL iso-surface is estimated to have been about 0.8% of the enclosure volume there was no measureable pressure rise. That ignition had occurred in these tests was confirmed by a video camera located within the enclosure. The video recordings showed flame spreading out from the point of ignition throughout the region where the flammable cloud would have formed. In most tests the flame then rapidly extinguished, but in a few of the tests the jet of gas issuing from the critical flow orifice continued to burn until the methane supply was turned off.

The CFD model has also been used to predict the explosion pressure for one of the ignition tests. For an unobstructed jet with a leakage rate of 0.022 kg s^{-1} and ventilation rate of 800 m^3 min^{-1} the predicted maximum pressure rise was 0.7 mbarg, while the measured pressure for these conditions was negligible. Although both measurement and simulation are in agreement to the extent that both show the pressure rise to be inconsequential, a rise of well below 1 mbarg would have been detected by the pressure transducers used. CFD modelling of explosions is a particularly demanding application. This work illustrates that although results are encouraging, further development is still needed.

For comparison purposes, the pressures generated by the ignition of known volumes of stoichiometric methane/air mixture within the enclosure were measured. In these tests there was no air ventilation, the air inlets and outlets were sealed and the gas mixtures were quiescent. The results obtained are plotted in Figure 3. The theoretical line shown on the plot was that obtained using an adiabatic mixing model[3] developed for predicting the pressures developed in enclosures partially filled with a flammable mixture - in this case a stoichiometric methane/air mixture.

CONCLUSIONS

Provided the enclosure is efficiently purged by the ventilation, i.e. no dead spaces, and there is no re-entrainment into the jet of gas from the leak, the size of the flammable cloud will be very small for the leak rates considered in this study. The flammable cloud size is then mainly controlled by the leakage rate rather than the ventilation rate. This is a consequence of the dilution of the leaking gas being controlled by turbulent jet mixing, rather than the forced ventilation flow through the enclosure. Impingement of the jet on an obstacle, provided it does not cause re-entrainment, tends to reduce the size of the flammable cloud. Configurations where re-entrainment is induced or where there is a dead space can, however, result in the formation of appreciable volumes of flammable mixture.

The results of the present study indicate that keeping the volume of the 50% LEL iso-surface below 0.1% of the enclosure volume should ensure that explosion pressures are well below the 10 mbarg level. Indeed the results suggest that this volume requirement could possibly be increased by a factor of two or three and still meet the pressure requirement. However, caution should be exercised in generalising this conclusion: Thus the behaviour of ignited gas clouds in very large volumes characteristic of acoustic enclosures is uncertain; more importantly, leaks into congested areas of plant would be expected to increase flame speeds and hence overpressures above those measured in this study. In addition the criterion must allow for a substantial margin of safety to accommodate larger leaks than that on which it is based. It is therefore recommended that the present criterion and guidelines for design and assessment of enclosure ventilation are not relaxed.

This study has also shown that CFD can be used to indicate the concentration distribution, and hence volume, of a flammable gas cloud arising from a high pressure leak inside a large empty enclosure, at least when the cloud is small in relation to the enclosure volume. There is more uncertainty in the modelling when the leaked gas impinges on an obstacle, with concentrations being over-predicted in the present case. Predicted concentrations very close to the leak source also differ from those measured as a consequence of the use of a point-source modelling approximation. The accuracy of the discretisation scheme has been found to exert a significant influence on the results, and it is recommended that higher-order discretisation schemes should be used as a matter of course. In general the model tends to over-predict the size of flammable clouds and can therefore be interpreted as being conservative.

Very low explosion overpressures were predicted by the CFD model, in qualitative agreement with the measurements. However, the model does over-predict the magnitude of the measured pressure rise.

The study highlights the need to interpret the results of CFD modelling with some caution; that sensitivity studies - particularly to mesh resolution, are recommended to address uncertainties; that CFD can nevertheless be used as a predictive tool to give an indication of the hazards associated with gas leaks in circumstances in which other techniques are less reliable - providing that its limitations are not ignored.

MITIGATION IN THE ABSENCE OF AN ACOUSTIC ENCLOSURE

MEASUREMENTS

Flow measurements were made by HSL in one of the Didcot B gas turbine halls, housing two units with silo combustors. The hall encloses a volume in excess of 20,000m^3 and is forced ventilated by fans exhausting to atmosphere via four outlets at roof level. Air is drawn into the hall from outside via 13 inlets arranged at low level. Figure 4 shows a schematic of the hall. Gas detectors, set to alarm at 10% LEL, are located above each silo combustor and close to the exhaust outlets. The measurements comprise velocity and temperature taken at 27 different locations around the hall, as well as at all ventilation inlets. Thermal imaging data on the gas turbines was supplied by National Power. Measurements were taken on two separate occasions, giving information on the variability of conditions inside the hall. Details can be found in Lewis[5]

CFD MODELLING

An initial CFD modelling study based on a simplified representation of the hall was undertaken by AEA Technology. That work modelled half the hall. This paper concentrates on the findings of further modelling work by HSL. In both cases the CFX-4 code was used.

The sub-models and assumptions listed above are also employed in the present case. The CFD grid consists of approximately 370,000 cells, which is sufficient to resolve the largest features in the hall - shown in figure 4, but still too coarse to represent pipework, etc. Porous regions were thus defined to represent congested regions above and below the gas turbines, in which the gross effects of blockage, increased flow resistance and turbulence generation are incorporated in the model by applying sub-grid source terms.

Simulations were initially conducted to model the background ventilation flow in the hall and results compared to measurements. A potential worst case leak, sited below a silo combustor, and directed underneath the gas turbine, was then introduced. The leak rate was chosen so as to just trigger the gas detection system when fully mixed in the ventilating flow. Both steady-state and transient simulations were undertaken.

A possible time for a gas leak to occur is probably following shut down for maintenance. Upon start-up the gas turbine will be cold and therefore the natural convection-driven flow above the turbine is not present. This will change the flow patterns inside the hall, with the possibility that any build-up of gas will take longer to reach and trigger the gas detectors. The above simulations were therefore repeated with cold gas turbines.

RESULTS AND DISCUSSION

The measured ventilation flow in this turbine hall was found to be significantly affected by the external atmospheric conditions - i.e. wind velocity and temperature. These are inherently unsteady and make the task of imposing model boundary conditions difficult. There are other minor openings to atmosphere in the hall - doors slightly ajar, gaps under roller-shutter doors, which again are difficult to represent. Steady-state boundary conditions were instead assumed. The prescribed flow boundary conditions are therefore likely to result only in a representation of the gross flow features. It is not realistic to expect the model to replicate the exact flow features on any particular day. These difficulties in modelling normal ventilation conditions are unlikely to be restricted to the Didcot B station and would probably be encountered for other large turbine halls.

The importance of carrying out sensitivity tests and, where possible, comparing to measurements, is illustrated by the above difficulty in defining appropriate boundary conditions. Following a series of sensitivity tests, a decision was taken to apply the design ventilation flows as boundary conditions coupled to an adiabatic wall boundary condition. This resulted in close agreement between simulation and measurement for the temperature field - which is strongly stratified in the vertical direction (Figure 5), but underprediction of measured flow speeds - particularly away from regions of strong forced or natural convection, i.e. away from the ventilation inlets and the buoyant plume rising above the hot turbine. An underprediction in the background flow speed implies that the rate at which dilute gas is transported by the background flow will also be underpredicted. That is, simulated times to detection could be conservative.

Repeating one of the simulated leaks studied in earlier work by AEA Technology gave a predicted 50% LEL iso-surface of 28m^3. This is consistent with the AEA figure of 40.5m^3 and to some extent illustrates that in certain limited circumstances a more simplified representation of the turbine hall may suffice. It should be pointed out that this level of agreement was only achieved if the leak source was in both cases sited outside of the porous region. In the present study it was found that locating the leak source inside a defined porous region could lead to the prediction of very large flammable clouds. Our interpretation of this finding is that this reflects an area of considerable uncertainty in CFD modelling and requires further study.

The results of steady-state simulations which model a potential worst case leak - that which is only just detectable when fully mixed at the ventilation outlet, is shown in Figure 6. The 50% LEL iso-surface is in this case predicted to occupy 7m^3 if porous regions are neglected, 10m^3 if they are included in the model - with the source located outside of, but directed towards, the porous region. These volumes of gas are substantially less than the 0.1% criterion. Figure 7 illustrates the extent of the 10% LEL iso-surface. It fills a large part of the hall and several gas detectors would have been triggered.

Transient simulations for this potential worst case leak allow an indication to be gained of the time to detection. It is found that gas detectors located immediately above the combustors would be triggered after approximately one minute. Time-dependent simulations also allow an investigation of the build-up and transport of gas inside the hall. In essence the results show that gas initially builds-up in the lower part of the hall - confined by the stable temperature stratification. In due course the gas cloud becomes entrained by the buoyant plume rising above the hot turbine, before becoming mixed within the upper part of the hall.

Steady-state and transient simulations of this leak case under start-up conditions show that close to the leak source the cloud behaves in a similar manner to that under normal operating conditions, i.e. the jet momentum dominates. As a consequence, the 50% LEL iso-surface volume is very similar to that under normal operating conditions. However, once momentum has decayed the behaviour is somewhat different: With cold gas turbines the cloud builds-up and rises mainly in the middle of the hall, driven by weak forced convection and natural buoyancy, reaching the extract ducts some 30 seconds later than under normal operating conditions. The 10% LEL iso-surface again fills a substantial volume of the entire hall, with the result that the time to detection is still approximately one minute.

The use of CFD for the simulation of flow throughout the entire space of a turbine hall is certainly more challenging than its application to acoustic enclosures. There are two main reasons for this: Firstly, as discussed above, the flow in a large hall is more likely to be influenced by external atmospheric conditions; the specification of appropriate boundary conditions thus becomes more uncertain. Secondly, the larger volume to be modelled means that it is more difficult to adequately resolve key geometrical and flow features with the CFD grid. The above results should therefore be treated with some caution.

CONCLUSIONS

It appears that potential worst case gas leaks in gas turbine halls are less likely to lead to the build-up of flammable clouds which would cause significant over-pressure than is the case for acoustic enclosures. This is a direct consequence of similar overall ventilation rates and detection levels in the two cases, which leads to a similar-sized worst case leak, yet sited in a much larger volume.

Ventilation flows in large turbine halls seem more likely to be affected by external atmospheric conditions than is the case in acoustic enclosures. This, together with the large volume of the hall, means that design and control of dilution ventilation is more difficult to achieve in a turbine hall. However, this study indicates that dilution ventilation is also less appropriate as a basis for safety in large turbine halls. Instead the focus is shifted towards gas detection.

This study indicates that detectors located immediately above turbine combustors will be triggered by a just-detectable leak after approximately one minute. Indeed the extent of the computed 10% LEL iso-surface suggests that the exact location of gas detectors may not be critical. However, this finding is based on a necessarily limited set of simulations, which are likely to be less reliable than similar simulations for acoustic enclosures. Other leaks which could be foreseeable may result in the build-up of a gas cloud in a part of the hall remote from detectors, with a subsequent increase in the time to detection. In this case the siting of deflectors designed to direct the passage of a potential gas leak towards detectors may be beneficial. This measure has been implemented at the Didcot B station. The design and assessment of performance of deflectors should be amenable to CFD modelling.

ACKNOWLEDGEMENTS

The authors gratefully acknowledge the assistance and co-operation of National Power, in particular Dr P Stephenson, for arranging access to and information on the Didcot B station. The contribution of J A Allsopp, G T Eaton, D Hedley, R Hambleton & J Saunders, of the Health and Safety Laboratory, in undertaking the experimental measurements reported in this paper, is gratefully acknowledged. The authors also thank the referee for helpful comments.

REFERENCES

1. Santon, R.C., 1998, Explosion hazards at gas turbine driven power plants, ASME TURBO EXPO '98, ref ASME 98GT-215, 2-5 June 1998, Stockholm, Sweden.

2. Cubbage, P A & Marshall, M R, 1972, Pressures generated in combustion chambers by the ignition of gas-air mixtures. IChemE Symp. Series No. 33, London.

3. Ogle, R.A., 1999, Explosion hazard analysis for an enclosure partially filled with a flammable gas, *Process Safety Progress* 18(3): 170-177.

4. Pritchard, D. K, Allsopp, J. A., Eaton, G. T., Hedley, D., and Lewis, M. J., 2000, Dilution ventilation criterion validation, HSL Report No. EC/99/50 - CM/99/17.

5. Lewis, M. J., 2000, Explosion hazards in gas turbine power plant: Ventilation and detection in the absence of an acoustic enclosure, HSL Report No. CM/99/12.

6. Guilbert, P.W. and Jones, I.P., 1996, Modelling of explosions and deflagrations, HSE Contract Research Report No 93/1996.

7. Lea, C. J et al, 1997, Assessment of gas dispersion in CCGT enclosures: Measurement and CFD modelling using the CFX software, Prevention and Control of Gas Explosions in CCGT and Steam Plants, IMechE HQ, 8 October, London.

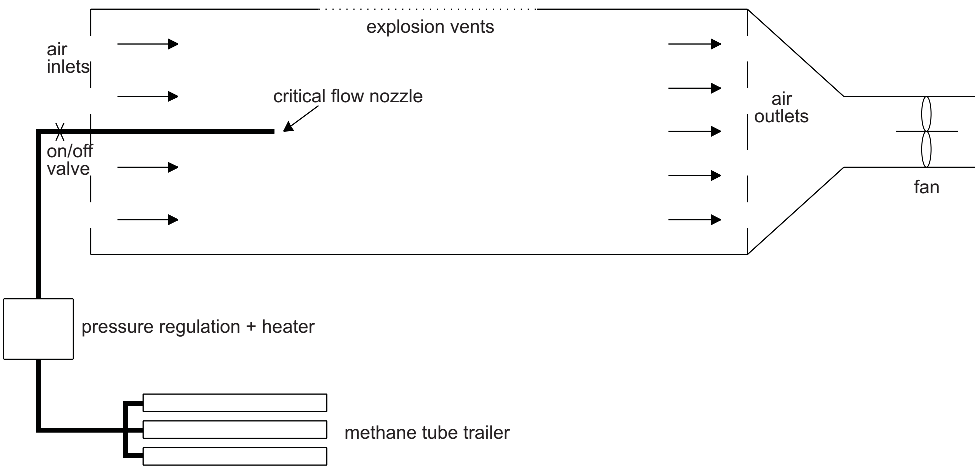
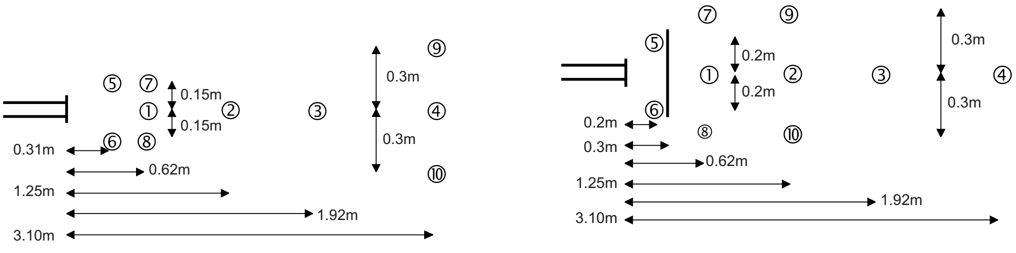


Figure 1 - Schematic arrangement of the experimental rig (not to scale)



(a) **(b)**

Figure 2 - Gas sampling probe positions

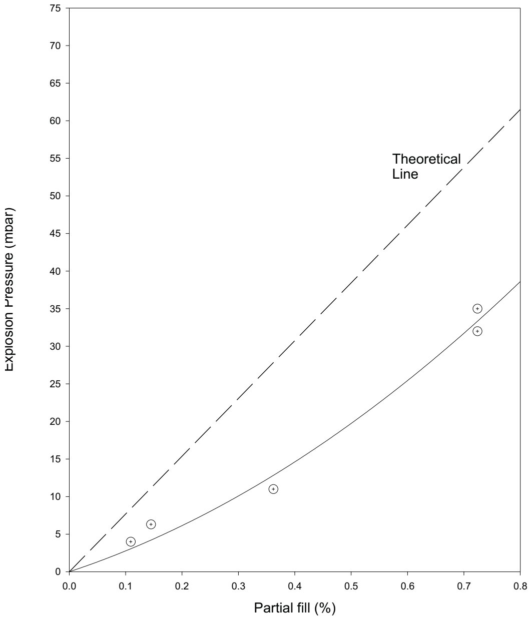


Figure 3 - Plot of explosion pressure versus percentage fill

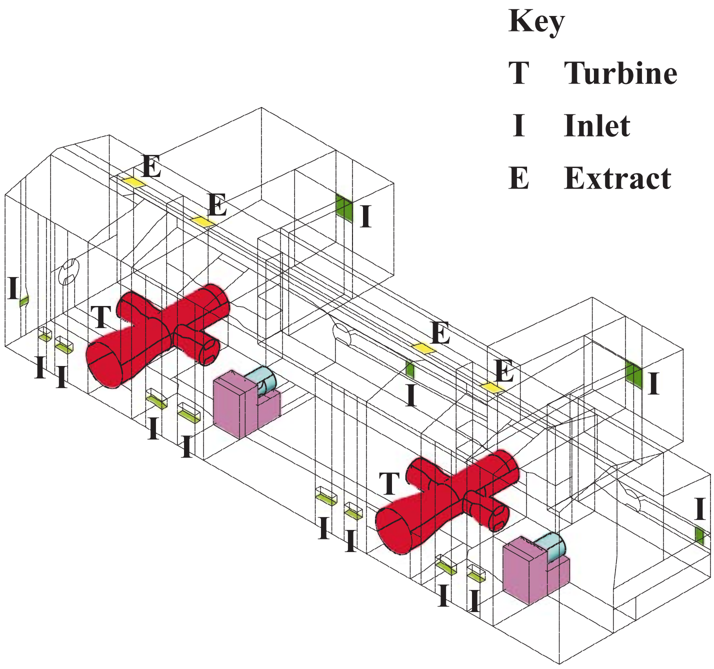


Figure 4 - Schematic of the turbine hall, showing the main features modelled

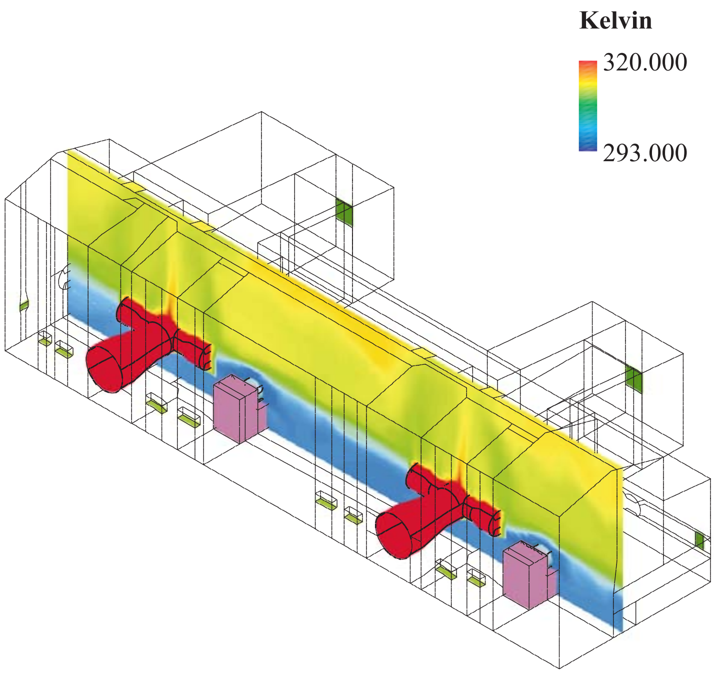


Figure 5 - Simulated steady-state temperature field under normal operating conditions

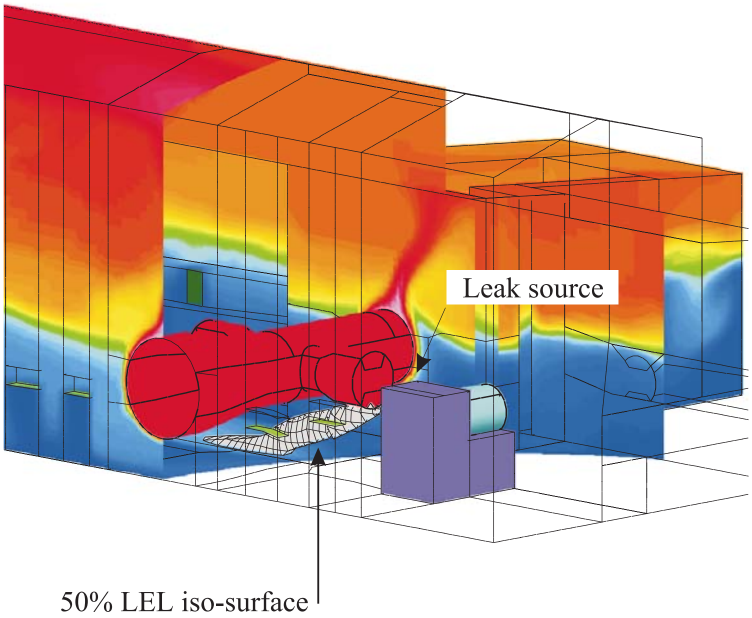


Figure 6 - Potential worst case leak; iso-surface at 50% LEL

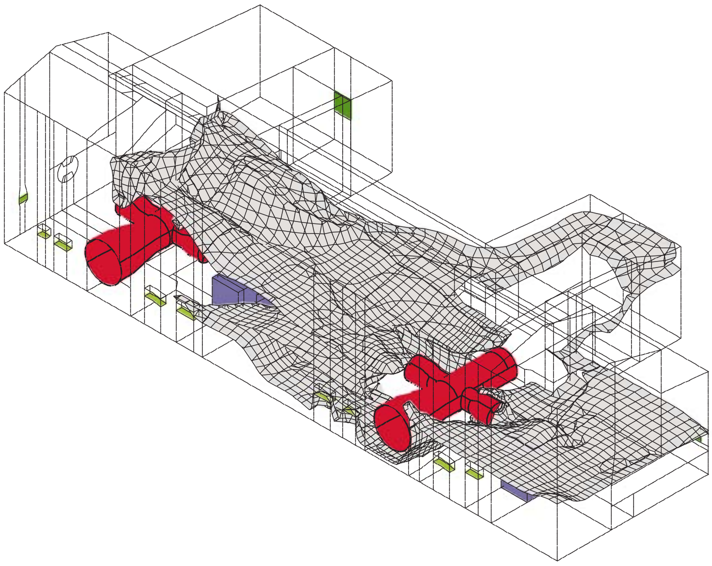


Figure 7 - Potential worst case leak; iso-surface at 10% LEL