Journal of Hazardous Materials 184 (2010) 170–176

Contents lists available at ScienceDirect

Journal of Hazardous Materials

journal homepage: www.elsevier.com/locate/jhazmat

Flammable gas cloud build up in a ventilated enclosure

M.J. Ivings*, S.E. Gant, C.J. Saunders, D.J. Pocock

Health and Safety Laboratory, Harpur Hill, Buxton SK17 9JN, UK

ARTICLE INFO

Article history:
Received 15 March 2010
Received in revised form 5 August 2010
Accepted 7 August 2010
Available online 14 August 2010

Keywords:
Ventilation effectiveness
Dispersion
CFD
Hazardous area classification
Gas jet

ABSTRACT

Ventilation is frequently used as a means for preventing the build up of flammable or toxic gases in enclosed spaces. The effectiveness of the ventilation often has to be considered as part of a safety case or risk assessment. In this paper methods for assessing ventilation effectiveness for hazardous area classification are examined. The analysis uses data produced from Computational Fluid Dynamics (CFD) simulations of low-pressure jet releases of flammable gas in a ventilated enclosure. The CFD model is validated against experimental measurements of gas releases in a ventilation-controlled test chamber. Good agreement is found between the model predictions and the experimental data. Analysis of the CFD results shows that the flammable gas cloud volume resulting from a leak is largely dependent on the mass release rate of flammable gas and the ventilation rate of the enclosure. The effectiveness of the ventilation for preventing the build up of flammable gas can therefore be assessed by considering the average gas concentration at the enclosure outlet(s). It is found that the ventilation rate of the enclosure provides a more useful measure of ventilation effectiveness than considering the enclosure air change rate.

1. Introduction

Ventilation is frequently used as a means for preventing the build up of flammable or toxic gases in enclosed spaces and its effectiveness at diluting these down to safe or tolerable levels often has to be considered as part of a safety case or risk assessment. The effectiveness of the ventilation will depend on a number of factors such as the ventilation rate, the distribution of the ventilation throughout the space, the amount of contaminant that needs to be controlled, obstacles and their geometry, and the location of the source of contaminant.

In the context used here, ventilation is the movement of air through or within an enclosure that is used to minimise the flammable extent of a release of natural gas. The applications to which this work is relevant are enclosures, rooms or buildings within which low-pressure natural gas pipes and fittings have been installed. These include, for example, low-pressure district regulator enclosures, gas distribution pipework in commercial and industrial premises and boiler houses. The Dangerous Substances and Explosive Atmospheres Regulations (DSEAR) [1] apply to all such cases and area classification is thus required. The majority of these applications are ventilated naturally. Forced ventilation is not normally used for safety reasons but for operational purposes, i.e. in air conditioned buildings or for the cooling of plant or machinery. In such cases, ventilation may not be seen as a safety feature and may therefore be of limited reliability.

This paper presents research that has been carried out to provide a methodology for assessing ventilation effectiveness which can be used in area classification. The gas leaks that are within the scope of this work are relatively small compared to the ventilation and size of the enclosure, i.e. the catastrophic failure of pipework/equipment is not considered. However, these small leaks are still sufficiently large for the ventilation to have an effect on the size of the resulting flammable gas cloud. Previous work by Cleaver et al. [2] has considered releases of natural gas into enclosures with very low levels of ventilation. In these low ventilation cases, the buoyancy of the gas plays an important role in determining the dispersion of gas within the enclosure. While in the present work, for the majority of cases, the momentum of the ventilation and jet release dominate the flow.

Pressurised leaks of natural gas are characterised by jets that rapidly mix with the surrounding air due to the shear induced turbulence generated by the jet momentum. In the absence of any obstacles in the outdoor environment a simple way of representing these releases is to model them as a discharge of gas from a round hole with a low co-flowing airflow. These assumptions generally lead to worst-case conditions in terms of the flammable gas cloud volume. In practice a higher air speed or air flow from a different direction is likely to lead to smaller gas cloud volumes. This approach allows relatively simple analytical or integral models to be used based on the well-understood behaviour of a free jet, see for example [3,4]. For unobstructed releases in enclosures it would be

---

* Corresponding author. Tel.: +44 0 1298 218133; fax: +44 01298 218840.
E-mail address: matthew.ivings@hsl.gov.uk (M.J. Ivings).

0304-3894/$ – see front matter. Crown Copyright © 2010 Published by Elsevier B.V. All rights reserved.
doi:10.1016/j.jhazmat.2010.08.020

expected that the flammable gas cloud volume would tend towards that of a release outdoors as the enclosure volume increases relative to the size of the release.

While the enclosure air change rate is widely used as a measure of ventilation effectiveness in area classification (e.g. "EI 15" [5], IEC 60079:10-1 [6]), it has distinct limitations. In particular, it does not take into account the distribution of the ventilation throughout the enclosure and for large enclosures a specified air change rate can be very difficult to achieve as it equates to a very high ventilation rate in terms of volume flow rate. This approach therefore, relates the ventilation rate to the size of the enclosure rather than to the processes that take place within it.

Natural ventilation is normally provided in accordance with building engineering standards or specific industry standards such as CIBSE [7], BS 5925 [8] or IGEM/SR/25 [9]. For boiler house design for example, the requirements for cooling and the availability of combustion air dictate the ventilation design. The ventilation requirements of current area classification standards specified in IEC 60079:10-1 [6] have not generally been applied to enclosure ventilation design in relation to low-pressure gas because there were no requirements for zoning of low-pressure systems before the implementation of DSEAR.

The work presented in this paper was carried out as part of a Joint Industry Project (JIP) that was set up to provide data to form the basis of an area classification methodology for low-pressure natural gas systems [10]. The work was carried out by the Health and Safety Laboratory and led to the development of a more rigorous approach to assessing ventilation effectiveness based on the behaviour of high momentum jet releases in enclosures. The implications of the work for area classification have been described by Ivings and Santon [11] and the current paper focuses on assessing ventilation effectiveness. The aims of the JIP were achieved by carrying out Computational Fluid Dynamics (CFD) simulations of gas releases in a ventilated enclosure and considering the effects of varying enclosure volume, leak location/direction, ventilation rate, gas releases rate and thermal effects. To provide confidence in the CFD model predictions, the model was validated against 32 experiments where gas was released and measured in a ventilated enclosure. These experiments are described in Section 2 of this paper and the details of the CFD modelling and model validation are given in Section 3. The overall results and discussion are provided in Section 4.

2. Experimental measurements

2.1. Introduction

The aim of the experiments was to validate and provide confidence in the CFD model of gas leaks in enclosures. The variables investigated covered the range of conditions of interest in terms of hole size, gas pressures, ventilation rates, leak rates and leak location/orientation.

Testing took place in a specially constructed chamber with the ability to vary mechanical ventilation rates between 2 and 24 air changes per hour (ach). The tests were carried out based on three different configurations of the leak location/direction and obstacle location as shown in Fig. 1. In Configuration 1, the nozzle directed the gas release into the centre of the room, in Configuration 2 it was aimed along a wall and in Configuration 3 the nozzle was placed inside a narrow cavity formed between a rectangular cuboid obstruction and the wall.

The detailed design of the third configuration was selected following a series of CFD simulations with different room configurations which showed that it provided a credible "worst-case" scenario, i.e. the leak location/direction leading to the largest gas cloud for given ventilation conditions and leak rate.

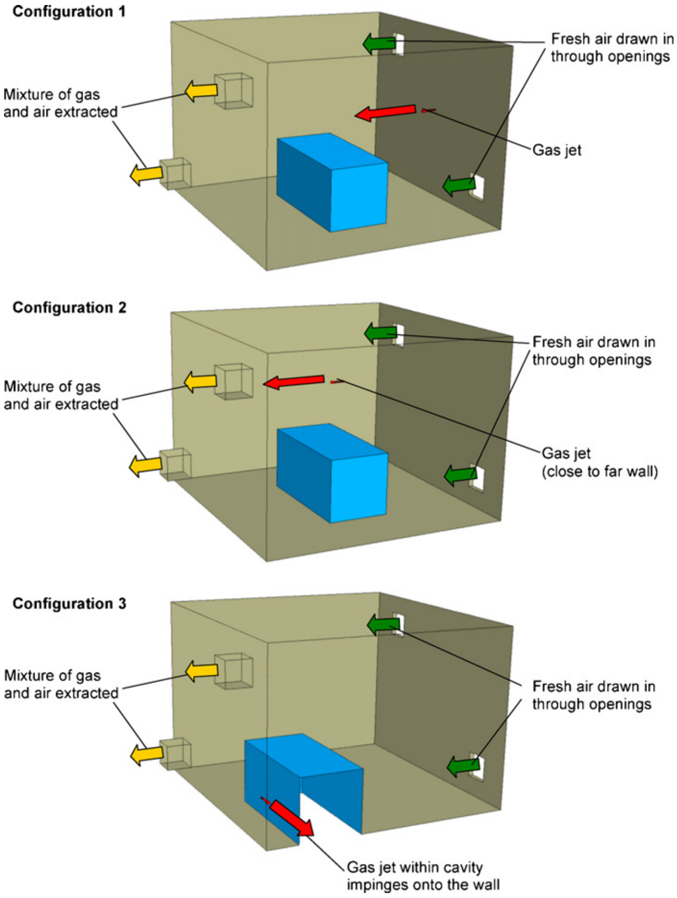


Fig. 1. Arrangements of the room, nozzle and obstruction for Configurations 1, 2 and 3.

For each configuration the hole sizes, ventilation rates and gas leak rates were varied leading to a total of 32 distinct tests, see Table 1. During each test the temperature and volume flow rate of air into and out of the room was logged. Gas concentration measurements were made at 14 locations, which formed the basis of the CFD model validation, and temperatures were also logged at a further eight locations.

2.2. Approach and experimental details

A tracer gas was released at a specified rate through a nozzle with a cross-sectional area of either 0.25 or 2.5 mm^2 into the test enclosure with a known ventilation rate. The tracer gas was a mixture of 1% isobutylene (iso-C_4H_8), 48% nitrogen and 51% helium. The mixture was selected to have the same mean molecular mass and density as methane. The isobutylene was the detectable component of the tracer gas and could be measured to an accuracy of 0.1 ppm at concentrations below 100 ppm and 1 ppm above 100 ppm.

All tests were conducted within the enclosure with internal dimensions 4 m × 4 m × 2.92 m high, located within a climate-

Table 1
Parameters and their range considered in CFD model validation cases.

Three configurations of release location/direction as shown in Fig. 1
Two hole size, 0.25 and 2.5 mm^2
Gas release rates equivalent to between 0.15 and 1.72 g s^{-1} of methane
Air change rates between 2 and 24 ach
Gas supply pressure between 0.06 and 10 barg

controlled laboratory. The enclosure had two inlets and two outlets with dimensions 0.4 m × 0.4 m located 0.5 m from the sidewalls, one at 2.3 m and one at 0.3 m from the floor and diagonally opposed on opposite walls. The volume flow rate at each inlet and extract was monitored using flow grids or orifice plates, both measuring differential pressure. To produce a uniform velocity at the two air inlets, the airflow in the 0.4 m × 0.4 m square section passed through a perforated plate with 25% open area (with the central portion blocked off), followed by a layer of porous foam and finally through a section of honeycomb flow-straightener. The latter ensured that the air entered the room perpendicular to the wall. The flow through the two inlets, and also the two outlets, were balanced so that the volume flow rate through each one was the same as the other. The uniformity of the airflow was verified by velocity measurements. The enclosure included a 2 m × 1 m × 1 m obstruction, which was positioned according to the configuration being investigated.

The test gas was supplied from a cylinder through a mass flow controller to the release head. Two release heads were used; a 2.5 mm^2 venturi designed nozzle and a 0.25 mm^2 nozzle.

Before sampling of the tracer gas, the gas concentrations in the enclosure were allowed to reach equilibrium. To determine when steady state conditions had been reached, gas concentrations were measured in real-time at two positions, one in the lower extract duct and the second inside the enclosure. Once a steady state was reached, which took between 17 and 95 min depending on the air change rate, gas was sampled from twelve predetermined positions for each configuration for ten minutes into gas sample bags. Sampling was carried out for this duration to allow sufficient gas to be collected to accurately measure the concentration. After the sampling period, the test was ended and the concentration in each bag was measured to determine the mean concentration of tracer gas at each point.

Measurements of gas concentrations in the sample bags were made using two MiniRae 2000 photo ionisation detectors (PIDs) and the two results were averaged. Each PID was calibrated with 100 ppm 'isobutylene in air' span gas on each day of testing and then checked against the span gas before and after each test.

3. CFD modelling

3.1. CFD modelling approach

All of the CFD simulations were performed using the general-purpose CFD code ANSYS CFX11.0. The model geometries used for the validation cases are shown in Fig. 1. To match the experimental arrangement, the ventilation velocity was specified at the face of the extract ducts and air was pulled in through the two inlets. The extract velocity was calculated from the prescribed air change rate, the room volume and the cross-sectional area of the inlets/outlets, taking into account the 2 m × 1 m × 1 m obstruction in the room. For a ventilation rate of 12 ach, this gave an inlet velocity of 0.47 $\mathrm{m\,s^{-1}}$. In the scenarios modelled where the ventilation rate was 2 ach, in order to balance the flow rate through the two inlets, the ventilation rate through the upper inlet was fixed in addition to the ventilation rate through the two outlets.

For the simulations involving choked gas releases (pressures above 0.85 barg), a pseudo-source approach was used where the gas was released at the local speed of sound through an opening downstream of the actual orifice at the point at which the pressure had dropped to ambient. This approach was adopted to avoid the difficulties of modelling the highly under-expanded region immediately downstream of the release point. The conditions at the pseudo-source were specified on an inlet boundary following the approach described by Ivings et al. [12,13]. For the 0.86 $\mathrm{g\,s^{-1}}$ release with a supply pressure of 1 barg through a 2.5 mm^2 orifice, which will subsequently be referred to as the baseline release, the cross-sectional area of the pseudo-source was 2.7 mm^2.

All of the walls were treated as adiabatic (i.e. perfectly insulated) and air entered the room at a temperature of 20 °C. Depending on the nature of the gas release (choked/subsonic), the gas was released into the room at different temperatures. For the 0.86 $\mathrm{g\,s^{-1}}$ baseline case, the gas temperature was −18 °C.

Turbulence was modelled using the industry-standard Shear-Stress Transport (SST) model in conjunction with ANSYS CFX's automatic wall treatment, which switches from a low-Reynolds-number treatment to a wall-function approach depending upon the near-wall resolution. The sensitivity of the model predictions to the turbulence treatment have been explored elsewhere [14]. Variation of fluid parameters (density, viscosity, etc.) due to changes in temperature and gas composition were accounted for in the model. In addition to the buoyancy force term in the momentum equations, buoyancy modifications were also incorporated into both production and dissipation terms in the turbulence transport equations.

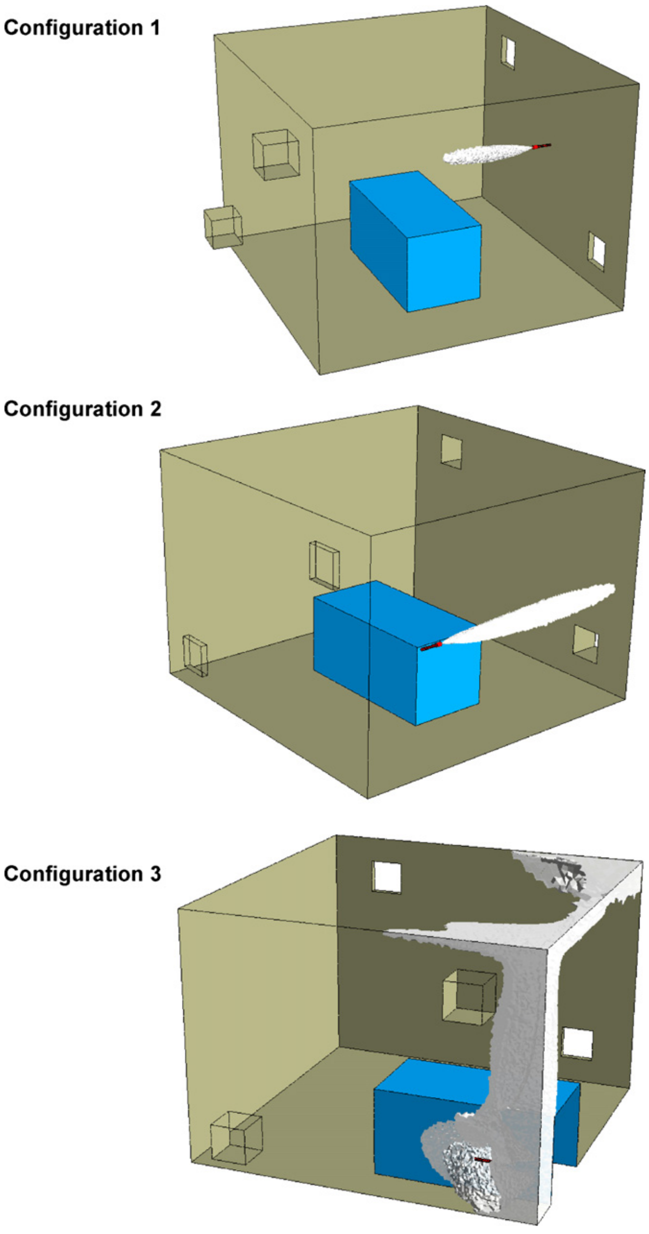
The computational grids comprised a mix of tetrahedral cells with prism layers on walls and refinement in regions where there were significant gradients in flow parameters. To assess grid sensitivity, three different grid resolutions were tested for each of the three room configurations, with the number of nodes in each mesh roughly doubling with each level of successive refinement. The finest grids tested comprised 583,000, 1,325,000 and 939,000 nodes for Configurations 1, 2 and 3, respectively. The grid-sensitivity study showed that results were adequately grid-independent and provided an appropriate balance of accuracy and computational cost. Further details are provided by Ivings et al. [10] and Gant [14].

3.2. CFD model validation

The CFD modelling approach described above was used to simulate the 32 experimental tests. The key parameter of interest was the volume of flammable gas. A conservative measure of the flammable gas volume has been used based on the volume of gas with an *average* gas concentration of half the Lower Explosive Limit (LEL). This gas cloud volume, referred to as 'Vz', is defined in IEC 60079:10-1 [9] and has been used here as it has been accepted for use in area classification. As it is very difficult to measure experimentally a gas cloud volume, the basis of the model validation was therefore a comparison of the gas concentration predictions in the region of the expected position of the gas cloud. In the experiments the gas concentrations were measured along the centreline of the jet axis and at varying positions offset from the axis. Typically 14 measurements were made per test with gas concentrations typically in the range 1–10% (v/v), where the LEL of methane is 4.4% (v/v) [15].

The Vz gas clouds predicted by the CFD model for Configurations 1, 2 and 3 are shown in Fig. 2. In each case the ventilation rate is 12 ach and the gas release rate is 0.86 $\mathrm{g\,s^{-1}}$, equivalent to a release through a 2.5 mm^2 hole at a pressure of 1 bar. For Configuration 3, the gas jet is confined within a narrow cavity in one corner of the room where the flow recirculates, giving rise to locally high gas concentrations. The buoyant gas then rises in a plume towards the ceiling. This was the most challenging case to model of the three configurations, owing to the strong interaction between the jet and the neighbouring surfaces, and the presence of flow recirculation and buoyancy effects.

Comparisons of CFD results versus experimental measurements for two cases are presented as coloured contours of gas concentration in Fig. 3. In each plot, black dots mark the location of the experimental measurements. Around each black dot is a circular fringe, the colour of which denotes the gas concentration mea-


Configuration 1

Configuration 2

Configuration 3

Fig. 2. Predicted Vz clouds for Configurations 1, 2 and 3 with a gas release rate of 0.86 g s^{-1} and ventilation rate of 12 ach.

sured experimentally. The coloured contours in the background are the CFD results, taken from a snapshot once the simulations have reached a fully-developed state, typically after around 2000 iterations. Adjacent to each black dot are given the gas concentrations in terms of percentage gas by volume for both the experimental and mean CFD values. The CFD numerical value of concentration is a time-averaged mean value since this provides the appropriate basis for comparison against the measurements, which were also time-averaged. The two cases shown in Fig. 3 have been chosen as providing indicative results for a typical case (Configuration 1, with a gas release rate of 0.49 g s^{-1} and ventilation rate of 12 ach), and one where there is worse agreement between the predictions and measurements (Configuration 3, with a gas release rate of 0.86 g s^{-1} and ventilation rate of 12 ach). Similar comparisons for all of the cases examined in the present study are provided by Ivings et al. [10].

Numerical values of the predicted gas concentrations for all of the cases are compared to the measurements in Fig. 4. The vast majority of the results are within a factor 4/3 of the measured results. In terms of absolute gas concentrations by volume, Fig. 5 shows that 90% of the CFD predictions were within 0.6% (v/v) of the measured values, and over half of the predictions were within 0.3% (v/v). Similar results were obtained for Configuration 2 whilst for the simpler free-jet flow in Configuration 1, 90% of the predictions were within 0.4% (v/v) of the measured values. These errors are small compared to the LEL of methane. More detailed results comparisons are provided by Ivings et al. [10].

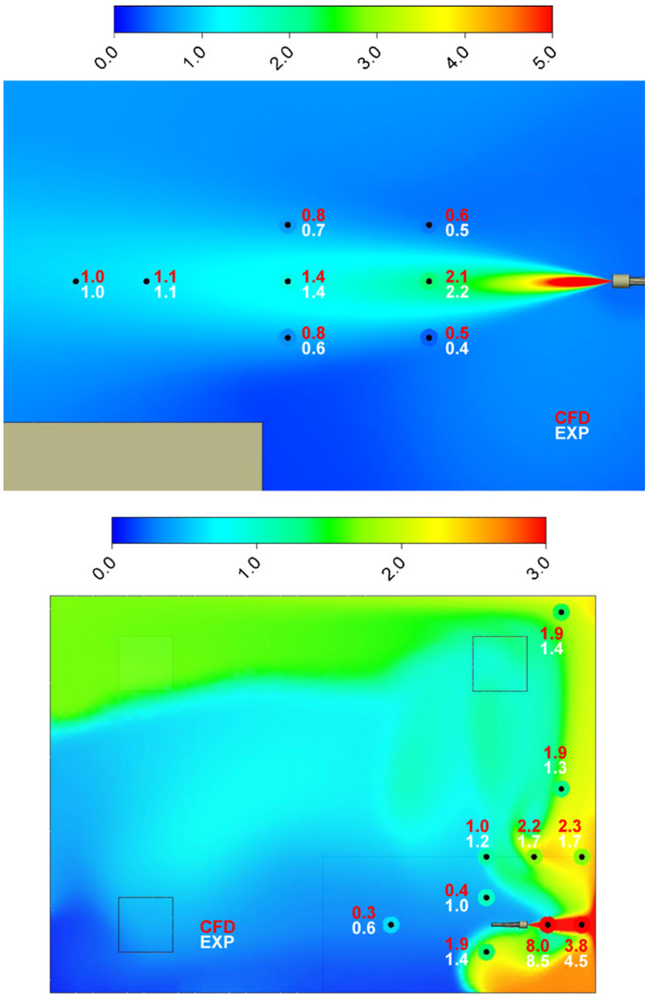


Fig. 3. Comparison of predicted and measured gas concentrations (%, v/v) for Configuration 1 with a gas release rate of 0.49 g s^{-1} and ventilation rate of 12 ach, and Configuration 3, with a gas release rate of 0.86 g s^{-1} and ventilation rate of 12 ach.

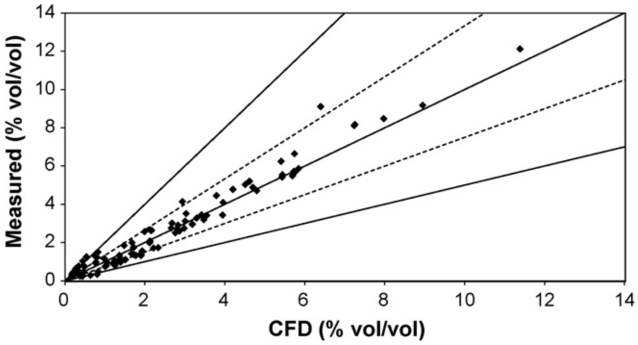


Fig. 4. Measured and predicted gas concentrations (%, vol/vol) for Configuration 3. Solid lines indicate a difference of a factor of 2 and dotted lines a factor of 4/3.

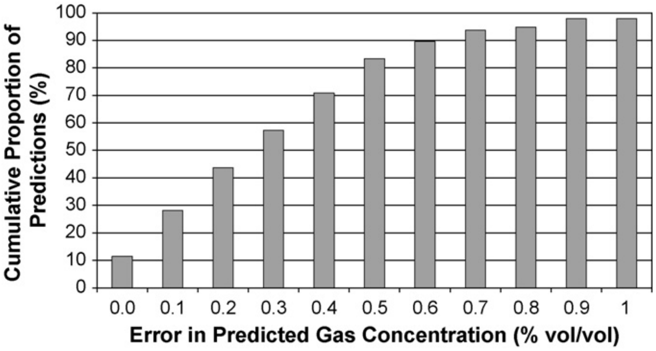


Fig. 5. Proportion of CFD predictions at the measurement points with errors less than or equal to the given value for Configuration 3.

3.3. *Sensitivity studies*

Having validated the CFD model against the experimental data, further CFD simulations were undertaken to assess the sensitivity of the predicted gas cloud volume to the enclosure size, the gas leak location, orientation and release rate, the ventilation rate, and the presence of hot or cold surfaces. Four different enclosure sizes were modelled with volumes of 1, 8, 45 and 400 m^3, and three non-isothermal configurations were examined: a cold floor, and a hot boiler-sized obstruction with or without insulated enclosure walls. In total 66 simulations were performed. The conditions used in these simulations were based on those of interest for the area classification of small unintentional releases. Therefore the leak rates that were considered were all less than 2 g s^{-1} of methane. The ventilation rates were chosen based on those that would be expected for an enclosure of that size.

Predictions for a large enclosure featuring a hot boiler-shaped obstruction with a surface temperature of 70 °C are compared to those obtained using the same geometry but with adiabatic wall surfaces in Fig. 6. In both cases, the gas release rate was 0.86 g s^{-1} and the ventilation rate 6 ach. A significantly larger Vz cloud was predicted in the non-isothermal case.

The results from the sensitivity studies suggested that the largest gas clouds are produced when the leak source is located in a tightly confined space. Here, as the jet impinges onto nearby surfaces it produces a local flow recirculation which causes gas to be re-entrained into the source region. This leads to a build up of gas near the leak and in some cases produces a low-momentum buoyant plume of gas rising from the confined space. The worst-case scenario comprises strong thermal stratification with higher temperatures near the ceiling. In this case the buoyant gas plume reaches a height where its density matches that of the surrounding warm air whereupon it spreads horizontally in a layer across the enclosure. The thermal stratification strongly damps turbulent mixing in the vertical direction, with the result that the gas cloud is much less effectively diluted than under isothermal conditions.

4. Results/assessment of ventilation effectiveness

One of the key tasks of the JIP [10] was to use the CFD data to determine an appropriate measure of ventilation effectiveness that would be suitable for use in area classification. The data provide gas cloud volumes that can be compared against a wide range of factors such as the gas release rate, air change rate and ventilation rate. The intended purpose of the measure of ventilation effectiveness is to be able to distinguish between cases that lead to significant gas cloud build up and those that do not. Given that this measure is intended to be specific to a single release (in the context of area classification) then clearly the measure will depend on the size of the release and on the ventilation flow rate. The measure may also depend on the enclosure volume and other factors such as the release temperature and density, and the ambient conditions. Other factors that have an effect on the gas cloud build up are harder to quantify, such as thermal stratification and the release location/direction and therefore have not been considered as part of the measure. Buoyancy has also been neglected based on the assumption that for the majority of releases a jet release of methane will dilute quickly.

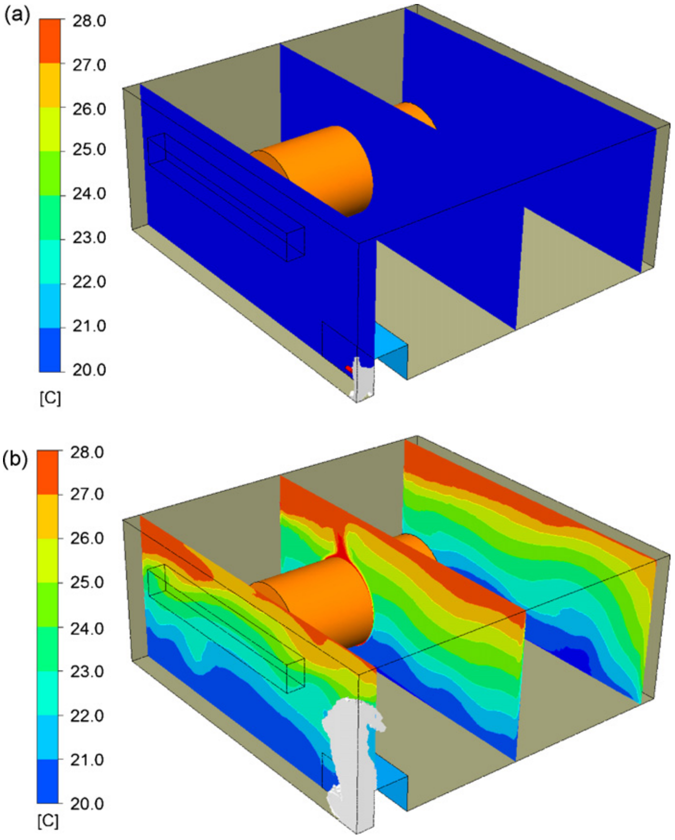


Fig. 6. Vz gas cloud and contours of temperature for a large enclosure with (a) adiabatic surfaces, (b) a boiler-sized obstruction with a surface temperature of 70 °C.

To assess ventilation effectiveness, IEC 60079:10-1 [9] introduces the concept of a hypothetical gas cloud with an average concentration of half LEL and presents a simple method to calculate its volume (Vz) based on the mass release rate of gas divided by the air change rate of the enclosure. The calculation method is flawed, however, and even in the simplest cases, the Vz it produces have been shown to be in error by three orders of magnitude [16,17]. An alternative approach [10] is to instead assess ventilation effectiveness by considering the average flammable gas concentration across the ventilation outlet(s), which is equivalent to the average gas concentration within the enclosure (ignoring any differences in gas temperature). This parameter is a function of the mass release rate divided by the ventilation rate, rather than the air change rate. The ventilation effectiveness measures specified by IEC 60079:10 [6], i.e. Vz, and presented by Ivings et al. [10], i.e. the average gas concentration at the outlet, differ principally by a factor equal to the enclosure volume.

The CFD predictions of the gas cloud volume Vz are compared to these two parameters, the 'IEC 60079:10-1 [6] calculated Vz' and the average gas concentration at the outlet, in Figs. 7 and 8 respectively. The key in each figure indicates the enclosure size (very small = 1 m^3, small = 8 m^3, medium = 45 m^3, large = 400 m^3) and the orientation of the leak (C1 = free jet into the middle of the enclosure, C2 = jet along a side wall and C3 = jet in the corner of the room

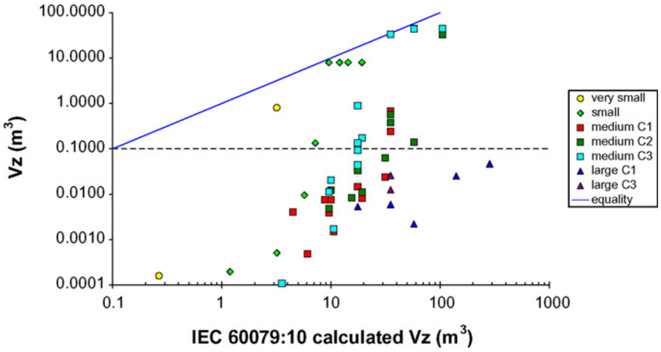


Fig. 7. CFD model predictions of gas cloud volume Vz against IEC 60079:10 calculation of Vz (a function of gas mass release rate divided by enclosure air change rate). Key: Enclosure volume: very small = 1 m^3, small = 8 m^3, medium = 45 m^3, large = 400 m^3. Orientation of the leak: C1 = free jet into the middle of the enclosure, C2 = jet along a side wall and C3 = jet in the corner of the room behind an obstruction.

behind an obstruction). The horizontal line in Figs. 7 and 8, at 0.1 m^3, indicates the size of Vz below which the gas cloud build up can be considered ‘safe’ as defined in IEC 60079:10-1 [6] and confirmed by Ivings et al. [10]. The aim therefore of the measure of ventilation effectiveness is to be able to distinguish between cases that lead to gas cloud volumes above and below this line.

Examining first the results for the small (8 m^3) enclosure, for low release rates or high ventilation rates, i.e. on the left hand side of the graphs, the gas cloud volume is not particularly sensitive to these two parameters and the jet behaves like a free unobstructed jet outdoors. As the release rate increases or the ventilation rate decreases, the gas cloud grows until it fills the enclosure and becomes equal to the enclosure volume. The results also show that the release location and direction have a significant effect on the gas cloud build up. Where the release location is in a corner of the room behind a large obstruction, the gas cloud volume is up to two orders of magnitude larger.

The results shown in Fig. 7 clearly indicate that the method used to calculate Vz described in IEC 60079:10-1 [6] does not provide a reasonable estimate of the gas cloud volume when compared to predictions from a validated CFD model. In fact the only cases where the agreement is reasonable, where the points lie near the line of equality in Fig. 7, is where Vz fills the entire enclosure. The calculated Vz values are in some cases up to four orders of magnitude larger. The use of Vz calculated using the IEC 60079:10-1 method as an indicator of ventilation effectiveness does not distinguish between the cases where gas cloud build up occurs and where it does not. The calculated Vz is heavily influenced by the enclosure volume as it is based on the air change rate rather than the ventilation rate. It can clearly be seen that for large enclosures, the Vz produced using the IEC 60079:10 method is correspondingly large.

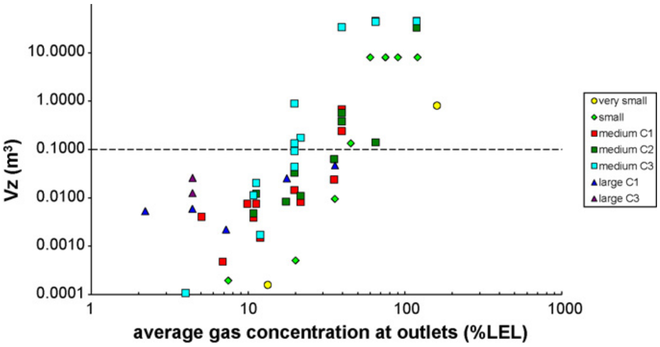


Fig. 8. CFD model predictions of gas cloud volume Vz against the average gas concentration at the ventilation outlet (a function of gas mass release rate divided by ventilation rate). Key: Enclosure volume: very small = 1 m^3, small = 8 m^3, medium = 45 m^3, large = 400 m^3. Orientation of the leak: C1 = free jet into the middle of the enclosure, C2 = jet along a side wall and C3 = jet in the corner of the room behind an obstruction.

The average gas concentration at the outlet provides a more consistent measure across the different enclosure volumes, although it does appear to be less conservative for large enclosures. The results in Fig. 8 indicate that, in the cases considered, if the average gas concentration at the outlet is less than about 20% LEL, then the gas cloud volume Vz remains below 0.1 m^3. The results from the CFD simulations where thermal effects were included in the model are not shown in Figs. 7 and 8. These results showed that the presence of a heat source in the enclosure or a cold floor could lead to thermal stratification and hence reduce the mixing within the room. In some cases the gas cloud volume Vz was up to three orders of magnitude larger in these cases, particularly where the thermal stratification was coupled with a release in a confined location.

5. Conclusions

This paper has considered the effect of the ventilation in an enclosure on the size of the flammable gas cloud resulting from a small leak of natural gas. The aim of the work was to develop a better understanding of this interaction and to suggest an approach for assessing ventilation effectiveness that can be used for hazardous area classification.

The results of the study have shown that the flammable gas cloud volume can be related to the average gas concentration at the ventilation outlets. Current approaches to hazardous area classification that rely only on the enclosure air change rate to calculate the gas cloud volume may not be properly assessing the ventilation effectiveness. This work has suggested that instead the average gas concentration at the outlet should be used to assess the ventilation effectiveness, which is calculated using the ventilation rate rather than the air change rate. A simple dimensional analysis shows that the air change rate can only provide the timescale for reaching steady state conditions.

The presence of obstacles near to the source of a gas release can have a significant effect on the flammable gas cloud volume as it can remove some of the initial jet momentum and provide a mechanism for gas becoming re-entrained into the jet. This factor is more important to consider in large enclosures where the ventilation distribution can be far from uniform and the air is not well-mixed. In large enclosures even a low air change rate will generally provide sufficient air to dilute a gas leak to safe levels. Therefore, in such cases an assessment of the ventilation should focus on the ventilation *local* to the release location. This can be done, for example, by carrying out air speed measurements or smoke tests. In the absence of significant obstruction or congestion a gas release in a large enclosure will behave similarly to a release outdoors.

Acknowledgements

The authors of this work would like to acknowledge the financial support of the sponsors of this joint industry project: Health and Safety Executive, Burgoyne Consultants, Corus Construction and Industrial, Danish Gas Technology Centre, Directorate for Civil Protection and Emergency Planning, EnergieNed, EON-UK, Epsilon Compliance (Europe), Gastec Technology, Gasunie, Hamworthy Combustion, Knauf Insulation Ltd., National Grid, Northern Gas Networks, SGL Technic Ltd., Slough Heat and Power.

We also gratefully acknowledge the input to this work provided by Roger Santon and Simon Clarke (HSL), the project sponsors, Barrie Church and Geoff Winckles (Global Energy Associates Ltd.), Chris Lea (Lea CFD Associates) and Malcolm Howe (IGEM).

References

[1] DSEAR Dangerous Substances and Explosive Atmospheres Regulations, SI 2002/2776, 2002.

[2] R.P. Cleaver, M.R. Marshal, P.F. Linden, The build-up of concentration within a single enclosed volume following a release of natural gas, J. Hazard. Mater. 36 (3) (1994) 209–226.

[3] E.J. List, Mechanics of turbulent buoyant jets and plumes, HMT, in: W. Rodi (Ed.), The Science & Application of Heat and Mass Transfer, vol. 6, Turbulent Buoyant Jets and Plumes, Pergamon Press, Oxford, 1982, pp. 1–68.

[4] A.D. Birch, D.R. Brown, M.G. Dodson, F. Swaffield, The structure and concentration decay of high pressure jets of natural gas, Combust. Sci. Technol. 36 (1984) 249–261.

[5] EI15, Model Code of Safe Practice Part 15 – Area Classification Code for Installations Handling Flammable Fluids, third ed., Energy Institute, London, 2005.

[6] IEC 60079-10-1, Electrical apparatus for explosive gas atmospheres. Part 10. Classification of hazardous areas, 2009.

[7] CIBSE, The Chartered Institution of Building Service Engineers, Guide B: Heating, Ventilating, Air Conditioning and Refrigeration, CIBSE, 2005.

[8] BS 5925:1991, Code of Practice for: Ventilation Principles and Designing for Natural Ventilation, British Standards Institute, 1991.

[9] IGEM/SR/25, Hazardous Area Classification of Natural Gas Installations, The Institution of Gas Engineers and Managers, 2000.

[10] M.J. Ivings, S. Clarke, S.E. Gant, B. Fletcher, A. Heather, D.J. Pocock, D.K. Pritchard, R. Santon, C.J. Saunders, Area classification for secondary releases from low pressure natural gas systems, Health and Safety Executive Research Report RR630, 2008.

[11] M.J. Ivings, R. Santon, Area classification of natural gas installations, in: Proc. IChemE Hazards XXI Symposium and Workshop, Manchester, UK, 2009.

[12] M.J. Ivings, C.J. Lea, H.S. Ledin, D. Pritchard, R. Santon, C.J. Saunders, Outstanding safety questions concerning the use of gas turbines for power generation – executive report, Health and Safety Laboratory Report CM/04/02, 2004.

[13] M.J. Ivings, C.J. Lea, H.S. Ledin, C. Carey, M. Azhar, C. Skinner, Y. Sinai, P. Stephenson, Outstanding safety questions concerning the use of gas turbines for power generation – final report on the CFD modelling programme of work, Health and Safety Laboratory Report CM/03/08, 2004.

[14] S.E. Gant, Reliability issues of LES-related approaches in an industrial context, Flow Turbul. Combust. 84 (2010) 325–335.

[15] PD IEC 60079-20:2000, Electrical apparatus for explosive gas atmospheres. Part 20. Data for flammable gases and vapours, relating to the use of electrical apparatus.

[16] S.E. Gant, M.J. Ivings, A. Jones, R. Santon, Hazardous area classification of low pressure natural gas systems using CFD predictions, in: Proc. IChemE Hazards XIX Symposium and Workshop, Manchester, UK, 2006.

[17] S.E. Gant, M.J. Ivings, CFD modelling of low pressure jets for area classification, Health and Safety Laboratory Report HSL/2005/11, 2005.