Investigation of Cargo Tank Vent Fires on the GP3 FPSO, Part 2: Analysis of Vapour Dispersion

Simon Gant*, Mark Pursell, Andrew Newton, Darrell Bennett and Louise O'Sullivan, Health and Safety Laboratory, Harpur Hill, Buxton, SK17 9JN, UK

David Piper, Maersk Oil North Sea UK Limited, Maersk House, Crawpeel Road, Altens, Aberdeen AB12 3LG

* Corresponding author: simon.gant@hsl.gsi.gov.uk, 01298 218134

This paper presents the second part of the analysis of the ignition incidents on the Global Producer III (GP3) Floating Production, Storage and Offloading (FPSO) vessel. A companion Hazards 26 paper (Part 1) presents the background to this work.

To assess the likelihood that the fires on the GP3 were caused by flammable vapour from the cargo vent being drawn into the flare, a programme of indoor and outdoor gas dispersion experiments and Computational Fluid Dynamics (CFD) modelling was conducted at the Health and Safety Laboratory (HSL).

The indoor experiments involved small-scale releases of butane in HSL's Burn Hall and ignition tests using a spark igniter. The outdoor experiments involved a 1:10 scale model of the GP3 flare tower, where the cargo vent vapour was released part-way up the tower and the flare was simulated by a propane jet fire. CFD model predictions were produced for both sets of experiments and the predicted gas concentrations were compared to the ignition measurements.

An important finding was that the vapour cloud was ignited in the experiments at locations where the CFD model predicted the vapour concentrations to be below half the lower explosive limit, or for there to be no vapour present at all. This result appeared to be a consequence of the strong influence of large-scale atmospheric turbulence (gusts of wind). The CFD model predicted the average flow behaviour and failed to account fully for time-varying fluctuations in concentration.

A simple, pragmatic method is proposed to account for these effects, which produces results that agree with all of the ignition measurements, although it is conservative. Results are presented using this method for the vapour dispersion from the cargo vent on the GP3.

Keywords: FPSO, cargo vent, vapour cloud, ignition, dispersion experiments, CFD

Introduction

This paper presents the second part of the analysis of the ignition incidents on the GP3. A companion paper at the Hazards 26 conference by Pursell *et al.* (2016) provides an introduction to the study. The four incidents on the GP3 all involved ignition of vapour released from the cargo tank vent, which is located part-way up the flare tower (Figure 1).

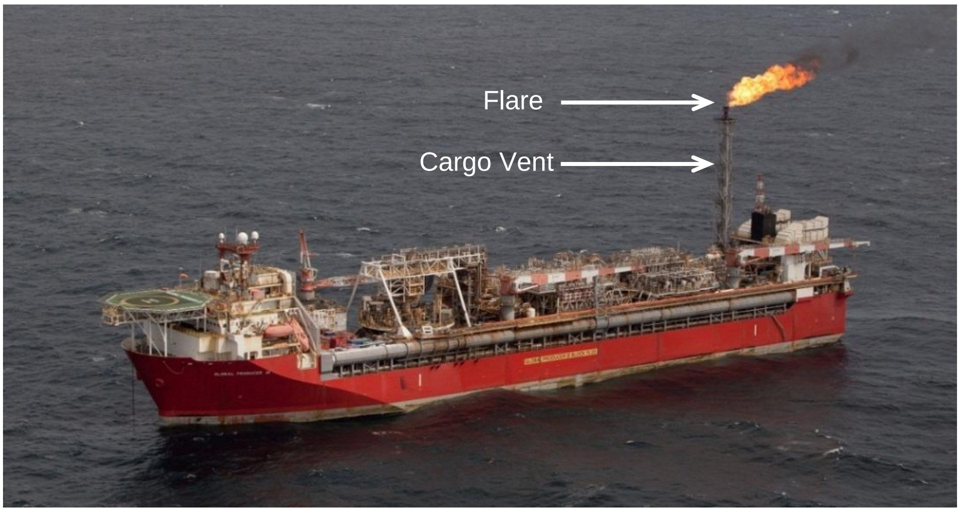

Figure 1 Location of the flare and the cargo vent on GP3 (Copyright: Maersk Oil)

Six potential ignition mechanisms were identified in the initial phase of the HSL investigation. Of these, two mechanisms (thermal radiation from the flare and electrical sparks) were soon discounted and further analysis was performed on the remaining four mechanisms, which included:

1.) The vent gas being drawn up into the flare and igniting
2.) Ignition of a stream of condensed liquid raining-out from the vent pipe
3.) Ignition of the vent gas cloud by burning droplets or hot particles falling from the flare
4.) Electrostatic ignition

The first of these mechanisms is the subject of this paper. Mechanisms 2, 3 and 4 are analysed in the paper by Pursell *et al*. (2016). To address Mechanism 2, Maersk undertook remedial measures in 2014 which prevented liquids condensing within the cargo vent pipe. However, a further ignition incident took place subsequently in 2015. Mechanism 3 was investigated experimentally by HSL and details of the tests are given by Pursell *et al*. (2016). The conclusion of that work was that burning droplets falling from the flare are deemed to be a credible cause of the incidents. Analysis of Mechanism 4 showed that a brush discharge from an insulating liquid slug cannot be ruled out on a worst-case analysis but it is highly unlikely, and all other electrostatic ignition sources were considered either not feasible or as having a negligible risk.

The present paper proceeds as follows. Background information on the flammability of gas jets is reviewed in the next section. Some initial CFD modelling of the GP3 is then analysed and results are presented from the indoor and outdoor gas dispersion experiments at HSL. The implications of these results are then discussed and a final set of CFD model predictions are presented for the GP3. Conclusions of the investigation are presented in the final section.

British Gas Research on Flammable Gas Jets

To help assess whether gas from the cargo vent was drawn up into the flare in the GP3 incidents, previous research on dispersion of gas jets and their ignition was reviewed. In the late 1970s and 1980s, a significant programme of work was undertaken by British Gas to investigate the flammability of gas jets. Fundamental work was first undertaken to measure concentration fluctuations in natural gas jets using laser Raman spectroscopy by Birch *et al*. (1978, 1979). This was followed by a combination of ignition tests and concentration measurements on free jets of natural gas, propane and town gas in a quiescent environment by Birch *et al*. (1981) and Smith *et al*. (1986). A final study by Birch *et al*. (1989) examined the ignition characteristics and mean concentrations in natural gas jets in a cross-wind.

Birch *et al*. (1981) found that along the centreline of a free jet, the edge of the flammable cloud (where the gas could be ignited and burn back to the source) coincided with a mean gas concentration that was close to the Lower Explosive Limit (LEL). On the periphery of the jet, the flammable cloud extended to a much lower mean concentration of around 10% LEL. These results were found to be consistent for gas release velocities of between 19 m/s and 32 m/s. Subsequently, Smith *et al*. (1986) found that for release velocities above 35 m/s, the extent of the flammable cloud was reduced as the gas velocity was increased.

Birch *et al.* (1989) conducted further tests where they varied the gas release velocity from 29 m/s to 123 m/s and the cross-wind speed from 1.3 m/s to 5.0 m/s. For the same jet exit velocity, increasing the cross-wind speed caused the jet to penetrate less and become more tilted over by the wind, which decreased the flammable cloud size.

One of the difficulties in applying these findings directly to the GP3, in order to define the edge of the flammable cloud, is that the British Gas experiments involved higher release velocities than those on the GP3. The cargo vent discharge velocity on the GP3 varies between 13 m/s and 21 m/s, whereas the British Gas experiments considered release velocities from 29 m/s to 123 m/s. A second difficulty is that the British Gas experiments were all conducted under carefully-controlled laboratory conditions. For their experiments on cross-winds, they used a wind tunnel with a very low background turbulence intensity of just 0.6%. For the GP3, the gas releases take place outdoors and turbulent eddies in the atmosphere or gusts of wind could produce greater variability in the extent of the flammable cloud than was observed in the British Gas tests.

Initial CFD Modelling of the GP3

CFD simulations of gas dispersion from the GP3 cargo vent were performed prior to HSL's investigation by an engineering consultant working for Maersk which showed that gas did not reach ignition sources at flammable concentrations. Further CFD simulations were performed by HSL to examine different release scenarios that had not been previously examined. These included high and low wind speeds, different flaring conditions, and simulations that accounted for heat sources and exhaust outlets located below the cargo vent outlet.

The results from HSL's initial CFD simulations showed that flammable vapour was drawn towards the flare in some cases, but the vapour was diluted to a concentration below 50% LEL before it reached the flare. Although these results suggested that the flare was unlikely to be the cause of the ignition incidents on the GP3, the results also showed that the vapour dispersion behaviour was very complex in very low wind speeds and sensitive to a number of factors, including the presence of hot surfaces, exhaust gases and the flare. The way in which the cargo vent gas dispersed was also considered likely to be affected by time-varying turbulent eddies in the atmosphere that may not have been accounted for well by the CFD model, which only simulated the mean flow behaviour.

To investigate these uncertainties in the CFD model predictions, it was agreed to conduct a series of experiments and further CFD modelling to meet two key objectives:

1) To examine experimentally at reduced scale whether it was possible for flammable vapour from the cargo vent to reach the flare and be ignited.

2) To develop and validate a CFD model for simulating vapour dispersion from the GP3's cargo vent

The experiments and modelling focused on low wind speed conditions since the first three ignition incidents on the GP3 all took place in low wind speed conditions (see Pursell *et al*., 2016).

Small-Scale Indoor Gas Dispersion Experiments

The purpose of the small-scale indoor gas dispersion experiments was to determine the flammable extent of a gas cloud under well-controlled ambient conditions, using a configuration that was similar to the GP3 cargo vent but without the flare being present. The experiments involved releases of butane gas through a 7.2 mm diameter pipe in the HSL Burn Hall (a large rectangular hall with an interior volume of approximately 3000 m^3). Butane gas was released, which has a similar molecular weight and LEL to the cargo vent gas.

The dispersion behaviour in the experiments was visualised using a hydrocarbon gas camera (FLIR GasCam GF320) and ignition tests were performed using a spark igniter at 50 mm intervals (vertically and horizontally) across the region where the edge of the flammable cloud was located. At each ignition test position, up to 10 repeat ignition tests were performed. If ignition occurred at any time during these 10 attempts and the flame burnt back to the source then this was recorded as a positive ignition response.

Figure 2 shows two images taken from the FLIR GasCam using its low and high resolution settings. The images provide a good illustration of the time-varying, intermittent nature of the flow. During the tests, the gas was released with a velocity of only 2 m/s and its dispersion behaviour was observed to be very sensitive to small air currents within the Burn Hall. The low release velocity of 2 m/s was selected to represent a reduced-scale model of the GP3 cargo vent, preserving the same Froude number (i.e. the ratio of inertial to buoyancy forces).

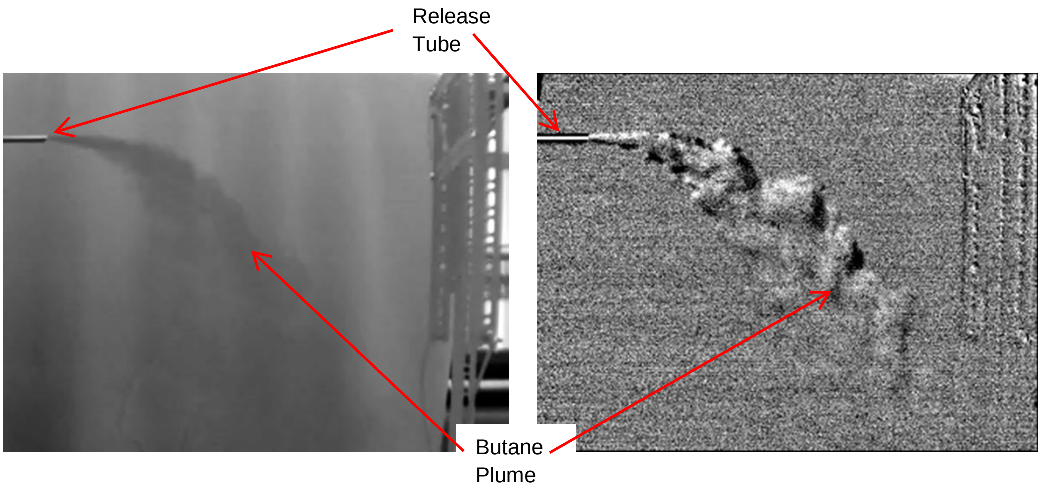

Figure 2 Two FLIR GasCam images of an indoor butane release, showing standard resolution and high resolution

The ignition data was compared to CFD model predictions produced using the ANSYS-CFX software with the SST turbulence model and standard buoyancy corrections. An unstructured grid of 8 million elements was used and tests were performed to assess the effect of the grid resolution.

Figure 3 presents the experimental ignition measurements, where red circles indicate successful ignitions and green circles indicate no ignition. Overlaid upon these results are the mean gas concentrations contours predicted by the CFD model (LEL, 50% LEL and 10% LEL). Also shown in the background of Figure 3 is an image taken from the FLIR GasCam which shows that the observed hydrocarbon plume lies within the LEL region predicted by the CFD model.

Many of the successful ignition locations were found to be located within the 50% LEL gas cloud predicted by the CFD model. However, a significant number of ignition locations occurred outside of the 50% LEL cloud. This was an important finding, since the 50% LEL cloud is often used in risk assessments to define the maximum extent of the flammable cloud (Webber, 2002).

CFD simulations were performed for two different co-flowing wind speeds (0.005 m/s and 0.1 m/s), which showed that the dispersion behaviour was very sensitive to very slight air movements. This result was consistent with the observations in the experiments that very low speed air currents generated within the Burn Hall were sufficient to affect the flow behaviour.

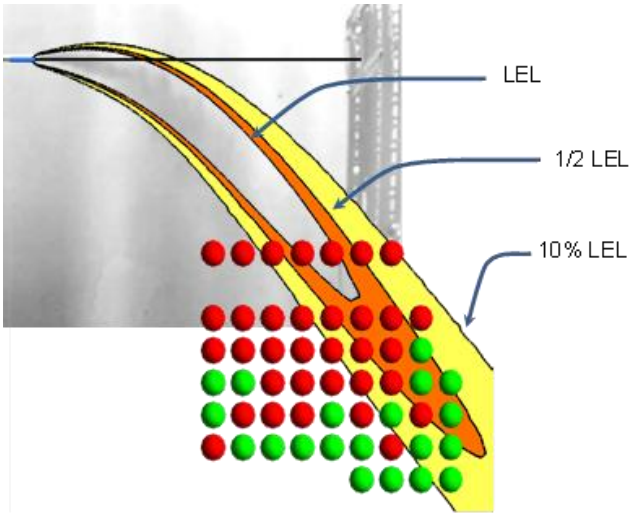

Figure 3 Comparison of hydrocarbon camera image (greyscale) and predicted mean gas concentrations from the CFD model (coloured contours) with the ignition data, where ● indicates successful ignition and ● indicates no ignition. The CFD results shown here used a 0.1 m/s wind flowing in the same direction as the gas release

Outdoor Gas Dispersion Experiments

The purpose of the outdoor gas dispersion experiments was to investigate further the extent of the flammable gas cloud and, in particular, to assess the impact of the flare on the dispersion behaviour. A 1:10 reduced-scale model of the GP3 flare tower and cargo vent was constructed on the HSL site using a scaffolding tower fitted with a flare line riser (Figure 4). The flare was fed with propane gas from a nearby LPG storage tank and vaporiser. Butane gas was released through a horizontal pipe positioned part way up the tower at a location that was representative of the GP3 cargo vent pipe. A second tower was located adjacent to the flare tower to support instrumentation and allow close-up video footage to be taken of the gas release. Spark ignition points were located in the vicinity of the vent pipe to help assess the extent of the flammable cloud. The butane gas cloud was also visualised using the FLIR GasCam. To assist with flow visualisation, the rig was configured so that carbon dioxide (CO_2) gas seeded with smoke could be released through the vent pipe, instead of butane.

A key aspect in designing the reduced-scale experiments was to ensure that the updraft created by the flare had a similar effect on the vapour released from the cargo vent as on the full-scale GP3. Various different approaches were investigated for the scaling parameters. It was concluded that the best approach was to preserve the Froude number in the reduced-scale experiments and maintain the same ratio of the buoyancy flux between the vented vapour and the flare. This scaling approach resulted in a target wind speed in the experiments of less than 0.8 m/s, in order to reproduce the low wind speeds that were representative of those recorded at full scale on the GP3 during the first three ignition incidents (a wind speed of 2.5 m/s or 5 knots).

Two experiments were conducted over relatively long periods (1 to 2 hours) in different wind conditions. For the majority of the time, the gas venting rate and flare flow rate were both set to high values that were representative of the maximum flow rates on the GP3. The flare was turned off for some short periods in order to assess the effect of the flare on the dispersion behaviour. In addition, CO_2 seeded with smoke was released at certain intervals to help visualise the flow behaviour.

In the first experiment, the wind blew mainly in a direction that was opposite to the vent gas release direction (i.e. counter-flowing) and the wind speed was nearly twice the target value of 0.8 m/s. The gas released from the vent was often blown back on itself but there were some short periods when the wind subsided and the vent gases circulated around the area in front of the vent, with the smoke showing the vent gas rising up towards the flare in some instances. The butane gas cloud from the vent was ignited once by the nearest spark igniter located a short distance (33 cm) above and in front of the vent outlet.

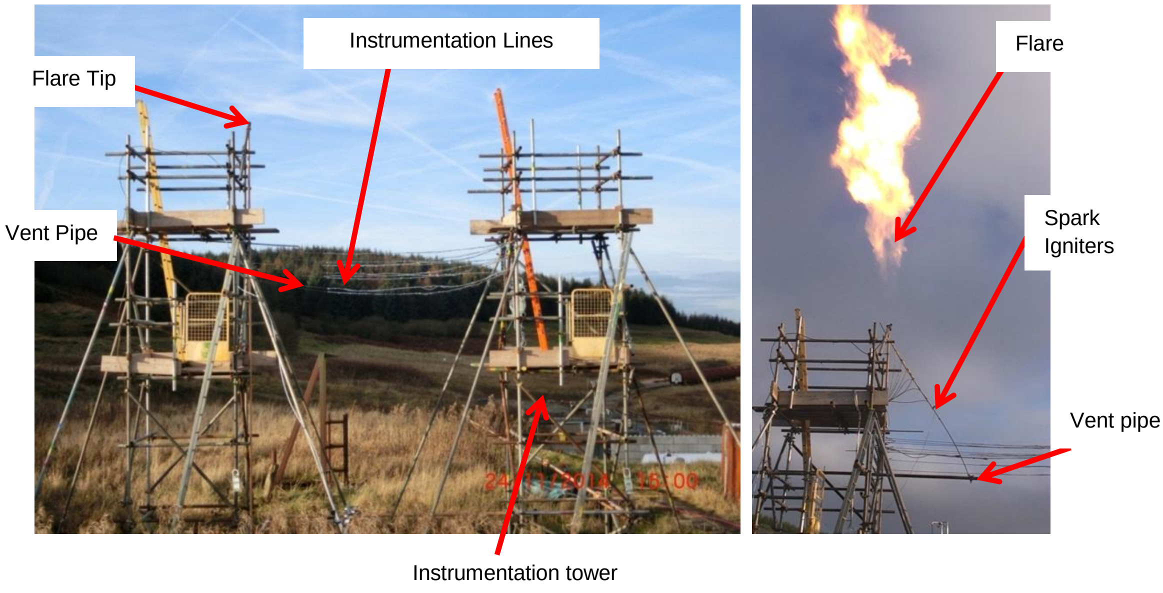

Figure 4 Photos of the outdoor gas dispersion experimental rig

In the second experiment, the wind blew mainly in the same direction as the gas jet from the vent (co-flowing) and the wind speed was slightly lower than the target of 0.8 m/s. There was no noticeable tendency for the gases to be drawn towards the flare and no successful ignitions of the butane gas cloud. Video footage of the two experiments was examined carefully during those periods both with and without the flare in operation. This showed that the behaviour of the gas released from the vent did not appear to be affected by the presence of the flare.

Two sets of CFD simulations were performed to support the experiments: an initial set of simulations to examine the sensitivity of the gas dispersion behaviour to various effects, and a second set of simulations of the actual experimental conditions. The sensitivity tests involved four different wind conditions: a very low wind speed of 0.005 m/s (which represented essentially the 'zero' wind speed case) and three simulations with a wind speed of 0.5 m/s in different wind directions. The flare was modelled using three different flaring rates that corresponded to normal operation and two higher rates, representative of the maximum instantaneous and continuous flaring rates on the GP3.

The CFD sensitivity tests showed that when the wind speed was effectively zero and the flare flow rate was at either of the two higher rates, the vent gases were drawn towards the flare. However, with a co-flowing wind of just 0.5 m/s the flare had minimal effect on the plume and the flow behaviour was instead dominated by the wind.

CFD simulations of the first of the two experiments (Figure 5) showed an important finding: the predicted butane gas concentration was zero at the location where the gas cloud was ignited in the experiment. The model used the mean wind direction that was counter-flowing, which meant that the butane jet was blown back on itself. Even at the high flaring rate, the model predicted that there was no tendency for gas to be drawn towards the flare.

Implications of the Results

The experiments and CFD simulations showed that it was difficult to relate the extent of the flammable gas cloud to a single mean gas concentration (e.g. LEL, 50% LEL or 10% LEL), due to the time-varying turbulent fluctuations in gas concentration that momentarily raised the concentrations at the ignition location to within the flammable range. This finding is consistent with the British Gas research, but the relatively low gas release velocity and presence of atmospheric turbulence meant that the results showed greater variability than the British Gas results.

There are three possible ways in which the effect of concentration fluctuations can be modelled using CFD, to relate the findings from the experiments to the GP3 ignition incidents:

1. Large-Eddy Simulation (LES) can be used to resolve the large-scale, time-varying turbulent flow structures.

   HSL attempted to model dispersion from the cargo vent on the GP3 using LES, but it soon became clear that this was not a viable option, due to the computing time required and problems faced in sustaining a realistic atmospheric turbulent velocity field.

2. The CFD model can be combined with a Probability Density Function (PDF) for the concentration fluctuations.

This approach has been demonstrated to work well for free jets in a quiescent environment, but there are several challenges to be overcome before it can be used reliably for more complex flows, like the GP3, as discussed by Gant *et al*. (2011).

3. The standard CFD modelling approach can be used that predicts the mean concentration, not the fluctuations, and then a pragmatic "safety factor" can be applied to account for the effect of the fluctuations.

   The commonly-used approach is to assume that the edge of the flammable cloud is located where the mean concentration is 50% LEL instead of 100% LEL, where the reduction in concentration by a factor of two is intended to account (in part) for the concentration fluctuations (see Webber, 2002).

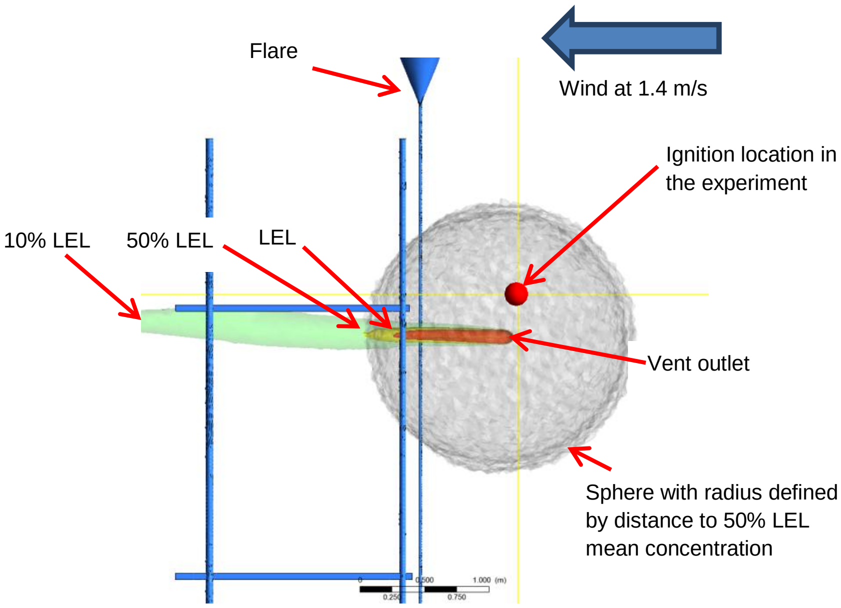

Figure 5 Side view of predicted butane gas cloud in the first outdoor gas dispersion experiment

Options 1 and 2 do not provide practical solutions, whilst the results from the indoor and outdoor HSL ignition experiments demonstrated that using Option 3 with a 50% LEL criterion to define the extent of the flammable cloud did not capture all of the observed ignition locations.

A pragmatic way forward was proposed which involved assuming that flammable gas could occur anywhere within a sphere around the release point, where the radius of the sphere is equal to the distance to the predicted 50% LEL mean concentration from the CFD model results (see Figure 5). It was demonstrated that this sphere enclosed all of the successful ignition locations in both the indoor and outdoor HSL experiments. Using a smaller sphere defined using the LEL rather than the 50% LEL concentration failed to account for some of the ignition locations.

This approach of defining the extent of the flammable gas cloud using a 50% LEL sphere is similar in concept to the spherical regions that are used to define hazardous area classification zones in EI15 (Energy Institute, 2015). In the case of the GP3, this approach provides a practical and simple way of accounting for the strong effect of atmospheric turbulence on the dispersion behaviour of the low-speed gas jet, but it is conservative in the sense that it over-predicts the extent of the flammable cloud.

The use of 50% LEL sphere approach makes the implicit assumption that the characteristics of turbulence on the HSL field test site are representative of turbulence on the GP3. In the experiments, the gas was released at a height of 4.6 m whereas on the GP3 the vent gases are released at a height of 46 m. The size of turbulent eddies in the atmosphere is likely to scale with distance from the ground (or sea), and other differences may arise between the two cases, due to the presence of terrain effects in the field-scale tests and heat sources and the presence of exhaust outlets on the GP3. The net effect on the dispersion behaviour of these differences is uncertain. The only way to fully assess the turbulence characteristics would be to conduct dispersion experiments on the GP3 itself but given the difficulties in conducting such tests, the sphere approach is considered to provide a pragmatic solution.

CFD Analysis of the GP3

The final stage of the investigation into the cause of the GP3 ignition incidents involved CFD modelling of the GP3 at full-scale using the 50% LEL sphere approach described in the previous section to define the extent of the flammable cloud. A model of the GP3 was constructed in ANSYS-CFX using geometry data from a Computer-Aided Design (CAD) file provided by Maersk. The CFD domain was 400 m high, 800 m wide and 2,250 m long and the grid comprised 9.2 million elements. Tests were performed to assess the effect of both the domain size and grid resolution. Further tests were performed using porous regions to assess the effect of small-scale obstructions that could not be fully resolved by the CFD model geometry.

CFD simulations were performed for the second, third and fourth GP3 ignition incidents (wind conditions were not recorded for the first of the four incidents). The results for the second incident are shown in Figure 6 and results of the third incident were similar. The 50% LEL cloud extended to a distance of approximately 25 m, which meant that the 50% LEL sphere reached the flare. A slightly smaller flammable cloud was produced in the fourth incident, which fell marginally short of the flare. However, the sensitivity tests on the grid resolution indicated that the flammable cloud size would probably reach the flare in this case, if a finer grid had been used.

It was therefore concluded that the cargo vent gas cloud could have been ignited in the last three ignition incidents by being directly ignited by the flare. The method used to arrive at this conclusion, involving the 50% LEL sphere concept, is a conservative approach, which may over-estimate the extent of the flammable cloud in certain directions. The flare is also near the outer-most edge of the 50% LEL sphere. It is therefore considered that the likelihood of this ignition mechanism being responsible for all four incidents is relatively low (as compared to the burning droplet ignition mechanism) but it cannot be ruled out.

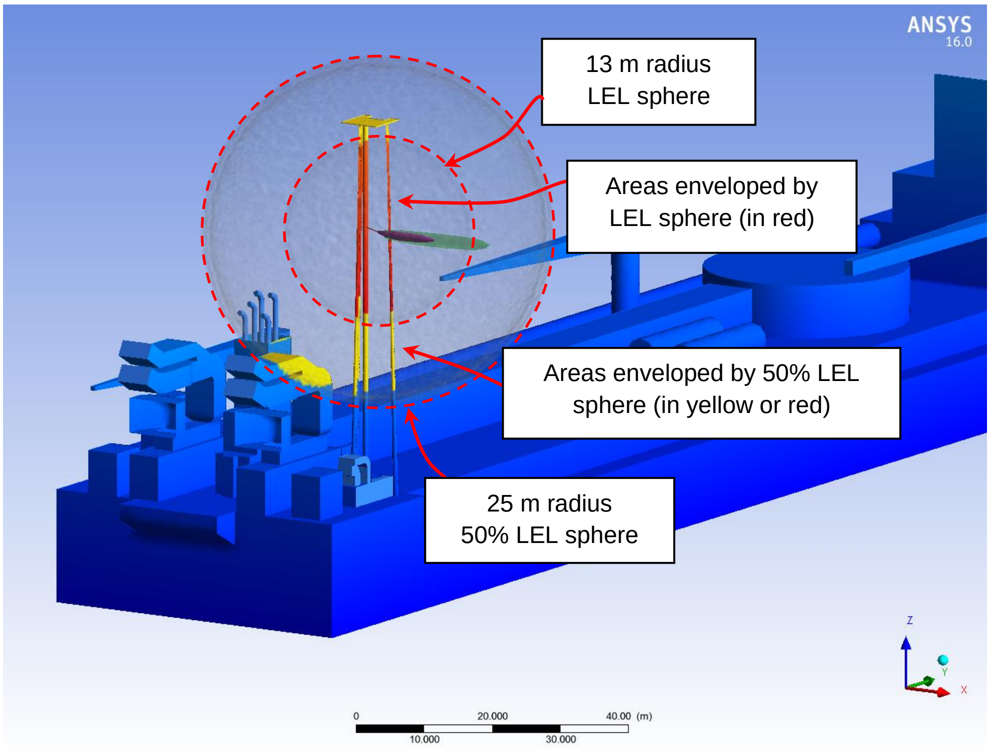

Figure 6 Side view of the predicted 50% LEL and LEL clouds for second GP3 ignition incident

Conclusions

This paper has presented a summary of the gas dispersion analysis performed by HSL to investigate whether the ignition incidents on the GP3 were caused by flammable vapour from the cargo vent being drawn into the flare. The analysis involved indoor experiments with small-scale releases of butane in HSL's Burn Hall and outdoor experiments with a 1:10 scale model of the GP3 flare tower. CFD model predictions were produced for both sets of experiments and the predicted gas concentrations were compared to the ignition measurements.

An important finding was that the vapour cloud was ignited in the experiments at locations where the CFD model predicted the vapour concentrations to be below half the lower explosive limit (50% LEL), or for there to be no vapour present at all. This result appeared to be a consequence of the strong influence of large-scale atmospheric turbulence. The CFD model predicted the average flow behaviour and failed to account fully for time-varying fluctuations in concentration.

A simple, pragmatic method was proposed to account for these effects, where it was assumed that flammable gas could occur anywhere within a sphere around the release point, where the radius of the sphere was equal to the distance to the predicted 50% LEL mean concentration from the CFD model results. It was demonstrated that this sphere enclosed all of the successful ignition locations in both the indoor and outdoor experiments. Using a smaller sphere defined using the LEL rather than the 50% LEL concentration failed to account for some of the ignition locations.

Simulations of gas dispersion on the GP3 using this 50% LEL sphere approach showed that flammable gas could have reached the flare. It was therefore concluded that the cargo vent gas cloud could have been ignited in the ignition incidents by being directly ignited by the flare. The method used to arrive at this conclusion, involving the 50% LEL sphere concept, is a conservative approach, which may over-estimate the extent of the flammable cloud in certain directions.

These findings should be viewed within the context of the alternative ignition mechanisms reviewed by Pursell *et al*. (2016). An important conclusion from that paper is that the ignition mechanism involving burning droplets falling from the flare through the cargo vent vapour cloud was deemed credible. Experiments conducted by HSL indicated that a burning hydrocarbon droplet would be able to continue burning for long enough to fall the 18 m from the flare to reach the cargo vent. There is also circumstantial evidence that supports this hypothesis; the last three ignition incidents on the GP3 (where the wind conditions are known) all involved a prevailing wind that would have aligned the direction of the flare tilt towards the cargo vent vapour cloud, so that any burning droplets falling from the flare would be likely to pass through the cargo vent vapour cloud.

The final conclusion of the investigation is therefore that the most likely cause of the ignition incidents on the GP3 was burning droplets falling from the flare, but it cannot be ruled out that flammable gas was ignited by being drawn up into the flare. Several other ignition mechanisms were investigated, but none of these were found to be credible.

Acknowledgement and Disclaimer

The work described in this paper was undertaken by the Health and Safety Laboratory under contract to Maersk Oil North Sea UK Limited.

References

Birch A.D., Brown D.R., Dodson M.G. and Thomas J.R., 1978, The turbulent concentration field of a methane jet, Journal of Fluid Mechanics, 88(3), p431-449.

Birch A.D., Brown D.R., Dodson M.G. and Thomas J.R., 1979, Studies of flammability in turbulent flows using laser Raman spectroscopy, 17th Symposium (International) on Combustion, The Combustion Institute, p307-314.

Birch A.D., Brown D.R. and Dodson M.G., 1981, Ignition probabilities in turbulent mixing flows, 18th Symposium (International) on Combustion, The Combustion Institute, p1775-1780.

Birch A.D., Brown D.R., Fairweather M. and Hargrave G.K., 1989, An experimental study of a turbulent natural gas jet in a cross-flow, Combust. Sci. and Tech., 66, p217-232.

Energy Institute, 2005, Area classification code for installations handling flammable fluids. Part 15 of the IP model code of safe practice in the petroleum industry, Energy Institute, London, UK.

Gant S.E., Lea C.J., Pursell M., Fletcher J., Rattigan W. and Thyer A., 2011, Flammability of hydrocarbon and carbon dioxide gas mixtures: measurements and modelling, HSL Report MSU/2010/21. Available from: HSL, Harpur Hill, Buxton, SK17 9JN, UK.

Pursell, M., Gant, S.E., Newton, A., Bennett, D., O'Sullivan, L., Hooker, P. and Piper, D., 2016, Investigation of Cargo Tank Vent Fires on the GP3 FPSO, Part 1: Identification of Ignition Mechanisms and Analysis of Material Ejected from the Flare, IChemE Hazards 26 Conference, Edinburgh, UK, 24-26 May 2016.

Smith M.T.E., Birch A.D., Brown D.R. and Fairweather, M., 1986, Studies of ignition and flame propagation in turbulent jets of natural gas, propane and a gas with a high hydrogen content. 21st Symposium (International) on Combustion, The Combustion Institute, p1403-1408.

Webber D.M., 2002, On defining a safety criterion for flammable clouds, Health and Safety Laboratory Report HSL/2007/30. Available from: http://www.hse.gov.uk/research/hsl_pdf/2007/hsl0730.pdf, accessed 12 July 2015.