# Elevated LNG Dispersion - Effects of topography and phase change

Felicia Tan, Process Safety Engineer, BP, Sunbury-on-Thames
Chris Savvides, Senior Consultant Fire & Blast, BP, Sunbury-on-Thames
Vincent Tam, Professor, University of Warwick, Coventry

The dispersion of vapour of LNG is generally assumed to be from a liquid spill on the ground in hazard and risk analysis. However, it is possible that this cold vapour can be discharged at height through cold venting. While there is similarity to the situation where heavier than air gases, e.g. CO2, is discharged through tall vent stacks, LNG vapour is cold and induces phase change of ambient moisture leading to changes in the thermodynamics as the vapour disperses. A recent unplanned cold venting of LNG vapour event due to failure of a pilot, provided valuable data for further analysis. This event was studied using CFD under steady-state conditions and incorporating the effect of thermodynamics due to phase change of atmospheric moisture. As the vast majority of processing plants do not reside in flat planes, the effect of surrounding topography is also investigated. This case study highlighted that in rare but possible scenarios, the application of a dispersion model could be extended beyond its range of applicability and key assumptions used to derive the models can be violated. The analysis suggested guidance and methodologies appropriate for modelling cold vent and flame out situations for elevated vents.

## Introduction

This paper illustrates the concerns of dispersion of a very cold momentum gas jet from height. The scenario involved the discharge of LNG vapour at speed and at height through a vent stack. One of the purposes of a vent stack is for the safe disposal of routine or emergency release of flammable gases. Some vent stacks have a pilot light to allow the flaring of these gases. In the event of the failure or absence of a pilot light, stack design usually ensures that the emitted gas would not be hazardous to people or facilities downwind, by virtual of its height and location.

### Common LNG Dispersion Scenarios

It is generally assumed in hazard and risk analysis that the dispersion of vapour of LNG from a liquid spill on the ground or on water or sea. This covers scenarios such as spill from low level pipework, leaks from storage tanks and spills onto sea during loading and unloading from LNG tankers. The physics involved is relatively simple to conceptualise and modelled.

The vast majority of consequence analysis for risk analysis purposes use evaporation-and-boiling model in conjunction with integral dense gas dispersion model for these situations. The most common model in used is dense gas dispersion model based on work by Haven (Spicer and Havens, 1987) for spill on land/sea/water.

### A Less Common Scenario

In the less common scenario, cold LNG vapour is vented in an elevated position, as mentioned earlier. This is usually addressed at the design of the vent stack/flare, a common method is to use an integral atmospheric jet dispersion model such as the one based on Ooms (Ooms, 1973).

Integral atmospheric jet dispersion models are simple and can be easily solved using computers at the time when the models were developed. However, these models contain many assumptions some of which are physically unrealistic for LNG vapour dispersion assessment.

### Assumptions

These integral models assume uniform conditions to simplify the physics and mathematics involved. The key assumptions are: (i) Constant wind velocities, (ii) constant and uniform turbulence which implies perfectly flat terrain and no large buildings or equipment close by upstream and downstream and (iii) there is no mass or energy source or sinks which implies there is no phase change. In practice all the above assumptions are violated for the dispersion of a cold LNG vapour.

### Objectives

The objectives of this analysis were to explore the effects of the common assumptions, as mentioned above, on conditions typically found on a LNG installation, such as topography and cold temperature of the LNG vapour. The results were also analysed against real-life data obtained from a recent unplanned cold venting of LNG vapour event due to failure of a pilot.

## Methodology

### Choice of Facilities

Figure 1 is a schematic layout of the LNG facility chosen for this study. The figure shows the location of the vent stack and topographical features. This facility is in the tropics with high ambient humidity throughout the year. Alongside detailed information on topography and plant layout, there was also LNG vapour concentration reading from gas detectors available for one time period from a recent unplanned cold venting event.

## Computational methods

As integral model was not adequate for this study, the computational fluid dynamics code, STARCCM+, was used for this study. The geometric model was constructed using the CAD data of the plant. Outside the confines of the CAD model, other data were used to include the vent stack, terrain, the two storage tanks explicitly.

### Domain

The calculation domain encompasses an area of 5 km$^2$. Care was taken to include sufficient distance upstream for realistic flow establishment at the vent stack, and downstream to beyond the concentration of interest.

### Gridding

Owing to the large area being considered, the number of grid cells was minimized using multi-block grid refinement techniques. Local grid refinements were used to ensure that the plume shape and behaviour was accurately captured (see Figure 2). Even so, 8 million grid cells were needed, comprising of a mixture of prisms, hexahedra and polyhedral.

### Subgrid

Though the plant is over half a kilometre away, the blockages created by plant equipment and supporting structures would have an impact on the plume behaviour. To resolve these items explicitly would render a vast increase in grid cell number and computer runtime. So, their 'blockage' effects on the flow were modelled, i.e. through local drag terms in the momentum equations derived from properties of items from the CAD model for each grid cells.

### Boundary Conditions

Wind inlet: the vertical profiles for a neutral atmospheric stability class was assumed; the appropriate velocity and turbulence (kinetic energy and dissipation rate) used. Domain outlet: flow split outlet conditions was assumed.

## Effect of water moisture

As the cold LNG vapour was well below the freezing point of water, moisture in the air would freeze, increasing the effective density of the plume. As well as freezing, condensation, melting and evaporation of ambient air moisture were modelled. The various phases of water were assumed to be in thermodynamic equilibrium with its surrounding and that slip velocity was zero (hence can be modelled as a fluid). For this study, relative humidity was assumed to be 90%, corresponding to a facility located by the sea.

## Topographical Effect

The bottom boundary of the calculation domain follows the contour of the terrain. The following conditions are applied: (a) no slip, (b) rough wall boundary condition with an appropriate roughness value to account for the terrain vegetation distribution, and (c) adiabatic thermal boundary.

## Steady state

As with integral models, this study focussed on steady state solution. The total calculated flammable volume was used as an indicator for steady state (or computational convergence). Results were taken only when this has stabilized for a continuous and sufficient number of timesteps (> 1000).

## Venting

Four venting rates were simulated: 16 kg/s, 25 kg/s, 30 kg/s and 45 kg/s. The LNG vapour was assumed to be have the properties of methane at a molecular weight of 17.2 and at a temperature of -161ºC, discharged at the top of the stack at a height of 35 m above local ground level.

## Wind Directions

The normal practice would be to align the wind direction towards the area of interest, in this case, the nearest point in the process plant. It was the only occupied and hazardous area with potential for ignition. Four wind directions about this direct alignment direction were also investigated and these are shown in Figure 3.

# Results

The results showed that the vent stack was sufficient to disperse the LNG vapour sufficiently that it does not pose a flammable hazard on the plant, even at a high vent rate of 45 kg/s. However, the focus of this study was not so much that the design was adequate but to explore factors which could have significant impact on the dispersion behaviour.

Figure 4 shows the plume shape. The plume rose, levelled off and after a distance, descended onto the ground along which the plume continued to disperse. For the high release rate case of 30 kg/s, the effect of the topography can be seen with the plume splitting up into two, close to the vent exit and then descending onto the ground at two locations.

A summary of results is given in Table 1, which shows the effect of release rates, wind directions and wind speed. There were results which showed trends which were consistent with the authors' current outstanding of dispersion; the increase in target concentration as release rate increased (Cases 1, 1a and 1b). There were also contrary results too; concentration at target area increased as wind direction deviated away from that directly aligned with the vent and the target area.

## Discussions

The CFD analysis demonstrated that results that countered results typically shown with using simple integral model i.e. as the release rate from the vent decreased, the concentration at the target sensor increased. This was the conditions that encouraged the gas plume to touch down early in low wind and low release rate conditions. At higher wind speed, the plume dispersed sufficiently that it did not descend to ground. No ground level concentrations were detected.

### Comparison with measurements

Readings from the log of gas detectors indicated that the LNG plume had touched down. This aspect was in agreement with results here. The reading further indicates that the gas concentration was between 2% LFL and 15% LFL. The calculated figures fell within this range (Table 1).

### CFD vs integral model

Prior to carrying out this CFD study, analysis using integral dispersion models were used. This included a jet dispersion model for an elevated source and a dense gas dispersion model for a low momentum source. The initial calculations using integral jet dispersion model showed that the LNG plume continued to rise after its release and remained aloft throughout. The calculated ground level concentration consequently was very low. When the heavy LNG vapour would descend to the ground and then dispersed, the calculated concentration level at the distance of the target area was about 30% to 40% LFL for a vent rate of between 20 and 40 kg/s (compared to 2 % and 15% LFL gas detector readings).

These calculations showed that commonly used integral jet dispersion model underestimated the flammable hazards as it did not predict the descend of the plume. The dense gas model, by not accounting for the initial momentum mixing, over-estimated the flammable hazards. It is therefore, in situations like those described in this paper, advisable to use CFD analysis.

### Moisture effect

Moisture in the air can affect the plume dispersion at various stages of the plume. The condensation and freezing led to formation of ice particles and release of latent heat. As the plume entrained warm air, ice melted and then evaporated, absorbing heat and cooling the plume in the process. The mean plume temperature along its trajectory deviated from that when there is no moisture. The evolution of different phases of moisture in the plume is shown in Figure 6.

It is common practice in dispersion calculations to ignore the effect of moisture in air (Hansen et al., 2010), (Fardisi and Karim, 2011). When this effect is recognised, a modifier of ambient air properties is used (Cormier et al., 2009).

In a recent CFD study on Burro and Coyote LNG spill tests carried out in deserts in the USA, the effects of moisture in air on dispersion behaviour was found to be significant (Zhang et al., 2015). At 5% relative humidity (RH), the difference in calculated concentration at a location for including or excluding moisture effect was relatively small (~ 10%). However, the difference quickly rose to about 30% at an RH of 22%.

### Good match with visible plume (not a reliable measure of flammable plume shape)

It was tempting to use the visible plume to inform oneself of the flammable hazard distances or predict position of the plume when it descended to the ground, or whether it would descend to the ground. As can be seen in Figure 7, the calculated plume shape matched well with the observed visible plume, however, it was difficult to deduce the complete plume trajectory based on visible plume information.

Furthermore, the visible plume was not a good indicator of flammable extent as it would be dependent on RH (Vílchez et al., 2013). For the plume section that was aloft, it was the RH local to the plume which may vary with height and locations; it was highly unlikely to be the same as that measured on the ground.

### Ground effect

Topographic effect was very evident from the results. It altered the wind velocity field and its distribution in the entire calculation domain. This resulted in the splitting up of the plume leading to two touch-down points and a meandering ground level plumes, the trajectories of which were determined by the ground contours and the wind field (see Figure 4). This was consistent with the long standing guidance for environmental assessment (Steven, R.; Hanna Gary A.; Briggs Rayford P.; Hosker, 1983).

### Effect of storage tanks

Large objects, such as storage tanks, had similar effect as topography, but the effects were local. These large objects could induce downdraft, dragging the plume downward, promoted earlier touchdown or increased ground level concentration. This effect can be seen in Case 4 and 5 of Figure 8.

### Plume touchdown location

The location of plume touchdown would affect the location and size of the hazard zone. A higher ambient wind increased the distance of the touchdown zone, giving a higher gas concentration at ground level than locations closer to the vent stack (see Table 1).

### The next step

As environmental conditions and vent rates changed with time, the next step is to assess these effects. This is the subject of a separate paper which is in preparation.

### Other scenarios

This study addressed scenario which is not routinely assessed. There are other scenarios where LNG vapour can be generated at height and they are beginning to be studied. This included the EU funded project, SafeLNG, that considers the vapour generation and its dispersion following a release of LNG at height which might have occurred after the rupture of an LNG import or export pipework at the top of an LNG tank (Wen, 2018).

## Conclusions

This CFD analysis demonstrated the effects of moisture in the air and topography when modelling LNG vapour dispersion from height. The cold vapour could induce phase change of ambient moisture, leading to changes in the thermodynamics as the vapour dispersed. This affected the dispersion and trajectory of the plume i.e. aloft time, distance, touch down location and local ground concentration. Topography altered the wind velocity field and its distribution in the entire calculation domain. The importance of large equipment on ground level should also be considered as shown in this analysis. It was also demonstrated that the storage tank downstream of the vent produced downwash effects, therefore dragging the plume aloft down towards the ground, promoting earlier touchdown or increasing ground level concentration.

The analysis also showed the effects of different release rates, wind directions and wind speeds. There were results which showed trends which were consistent with the authors' current understanding of dispersion i.e. increase in target concentration as release rate increased, but also contrary results i.e. concentrations at target area increased as wind direction deviated away from that directly aligned with the vent and target areas. When compared with real-life cold venting situation, the results from this analysis showed broadly good agreement with key observations.

This analysis highlighted the possible scenarios where the application of an integral dispersion model could be extended beyond its range of application and key assumptions used to derive the models can be violated.

## Acknowledgement

Authors would like to thank Christophe Mabilat (then Atkins UK) for the generation of results and figures in this paper.

## References

Cormier, B.R., Qi, R., Yun, G., Zhang, Y., Sam Mannan, M., 2009. Application of computational fluid dynamics for LNG vapor dispersion modeling: A study of key parameters. J. Loss Prev. Process Ind. 22, 332–352. https://doi.org/https://doi.org/10.1016/j.jlp.2008.12.004

Fardisi, S., Karim, G.A., 2011. Analysis of the dispersion of a fixed mass of LNG boil off vapour from open to the atmosphere vertical containers. Fuel 90, 54–63. https://doi.org/https://doi.org/10.1016/j.fuel.2010.09.008

Hansen, O.R., Gavelli, F., Ichard, M., Davis, S.G., 2010. Validation of FLACS against experimental data sets from the model evaluation database for LNG vapor dispersion. J. Loss Prev. Process Ind. 23, 857–877. https://doi.org/https://doi.org/10.1016/j.jlp.2010.08.005

Ooms, G., 1973. A new method for the calculation of the plume path of gases emitted by a stack. VDI Berichte. https://doi.org/10.1016/0004-6981(72)90098-4

Spicer, T.O., Havens, J.A., 1987. Field test validation of the degadis model. J. Hazard. Mater. 16, 231–245. https://doi.org/https://doi.org/10.1016/0304-3894(87)80036-5

Steven, R.; Hanna Gary A.; Briggs Rayford P.; Hosker, J., 1983. Handbook on atmospheric diffusion; prepared for the U.S Department of Energy, Atmospheric Environment (1967). https://doi.org/10.1016/0004-6981(83)90164-6

Vílchez, J.A., Villafañe, D., Casal, J., 2013. A dispersion safety factor for LNG vapor clouds. J. Hazard. Mater. 246–247, 181–188. https://doi.org/https://doi.org/10.1016/j.jhazmat.2012.11.045

Wen, J.X., 2018. Advances in consequence modelling for LNG safety: outcomes of the SafeLNG Project. FABIG Newsl. 25–32.

Zhang, X., Li, J., Zhu, J., Qiu, L., 2015. Computational fluid dynamics study on liquefied natural gas dispersion with phase change of water. Int. J. Heat Mass Transf. 91, 347–354. https://doi.org/https://doi.org/10.1016/j.ijheatmasstransfer.2015.07.117

## Figures

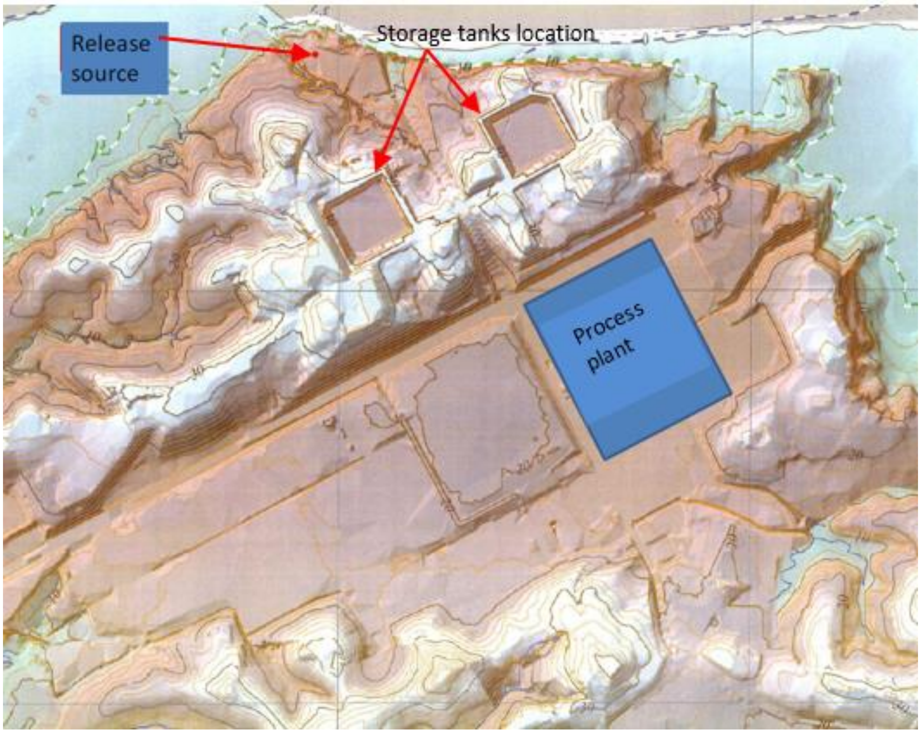

Figure 1 Schematic diagram showing the layout of key items.

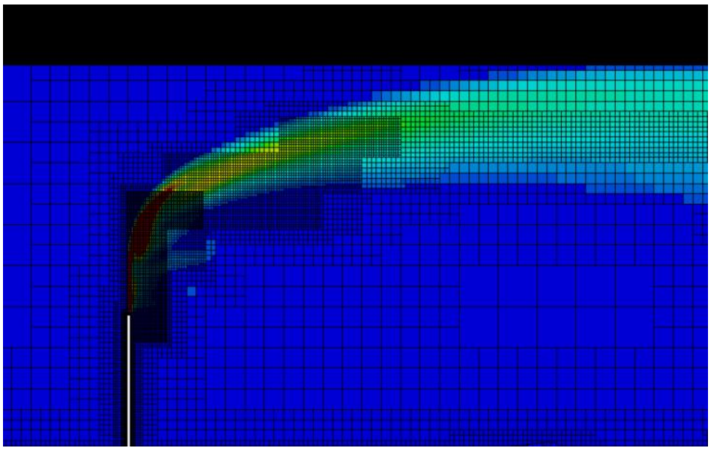

Figure 2 An example of grid layout close to the vent stack.

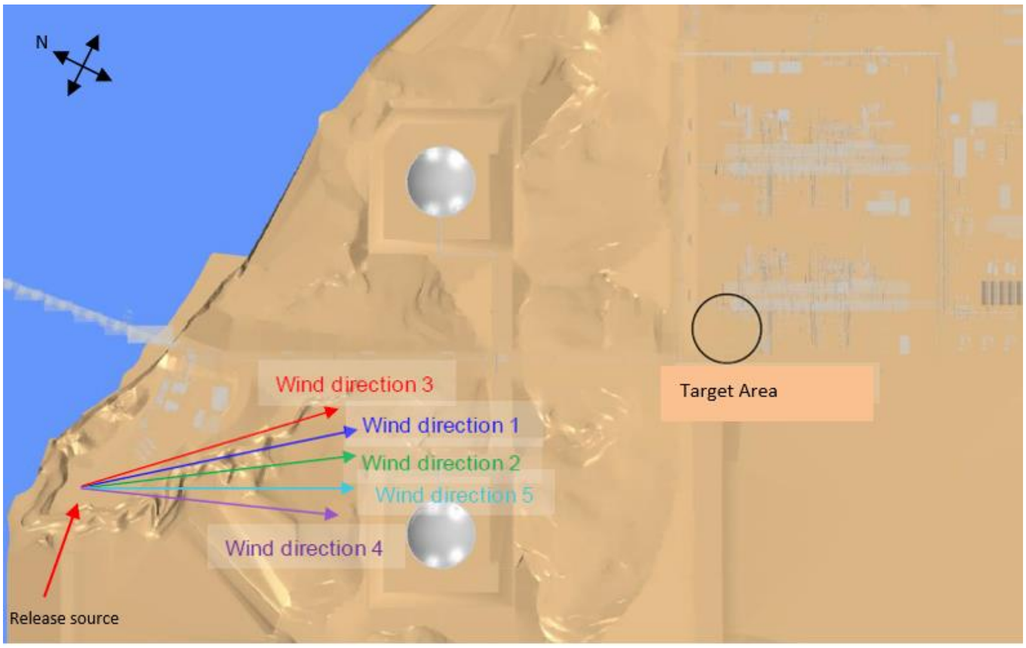

Figure 3 Schematic diagram showing the four wind directions. A target area is also shown; the gas concentration calculated for this area will be used for comparison.

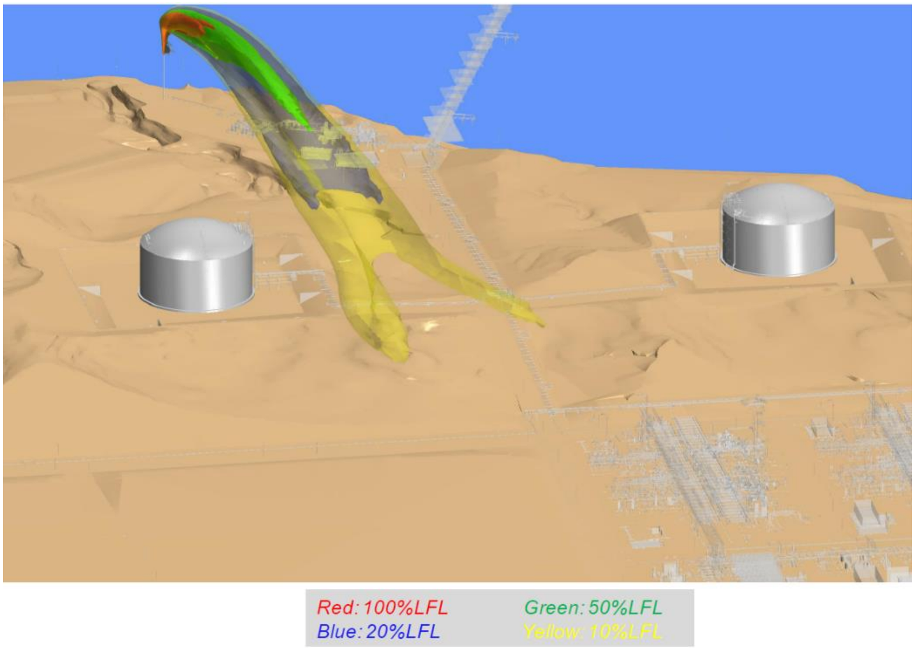

Figure 4 A 3D picture of the calculated plume envelops for 4 gas concentrations (100% LFL, 50% LFL, 20% LFL and 10% LFL).

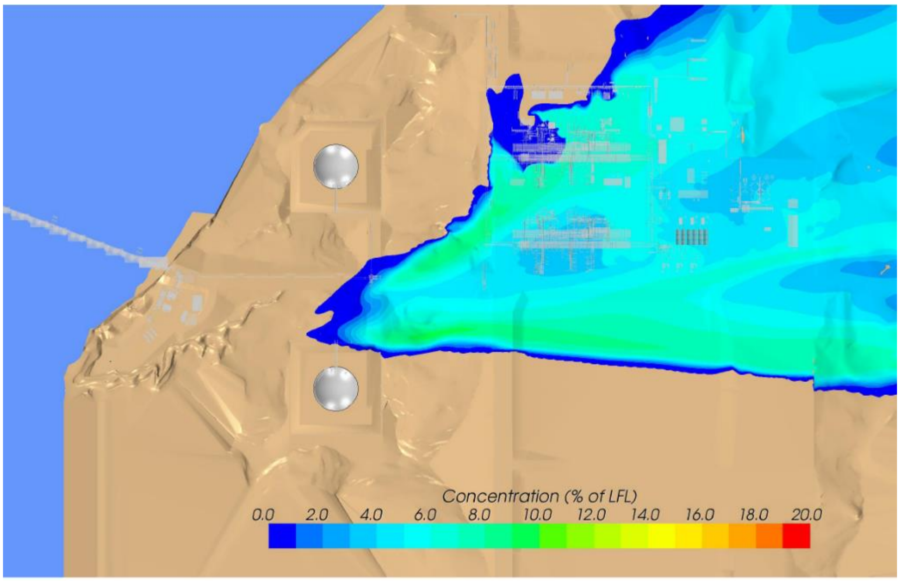

Figure 5 A corresponding ground level concentration for a 30 kg/s release showing the two touch down locations and two distinct ground level plumes.

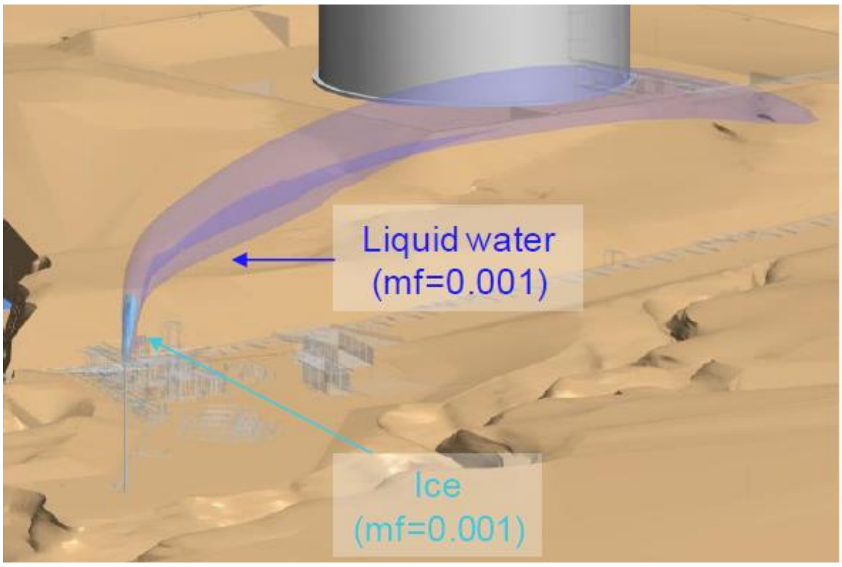

Figure 6 Evolution of moisture at various point in the plume.

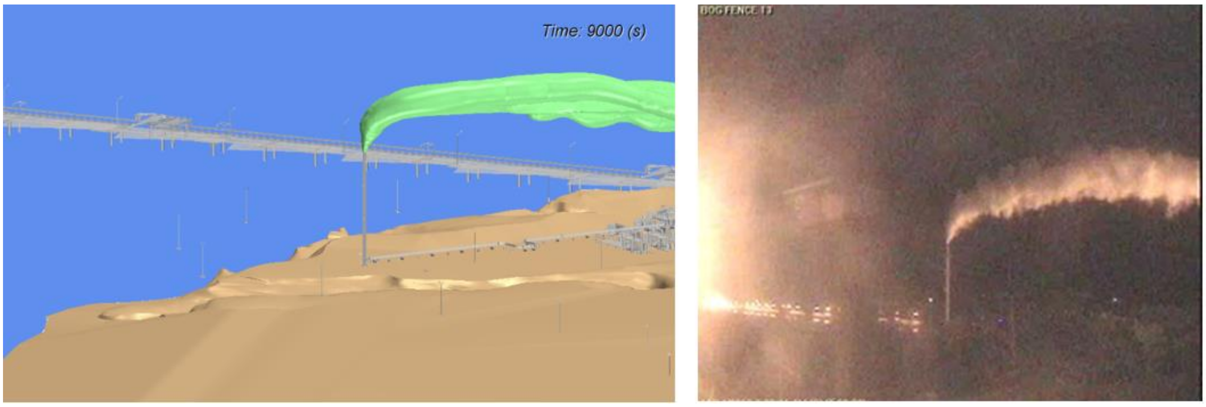

Figure 7 Calculated plume compared with a picture of visible plume.

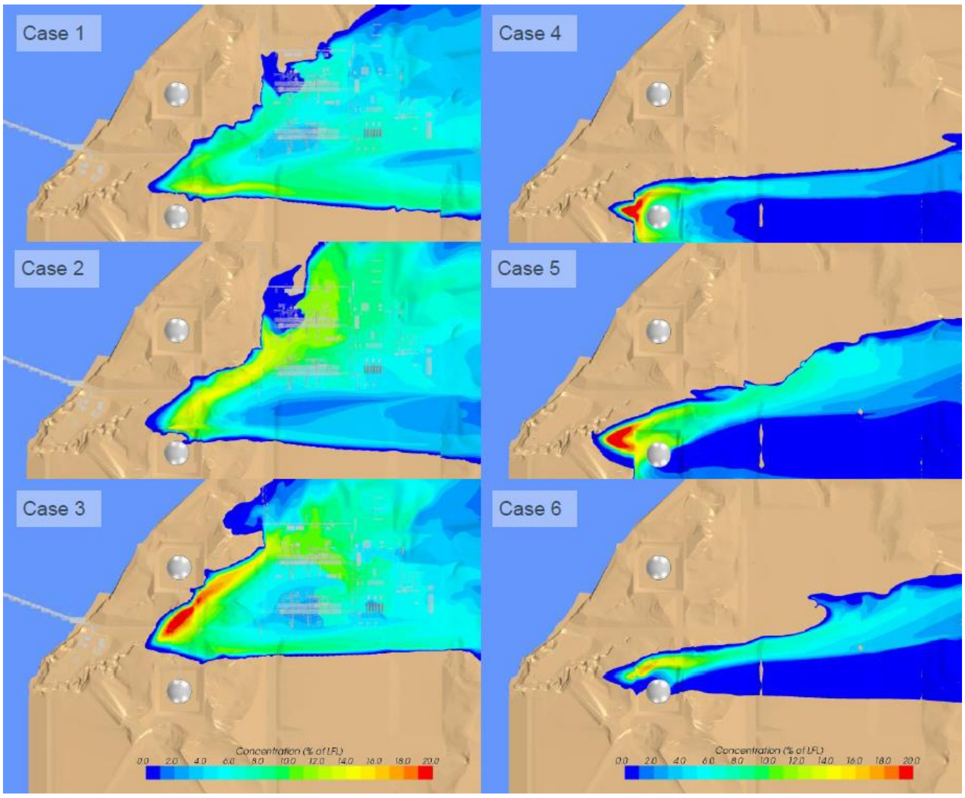

Figure 8 Ground level concentration for various wind directions and release rates.

## Tables

| Case no. | Vent rates (kg/s) | Wind Directions | Wind Speed (m/s) | Concentration (% LFL) |
|---|---|---|---|---|
| 1 | 25 | 1 | 2 | 3.7 |
| 1a | 30 | 1 | 2 | 4 |
| 1b | 45 | 1 | 2 | 12.5 |
| 2 | 25 | 2 | 2 | 5.5 |
| 3 | 25 | 3 | 2 | 3.8 |
| 4 | 16 | 4 | 3.9 | 0 |
| 5 | 16 | 5 | 3 | 3.7 |
| 6 | 16 | 5 | 3.9 | 0 |

Table 1 Summary table of results showing the ground level concentration at the target area in Figure 3**.**