Evaluating the potential for overpressures from the ignition of an LNG vapor cloud during offloading

Filippo Gavelli, Scott G. Davis and Olav R. Hansen
GexCon US, 7735 Old Georgetown Rd suite 1010, Bethesda, MD 20814
***Email: fgavelli@gexcon.com**

Abstract

Natural gas (composed primarily of methane) is generally not considered to pose overpressure hazards when ignited in unconfined and low- or medium-congested areas, as most of the areas within LNG regasification facilities can typically be classified. However, as the degrees of confinement and/or congestion increase, the potential exists for the ignition of a methane cloud to result in damaging overpressures (as demonstrated by the recurring residential explosions due to natural gas leaks). Therefore, it is prudent to examine a proposed facility's design to identify areas where vapor cloud explosions (VCEs) may cause damage, particularly if the damage may extend off site. An area of potential interest for VCEs is the dock, while an LNG carrier is being offloaded: the vessel hull provides one degree of confinement and the shoreline may provide another; some degree of congestion is provided by the dock and associated equipment.

In this paper, the computational fluid dynamics (CFD) software FLACS is used to evaluate the consequences of the ignition of a flammable vapor cloud from an LNG spill during the LNG carrier offloading process. The simulations will demonstrate different approaches that can be taken to evaluate a vapor cloud explosion scenario in a partially confined and partially congested geometry.

Introduction

Safety studies for LNG facilities (both liquefaction and regasification) are typically focused on the analysis of accidents that may occur within the onshore portion of the plant. This includes, for example, the dispersion of and potential overpressures from the ignition of a flammable vapor cloud in a liquefaction plant, or the pool fire from a large LNG spill in a regasification plant. Accidents on the marine side of an LNG terminal are rarely considered in detail.

In the United States, for example, the marine side of a terminal is evaluated as part of the Waterway Suitability Assessment; however, the WSA procedure does not require the analysis of specific accident scenarios – rather, it provides generally applicable guidelines to set the safety distance around a moored LNG vessel. To date, only one proposed onshore LNG receiving terminal in the US has been asked by regulators to perform a specific analysis to evaluate the consequences of a vapor cloud explosion at the unloading pier [1].

Even when required, these studies are typically performed using simple models, such as the TNO-Multi Energy Method (MEM) or the Baker-Strehlow-Tang (BST) method. However, these models perform best when used to estimate far-field overpressures, rather than near-field as in the case of a moored vessel. Additionally, the MEM and BST models are not particularly sensitive to the degree of congestion in the region occupied by the flammable cloud and cannot properly account for obstacles or asymmetric geometries.

The purpose of this paper is to demonstrate the application of an advanced modeling tool – FLACS – to evaluate the consequences of an LNG vapor cloud explosion at the unloading pier. The use of a model capable of calculating LNG pool spreading, vapor dispersion and flame growth and acceleration, while taking into account the detailed geometrical features of the scenario allows for an accurate prediction of the consequences of an accident. This, in turn, provides the opportunity not only to verify and demonstrate compliance with safety regulations, but also to identify potential problem areas and evaluate the effect of mitigation measures.

FLACS

FLACS is a specialized CFD tool developed to address process safety applications such as:

- Dispersion of flammable or toxic gases;
- Gas and dust explosions;
- Propagation of blast and shock waves.

For the purpose of simulating the dispersion of vapor clouds from liquefied natural gas (LNG) releases, FLACS incorporates a fully two-dimensional shallow water-based model for the simulation of LNG pool spreading and vaporization, and a three-dimensional model for the simulation of LNG vapor cloud dispersion. As such, FLACS provides a unified environment in which the entire LNG spill and dispersion scenario can be simulated efficiently and accurately.

The pool model in FLACS allows for the formation and spreading of the pool, accounting for the presence of obstacles and sloped terrain. The LNG evaporation rate is calculated locally (grid cell by grid cell) and is the sum of contributions from heat transfer from the substrate (ground or water), solar radiation and convective heat transfer from the air [2]. The rate of vapor generation is also affected by physical variables such as local wind speeds and turbulence levels, as well as the local vapor pressure above the pool, all of which can be calculated at every time step thanks to the simultaneous solution of both the liquid pool spread and the vapor cloud dispersion.

Vapor cloud dispersion is calculated by solving the Reynolds Averaged Navier-Stokes equations for a compressible fluid using a finite volume method [3, 4], coupled with the k-ε model with the standard set of constants (Launder & Spalding [5]) for the turbulent closure. Buoyancy effects are taken into account in the turbulent equations. The atmospheric boundary layer is modeled by forcing profiles for the velocity, temperature and the turbulence parameters on inlet boundaries. Wind inlet profiles are imposed according to the Monin-Obukhov length L and the atmospheric roughness length z_0 for the given atmospheric stability class.

The FLACS pool and dispersion models have been used to simulate the full set of experimental tests (field trials and wind-tunnel tests, obstructed and unobstructed) included in the Model Validation Database of the Fire Protection Research Foundation’s Model Evaluation Protocol (MEP) [6]. The MEP is used to assess the suitability of dispersion models for predicting hazard ranges associated with large spills of LNG. The FLACS model predictions compared well with the experimental data and met the statistical performance requirements specified in the MEP [7].

The same compressible flow solver used for vapor dispersion is also used for vapor cloud explosion simulations – vapor cloud explosion modeling was in fact the original application for which FLACS was developed in the 1980s. The FLACS compressible solver includes a model for the generation of turbulence behind sub-grid objects and a model for flame folding around them [8]. A model for development of the flame that describes how the local reactivity changes with parameters like concentration, temperature, pressure, turbulence, etc. is implemented to model combustion and explosion dynamics. A good description of geometry and the coupling of geometry to the flow, turbulence, and flame is one of the key elements in the modeling. The real flame area is then properly described and corrected for curvature at smaller scales than the grid, using sub-grid models

Case Study: LNG Carrier Offloading at Near-Shore Pier

Perhaps the most common layout for LNG carrier offloading at an onshore receiving terminal is that of a pier very near the shoreline. Figure 1 shows a hypothetical 140,000 m3 LNG carrier moored at a near-shore offloading pier. The pier deck measures approximately 31 m by 19 m (102 ft by 62 ft) and is 11 m (36 ft) above the water surface; the deck is supported by a square array of 0.9 m (3 ft) diameter pylons spaced 6 m (20 ft) apart.

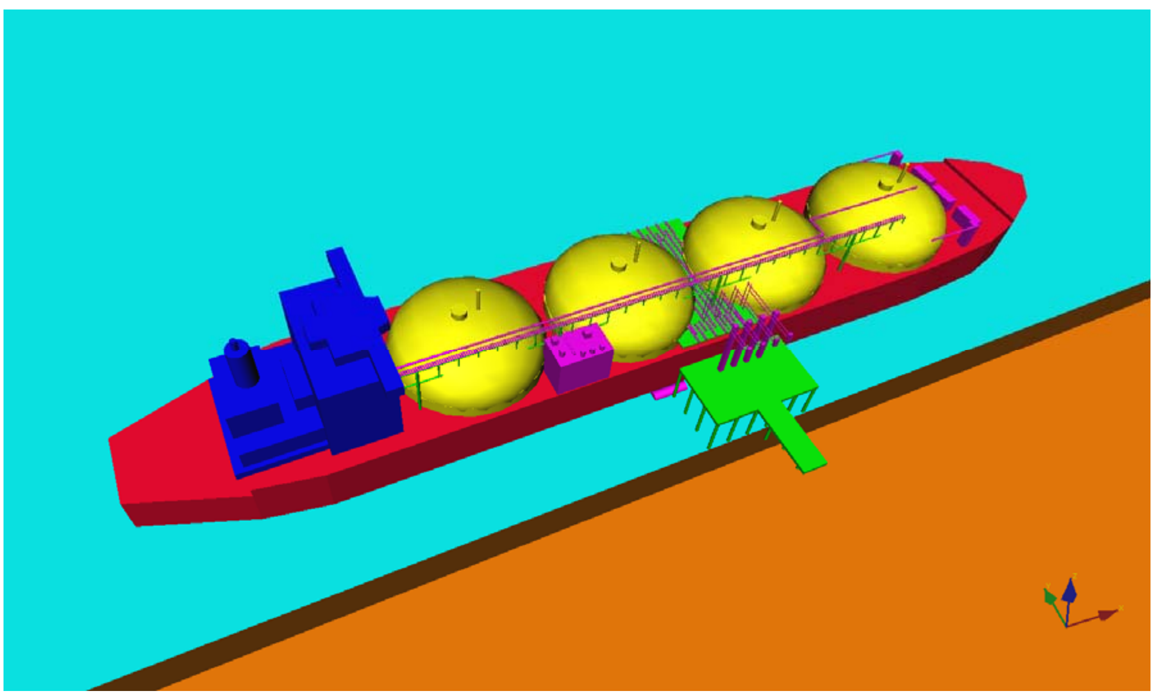

Figure 1. Geometry model of LNG carrier at near-shore pier.

The scenario presents some degree of confinement – due to the LNG carrier hull, the pier deck and the sloped shoreline – and of congestion, primarily due to the pylons supporting the pier. The vapor cloud explosion calculations will be performed in two steps:

1. Worst-case scenario: the volume between LNG carrier and shore (including the pier) is assumed to be filled with a near-stoichiometric gas cloud; the cloud is assumed to be ignited in a confined corner;
2. Realistic scenario: an LNG spill is assumed to occur, leading to the formation of a pool on the water surface. The LNG vapor cloud resulting from the pool vaporization is allowed to disperse before being ignited.

Consequence analyses for LNG releases typically assume the vapor cloud to be composed of 100% methane; this is a reasonable assumption for large scale releases, as methane is by far the main component of LNG and also the most volatile. However, heavier hydrocarbons such as ethane, propane etc., will also evaporate from an LNG pool on water, albeit at later times. Given the higher reactivity of heavier hydrocarbons, assuming an LNG vapor cloud to be composed of 100% methane may lead to non-conservative results. Therefore, the examples described in this paper are performed assuming the LNG vapor cloud to be given by a mixture of hydrocarbons as summarized in Table 1.

Table 1. LNG vapor cloud composition for the case studies.

Component	**Percentage (by volume)**
Methane	87.93
Ethane	7.73
Propane	2.51
Butane	1.22
Pentane	0.61

Worst-case scenario

A near-stoichiometric gas cloud (equivalence ratio = 1.05) is assumed to fill the region between the LNG carrier and the shore, including the offloading pier, as by the green box shown in Figure 2. The flammable cloud is then assumed to be ignited near the vessel hull, just above the water level, in proximity of the aft breasting dolphin (see Figure 2).

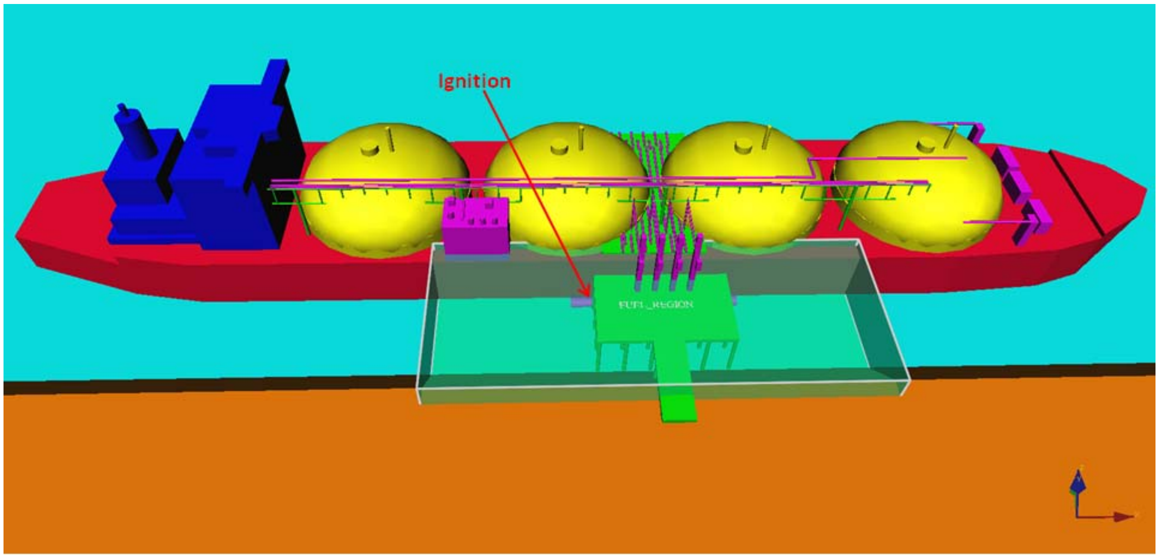

Figure 2. Flammable cloud (semi-transparent green volume) for worst-case scenario VCE for a moored LNG carrier.

The sequence of images in Figure 3 shows the evolution of the flame front following ignition of the flammable cloud. The acceleration of the flame front as it interacts with the pier pylons can be observed in the images and results in preferential propagation of the flame front underneath the pier.

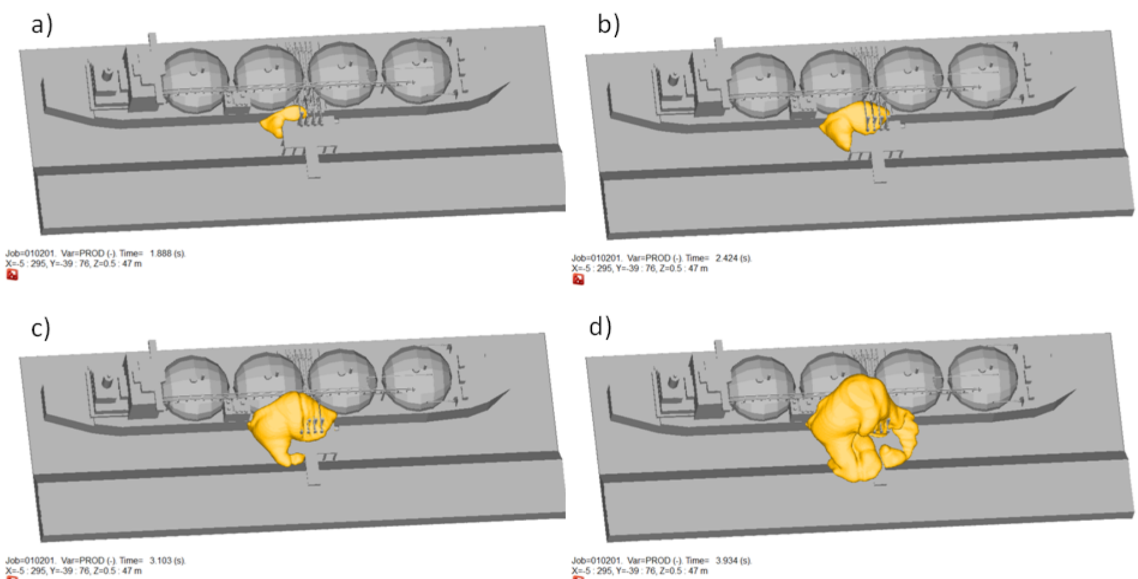

Figure 3. Evolution of the flame front for worst-case scenario VCE. Snapshots after: a) 1.88 s; b) 2.42 s; c) 3.10 s and d) 3.93 s from ignition.

Due to the lower reactivity of methane (which comprises approximately 88% of the LNG vapor cloud mixture in this example), the flame acceleration due to the pylons is insufficient to cause significant overpressures. The maximum overpressure calculated for this scenario is approximately 20 mbar (0.3 psig) and occurs just outside of the pier deck, on the side opposite the ignition location, as shown in Figure 4.

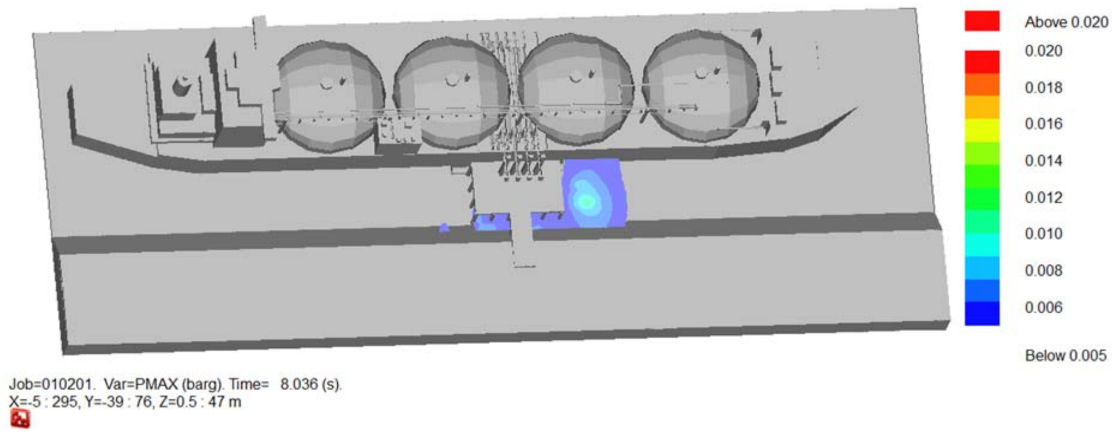

Figure 4. Maximum overpressure on a horizontal plane 4 m above water level, for the worst-case scenario VCE. Overpressure color contours are in barg.

Therefore, in this case, the VCE overpressure does not represent a concern for the public or for the facility's structures. The consequences of a worst-case vapor cloud explosion, however, are highly dependent upon the geometric details of the region occupied by the vapor cloud – namely, the degree of confinement and congestion.

In fact, the same VCE scenario results in maximum overpressures of approximately 180 mbar (2.5 psig) – or almost one order of magnitude higher than in the "reference" case – if the pylon spacing is changed from 6 m (20 ft) to 3 m (10 ft), as shown in Figure 5.

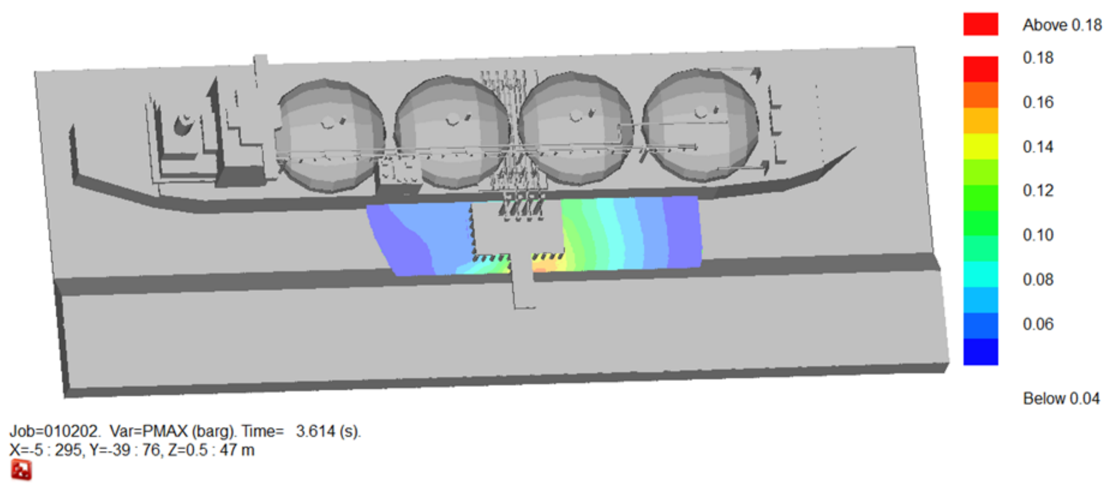

Figure 5. Maximum overpressure on a horizontal plane 5 m above water level, for the worst-case scenario VCE with reduced pylon spacing. Overpressure color contours are in barg.

As expected, the higher congestion due to the increased number of pylons causes stronger flame acceleration below the pier deck (Figure 6), as demonstrated by the faster fuel burning rate than in the previous case (see Figure 7).

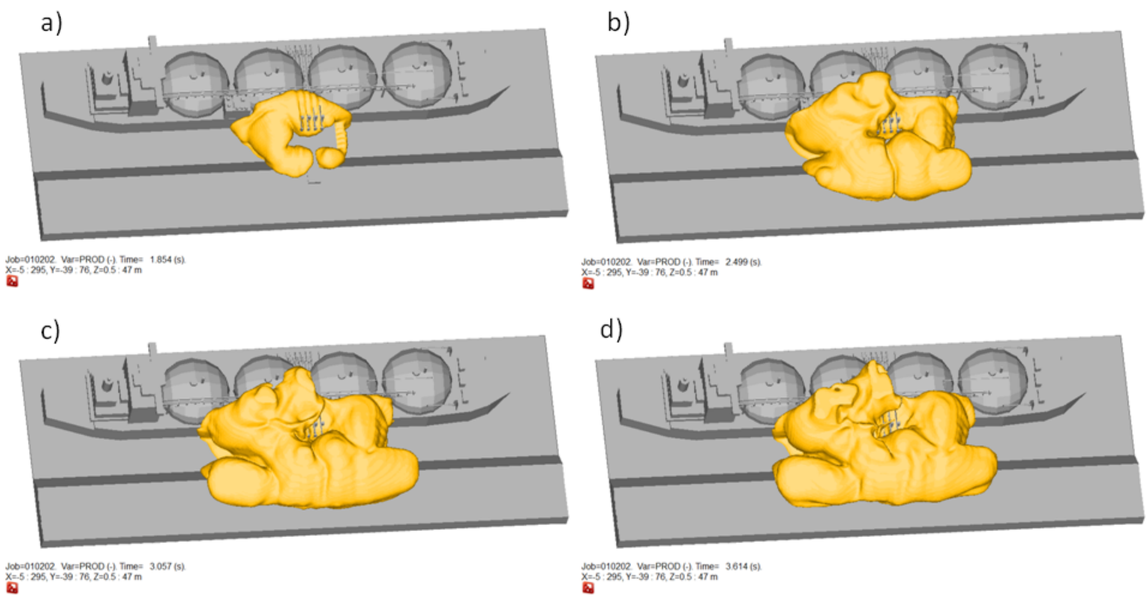

Figure 6. Evolution of the flame front for worst-case scenario VCE with reduced pylon spacing. Snapshots after: a) 1.85 s; b) 2.50 s; c) 3.06 s and d) 3.61 s from ignition.

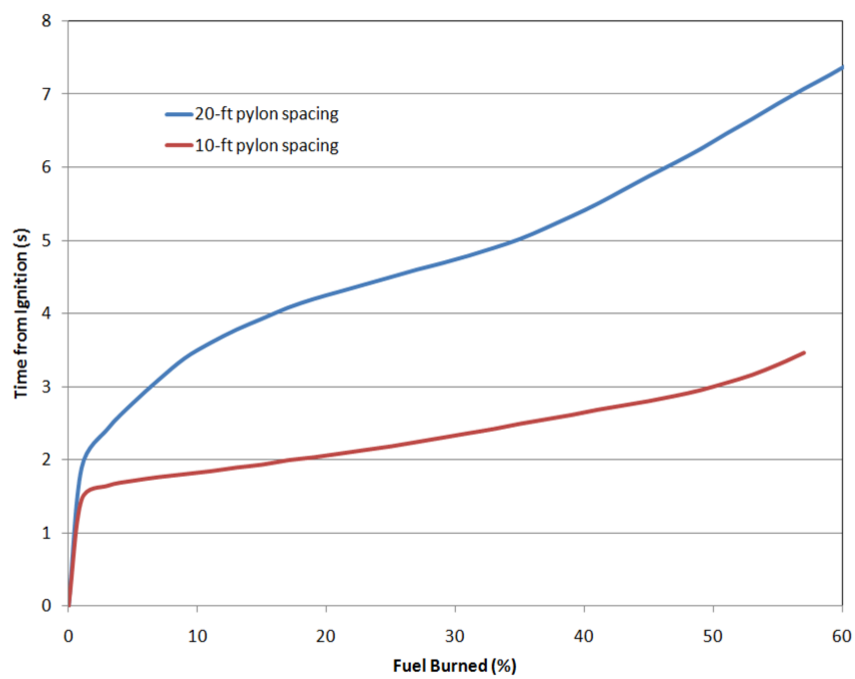

Figure 7. Fuel burning rate following the ignition of the gas cloud with nominal (20 ft) and reduced (10 ft) pylon spacing.

Full accident sequence

The worst-case approach described above, in some cases, may be sufficient to determine the safety of the LNG unloading pier. In general, however, the worst-case approach examines non-credible scenarios and, therefore, may result in consequences unnecessarily difficult and expensive to mitigate. In order to evaluate more "realistic" scenarios, it is important to take into account the entire sequence of events, namely:

1. The LNG spill;
2. The formation, spreading and vaporization of an LNG pool;
3. The dispersion of the LNG vapor cloud by the wind;
4. The ignition of the flammable vapor cloud.

An example of a realistic scenario analysis is provided below. It is assumed that a leak occurs at one of the unloading arms and results in a steady LNG spill rate of 612.5 kg/s for 10 seconds. The LNG spill is assumed to be 100% liquid (i.e., no flashing), therefore, a pool is formed on the water surface. Wind is blowing from sea to shore (across the LNG carrier) at a speed of 2 m/s (at 10 m elevation) and with Pasquill stability class F. The LNG vapor cloud formed by the pool vaporization is allowed to disperse for 3 minutes, after which ignition occurs in proximity of the pier deck. The "realistic" scenario is simulated entirely within FLACS, which minimizes the assumptions and simplifications that need to be made when different models need to be combined to perform an analysis (for example, when an external model is used to calculate the spreading and vaporization of the LNG pool). It should be mentioned that the selected scenario is only one out of many potential incident scenarios, and that the associated hazard for a given scenario depends to a great extent on the size of the vapor cloud formed. The most significant hazard will occur if a large, homogenous cloud at near stoichiometric concentration can be formed; the size of the flammable cloud is not directly proportional to the LNG spill rate because it is also strongly affected by other parameters, such as wind speed and direction. Therefore, identifying the most hazardous scenarios a priori can be quite difficult, especially in complex geometries; instead, the sensible approach is to perform several vapor cloud dispersion simulations with different spill rates and locations, wind speeds and directions. This process can either be performed iteratively, starting with a defined base scenario and evaluating the results from each simulation before selecting the conditions for the next one (this approach may require approximately 10-20 simulations to complete), or by simulating a larger number of parameter combinations and then evaluating the results (this approach, which is typically used for probabilistic quantitative risk assessments, may require approximately 100-200 simulations to complete). Due to the large number of parameter combinations, the most hazardous consequences are more likely to be identified by performing the latter method.

The spreading of the LNG pool on the water is shown in Figure 8. As can be seen in the time sequence, the pool spreading is far from symmetric (circular or semi-circular) as commonly assumed. Instead, the spreading pool interacts with the array of pylons creating a complex LNG pattern; additionally, the obstacles represented by the carrier hull and by the breasting dolphins cause sloshing and reflection of the pool front.

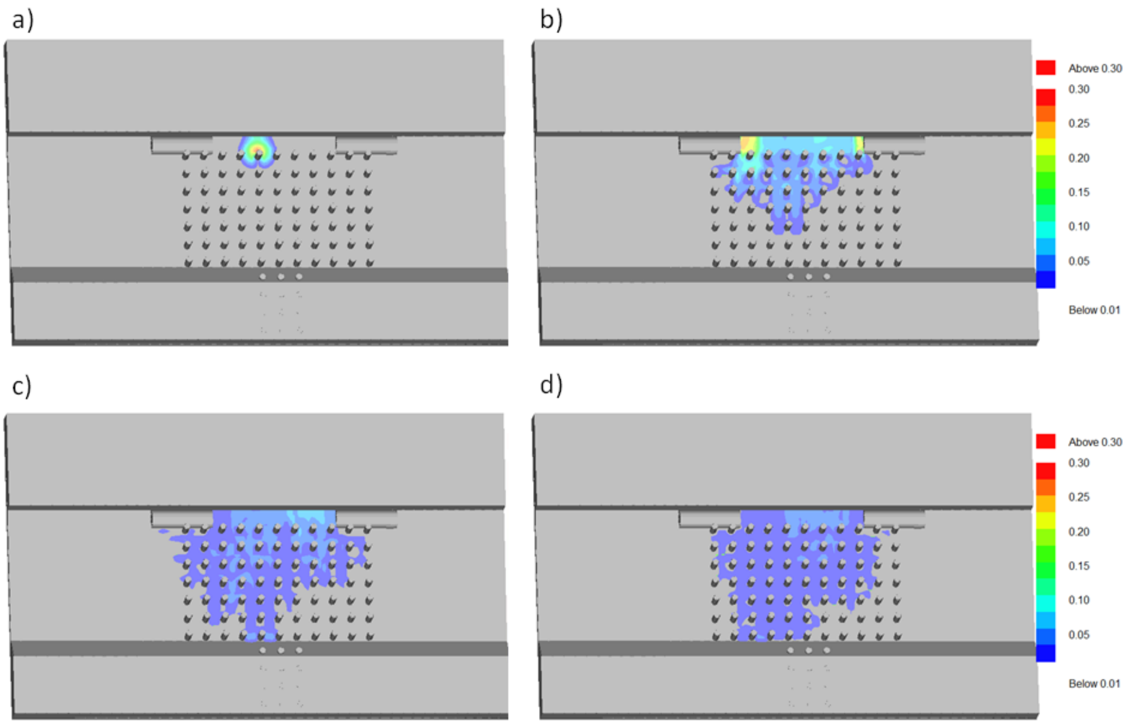

Figure 8. Spreading of the LNG pool (color coded according to pool depth, in meters) from an unloading arm leak, in the case of reduced pylon spacing. Snapshots after: a) 2 s; b) 14 s; c) 26 s and d) 38 s from the beginning of the spill.

The dynamics of the pool spread determine where LNG vapors are being formed. In this case, vapors formed below the pier deck are more likely to be shielded from the wind, but are also in a region of higher turbulence. The complex interaction of these opposing contributions ultimately determines the characteristics of the flammable cloud at the time of ignition and, therefore, are critical to the outcome of the incident.

The growth of the flammable vapor cloud in the "realistic" scenario is shown in Figure 9. At the time of ignition (180 s, the bottom right image in the figure) the cloud is quite different from the uniformly mixed "worst-case" cloud (see Figure 2). In particular, the "realistic" cloud is highly stratified so that the volume of gas that would contribute to an explosion is much smaller than for a well-mixed cloud of the same size. As a result, the overpressures from the ignition of the "realistic" cloud are expected to be lower than for the "worst-case" scenario.

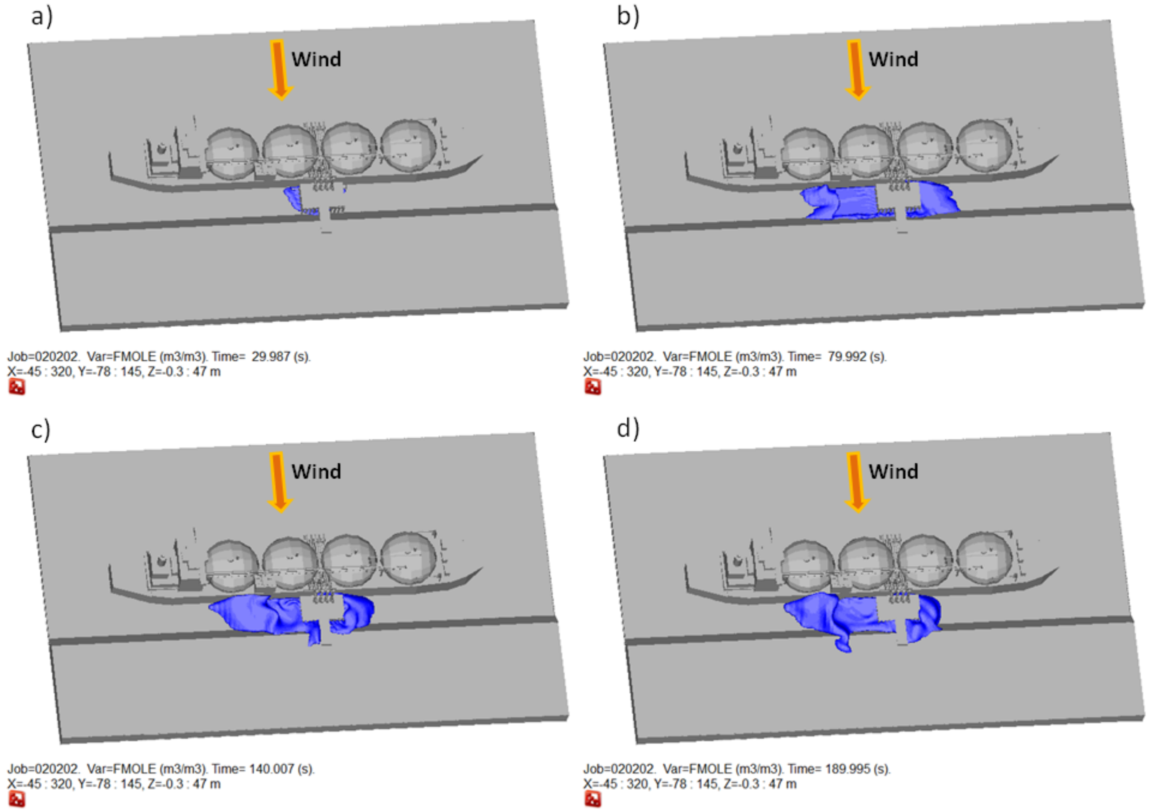

Figure 9. Growth of the flammable LNG vapor cloud (isosurface shows the LFL concentration) from an unloading arm leak. Snapshots after: a) 20 s; b) 70 s; c) 130 s and d) 180 s from the beginning of the spill.

The evolution of the flame front following the ignition of the "realistic" flammable cloud is shown in Figure 10. Once the flame enters the congested area, the flame front is accelerated but it burns at a slower rate than in the previous cases due to the smaller flammable volume at near stoichiometric concentrations available to sustain the flame propagation. As a result, the maximum overpressure is only approximately 22 mbar (0.3 psig) with the denser pylon spacing, as shown in Figure 11.

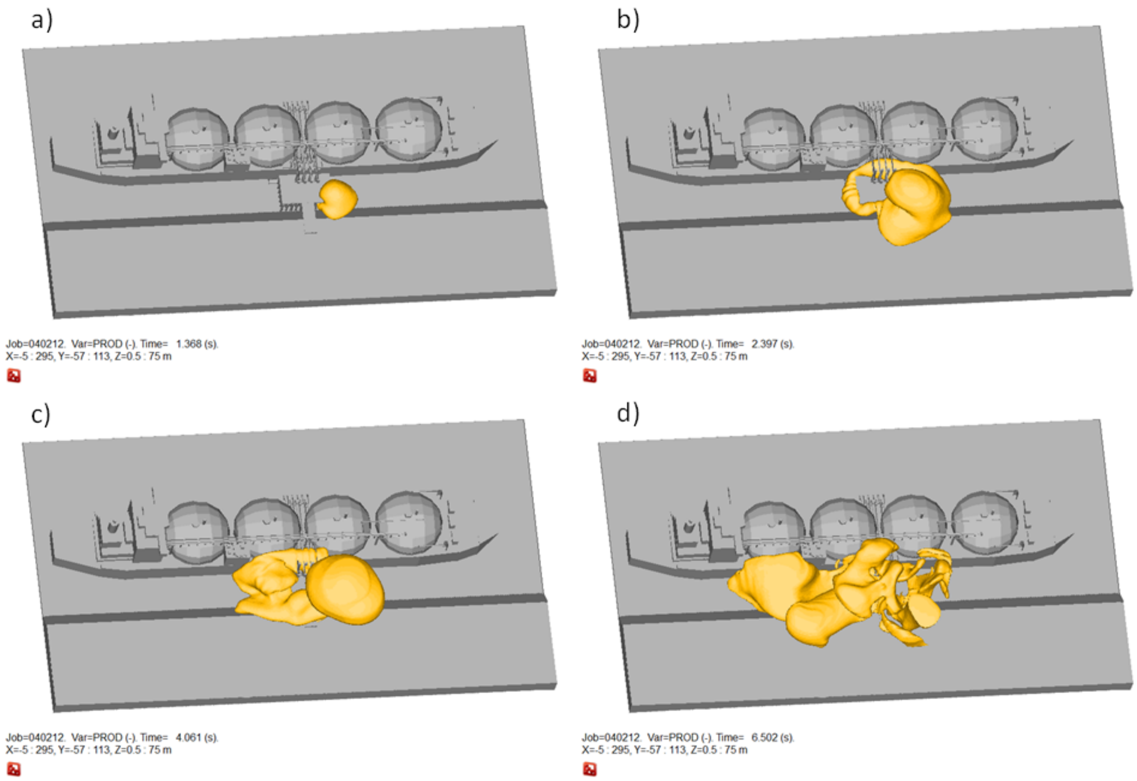

Figure 10. Evolution of the flame front from ignition of "realistic" VCE.

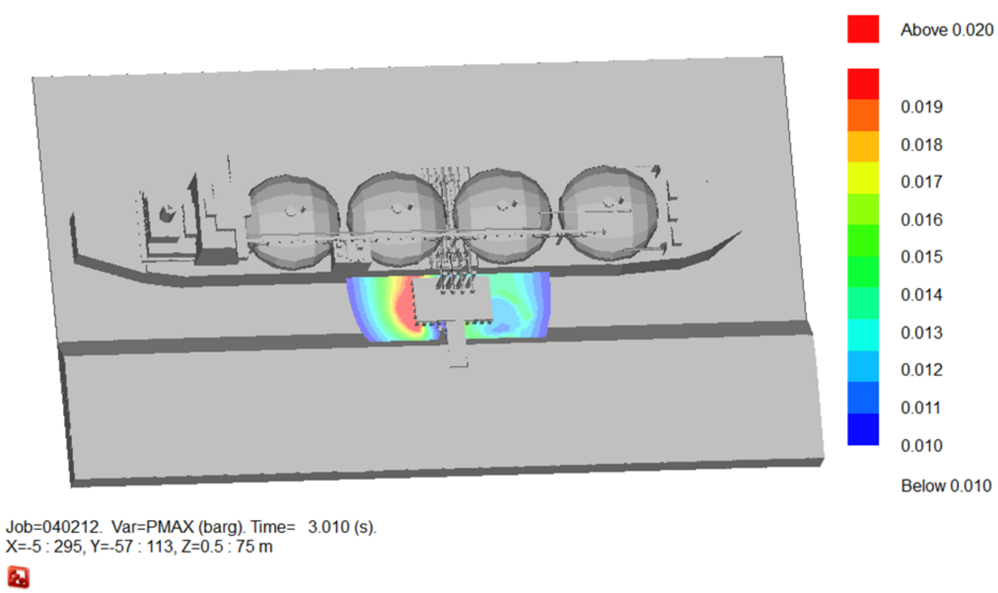

Figure 11. Maximum overpressure on a horizontal plane 7 m above water level, for the realistic scenario VCE. Overpressure color contours are in barg.

Comments

In practical terms, these scenarios are relatively harmless, as overpressures potentially harmful to public safety remain well within the restricted area and the unloading pier and associated equipment are unlikely to be affected. Nonetheless, the case studies show that parameters such as the degree of congestion can have a significant impact on the consequences of vapor cloud explosions. Therefore, it is important to avoid dismissing potential accident scenarios without careful consideration to their potential consequences.

In light of recent events, such as the fuel depot explosion at Buncefield, particular caution is recommended if the area where a spill is possible is surrounded by natural high-congestion areas (e.g., trees, bushes, mangroves, etc.) or if there are highly congested and semi-confined processing areas on the jetty or onshore.

Additionally, even scenarios that do not present a serious safety hazard may benefit from mitigation. For example, a scenario resulting in a maximum overpressure of 1 psig would not require mitigation, however, a realistic simulation of the vapor cloud formation, dispersion and ignition could show a simple and economical way to reduce even this minor hazard.

Conclusions

This paper demonstrated the application of FLACS to the analysis of vapor cloud explosions from LNG spills at an LNG carrier unloading pier. FLACS is a specialized CFD tool developed to address process safety applications such as gas dispersion and vapor cloud explosions. The FLACS vapor dispersion model was recently validated against the entire set of experimental data in the Model Evaluation Protocol for LNG vapor dispersion models. The FLACS vapor cloud explosion model has been validated against hundreds of experiments involving many different flammable gases and geometry configurations.

Even though the low reactivity of methane (the main component of LNG) makes high overpressures difficult to achieve in typical LNG receiving terminals, the possibility of such an occurrence should not be summarily dismissed. Further, even scenarios that do not present a serious safety hazard may benefit from mitigation.

By simulating the entire sequence of events leading up to a vapor cloud explosion (i.e., the LNG spill, pool spread and vaporization, vapor cloud dispersion and ignition) it is possible to obtain realistic and not overly conservative estimates of the potential consequences of such a scenario.

References

[1] AES Sparrows Point, FERC docket CP07-62.

[2] Hansen, O.R., Melheim, J.A., Storvik, I.E., 2007. "*CFD-Modeling of LNG Dispersion Experiments*". AIChE Spring National Meeting, 7th Topical Conference on Natural Gas Utilization, Houston, USA.

[3] Patankar, SV, *"Numerical Heat Transfer and Fluid Flow"*, Hemisphere Publications, New York, NY, 1980.

[4] Hjertager, BH, *"Computer simulation of turbulent reactive gas dynamics"*, J Model Identification Control, 5 (1985), 211–236.

[5] Launder, B.E., Spalding, D.B., 1974, *"The numerical computation of turbulent flows"*. Comp. Meth. In App. Mech. And Eng., **3**, 269-289.

[6] Ivings M J, Jagger S F, Lea C J and Webber D M, 2007, *"Evaluating vapor dispersion models for safety analysis of LNG facilities"*, The Fire Protection Research Foundation, May 9 2007.

[7] Hansen, O.R., Gavelli, F., Ichard, M. and Davis, S.G., 2010 "*Validation of FLACS for Vapor Dispersion from LNG Spills: Model Evaluation Protocol*", J. Loss Prev. Proc. Ind. (in press)

[8] Arntzen, BJ, *"Modeling of turbulence and combustion for simulation of gas explosions in complex geometries"*, PhD Thesis, NTNU, Trondheim, Norway, 1998.