A Unified Model for LNG Pool Spread and Vapor Dispersion: Is Wind Scooping Really A Factor?

Filippo Gavelli[1], Scott Davis, Olav Roald Hansen

GexCon US, 7735 Old Georgetown Rd, Suite 1010, Bethesda, MD 20814

Abstract

U.S. regulations for the siting of LNG facilities require LNG spills to be collected into impounded areas, and the impoundments must be sited in such a way that LNG vapor clouds formed as a result of defined spills dissipate to below ½ LFL before reaching a property line that can be built upon. The traditional method to verify facility compliance with these regulations, which coupled the SOURCE5 model to calculate the LNG vaporization rate and the DEGADIS model to calculate vapor cloud dispersion, has recently been the subject of criticism.

In this paper, the vapor generation and vapor cloud dispersion scenario is simulated using an alternative tool: GexCon's computational fluid dynamics (CFD) model FLACS. FLACS incorporates a 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. In fact, FLACS was successfully validated against the entire Model Evaluation Protocol database for LNG vapor dispersion.

The paper first compares FLACS' unified pool spread and vapor generation model with the traditional, (SOURCE5) method for LNG spills into a sump. The paper then utilizes the FLACS model to evaluate the effect of wind-driven turbulence (the "wind scooping" effect) on vapor dispersion from sumps, to demonstrate that the low wind speed requirement (2 m/s) in current regulations does indeed represent the worst-case scenario for vapor dispersion distances.

Background

U.S. regulations for the siting of LNG facilities require LNG spills to be collected into impounded areas, and the impoundments must be sited in such a way that LNG vapor clouds formed as a result of defined spills dissipate to below 50% of the lower flammable limit (LFL) before reaching a property line that can be built upon [1].

Traditionally, LNG terminal applications have verified the compliance of an impoundment with the flammable vapor cloud dispersion requirement by using the combination of the following models: SOURCE5 to calculate the LNG pool spread and vaporization rate within the impoundment; and DEGADIS to calculate the vapor cloud dispersion.

---

[1] Corresponding author: fgavelli@gexcon.com

Over the last few years, several concerns were raised about this approach, particularly in relation to the assumptions built into the SOURCE5 model. For example, it was pointed out that SOURCE5 assumes that the vapor cloud formed by vaporization of the LNG pool does not mix with air and remains at the boiling temperature until the entire sump volume has been filled [2]. In particular, the model was criticized for not including the "wind scooping" effect, that is, turbulence induced by the air flowing over the sump "scooping" LNG vapors out of the sump. Some have argued that wind scooping may result in longer vapor dispersion distances for higher wind speeds than the regulatory value of 2 m/s. More recently, other incorrect assumptions were identified in the treatment of the LNG pool spread within SOURCE5 [3], leading regulatory authorities in the United States to question its applicability to LNG spills into impoundments and to seek alternative models to evaluate these scenarios. Finally, it must be pointed out that SOURCE5, as most other pool spreading and vaporization models, only accounts for heat transfer from the impoundment floor and not from the walls in calculating the vapor generation rate. While this assumption is reasonable for spills in large and shallow impoundments, it neglects a significant contribution to vapor generation in smaller, deeper sumps.

This paper presents an alternative tool to calculate the vapor generation and vapor cloud dispersion from an LNG spill into a sump: GexCon's computational fluid dynamics (CFD) model FLACS. The model will be applied to hypothetical LNG spill scenarios and the results compared with the SOURCE5 vapor generation predictions. Additionally, FLACS will be used to evaluate the effect of wind scooping on vapor dispersion distances as wind speed increases above 2 m/s.

FLACS

GexCon's CFD model 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.

Pool Spread Model

FLACS incorporates a pool model that allows for the formation and spreading of the pool, with local evaporation rates based on the heat transfer from the ground or water, radiation, local wind speeds and turbulence levels, and the local vapor pressure above the pool [4]. Figure 1 shows a schematic of the physical processes modeled in the FLACS pool spreading model.

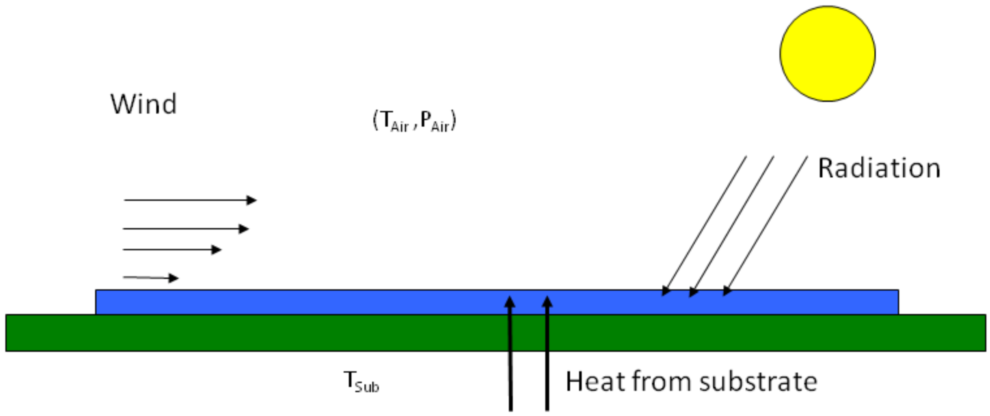

Figure 1. Schematic of the physical processes modeled in FLACS

Vapor Dispersion Model

FLACS solves the Reynolds Averaged Navier-Stokes equations by using the k-ε model with the standard set of constants taken from 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.

Validation

The FLACS CFD tool has been used to simulate all the experimental tests of 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 models need to be validated against key experimental data which are provided in the Model Validation Database, which consists of both large-scale field trials as well as wind tunnel tests. The 33 tests of the Model Validation Database have been simulated with FLACS and the model predictions matched well with the experimental data, meeting the statistical performance requirements in the MEP [7].

LNG Spill and Vapor Dispersion Scenarios

Example 1 – SOURCE5 and Wall Heat Transfer

The first example presented in this paper addresses the LNG vaporization rate as calculated by the following methods:

1. The SOURCE5 model;

2. The pool spread and vapor generation model built in to FLACS, neglecting LNG heat transfer from the walls to the LNG pool;
3. The unified pool spread and vapor dispersion model built in to FLACS, accounting for LNG vapor formation by heat transfer from the walls.

The scenario consists of a 10-minute LNG spill with a flow rate of 0.547 m^3/s (8,670 gpm) into a concrete-lined sump measuring 10 m by 10 m by 3.3 m deep. The terrain is assumed to be flat, with 0.01 m surface roughness, and no obstructions are present other than the sump cavity (Figure 2). The ambient temperature is assumed to be 20°C (68 F), the wind speed is 2.0 m/s (4.5 mph) and the atmospheric stability class is F (stable).

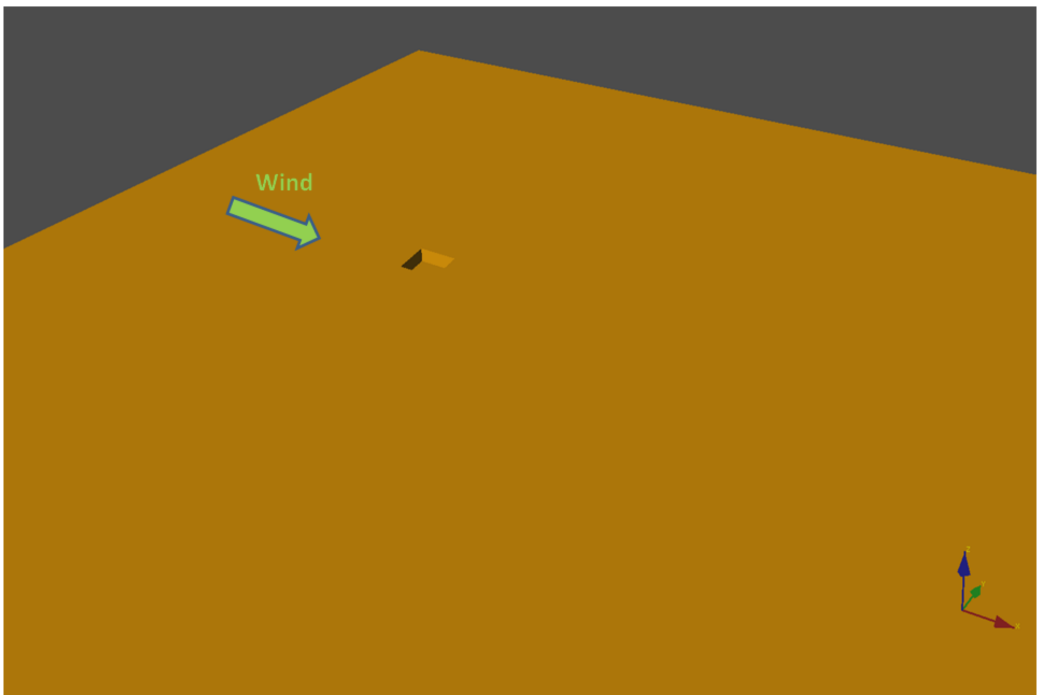

Figure 2. Geometry for scenario LNG sump simulation.

Figure 3 shows the LNG vapor generation rates as obtained following each of the methods above. The vapor generation rates during the floor wetting phase are significantly different between SOURCE5 and FLACS. This is due to the assumption made in SOURCE5, where the vapor generation rate is equal to the LNG spill flow rate until the entire sump floor has been wetted. This is clearly a non-physical assumption, as it would be impossible for the LNG pool to spread on the sump floor if the entire spill flow rate is immediately converted to vapor. Additionally, this assumption makes the SOURCE vapor generation term independent of the sump substrate material until the sump floor is wetted – another non-physical approximation that prevents designers from taking proper credit when utilizing insulating materials as substrate. The FLACS pool spreading model, on the contrary, provides a more realistic estimate of the vapor generation by calculating heat transfer from the floor to the pool based on the distribution of the LNG pool as well as the substrate material of the sump. When only heat transfer from the sump floor is considered, the highest vapor generation rates are obtained at the beginning of the spill, as the pool spreads to wet the sump floor. Once the sump floor is wet, the surface cools down and heat transfer to the pool is rapidly reduced.

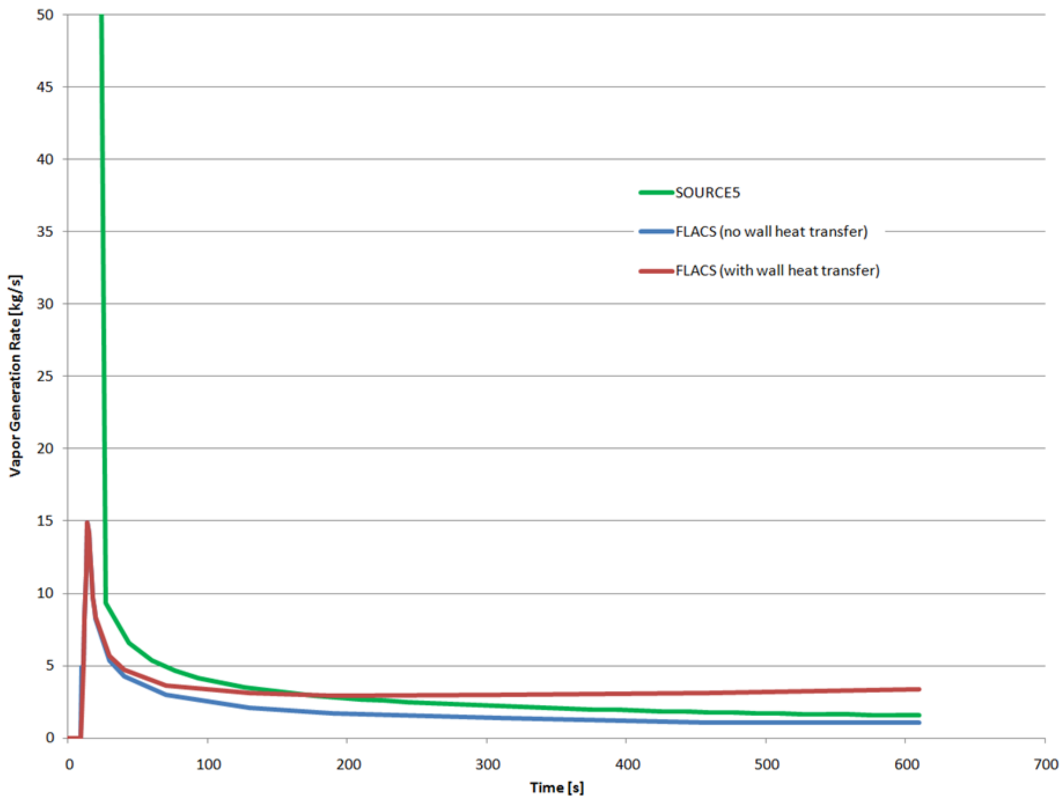

Figure 3. Vapor generation rates for a design spill into a sump. The peak vapor generation rate for the SOURCE5 model, in this example, is approximately 241 kg/s (the plot range is limited to 50 kg/s to better resolve the difference between the curves after the initial phase).

The comparison in Example 1 also demonstrates the effect of heat transfer from the sump walls to the LNG pool, once the floor of the sump has been wetted: as long as LNG is spilled into the sump and the liquid pool depth increases, new portions of the vertical walls continue to be wetted and, as a result, the heat transfer from the walls to the pool increases with time. The contribution of wall heat transfer is responsible for the difference between the red and blue traces in Figure 3. The vapor generation from the walls contributes to the growth of the vapor cloud within the sump and, therefore, results in a larger cloud emerging from the sump, as shown in Figure 4 (approximately 150 s into the spill, which started at time = 10 s) and in a longer vapor cloud dispersion distance, as shown in Figure 5 (approximately the maximum downwind cloud distance).

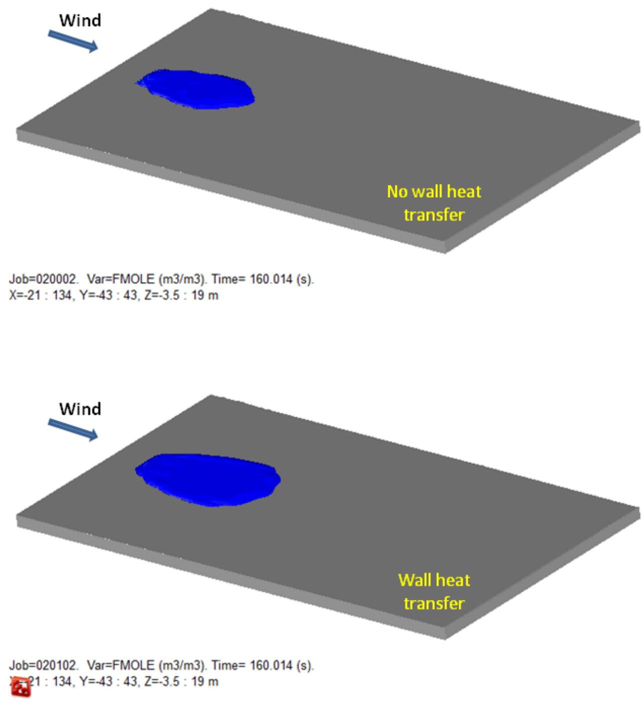

Figure 4. ½-LFL gas concentration isocontours for simulations without (top) and with wall heat transfer (bottom), approximately 150 s into the spill.

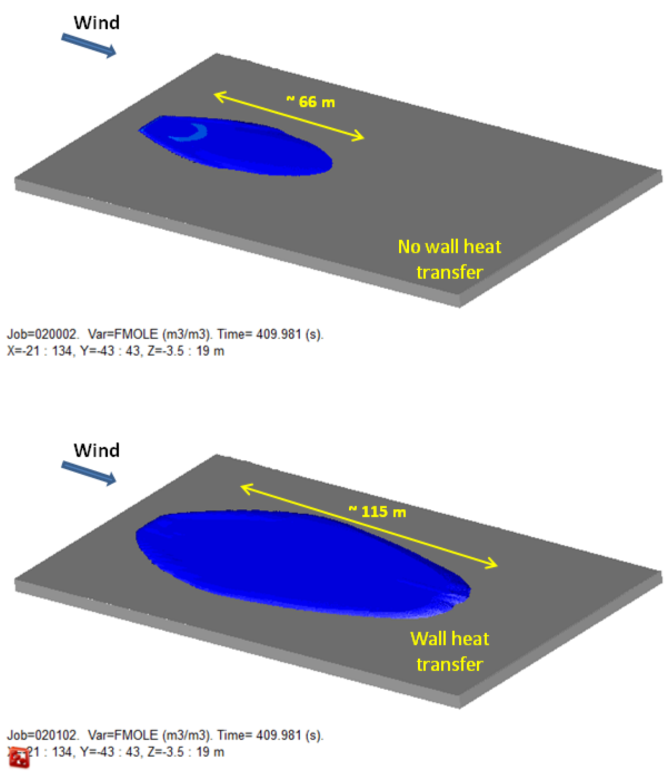

Figure 5. ½-LFL gas concentration vapor cloud at approximately the maximum cloud size, for simulations without (top) and with wall heat transfer (bottom).

The maximum vapor dispersion hazard distances for the FLACS simulations, with and without the contribution of sump wall heat transfer are summarized in Table 1.

Table 1. Maximum distances to 1/2-LFL for FLACS simulations with and without sump wall heat transfer

Solution Method	Max. Distance to ½-LFL
1) FLACS without wall contribution	66 m
2) FLACS with wall contribution	115 m

Example 2 – Wind Scooping

The second example presented in this paper addresses the so-called "wind scooping" effect. The purpose is to evaluate whether the current wind speed requirements in NFPA 59A and in U.S. federal regulations (i.e., a wind speed of 2.0 m/s) represent the worst-case ambient conditions, or a higher wind speed may indeed result in increased hazard to the public.

The scenario consists of a 10-minute LNG spill with a flow rate of 0.547 m^3/s (8,670 gpm) into a concrete-lined sump. The terrain is assumed to be flat and no other obstructions are present. The ambient temperature is assumed to be 20 C (68 F). The wind speed is varied parametrically from 2 to 5 m/s (4.5 to 11.2 mph) and the most stable atmospheric stability class compatible with each wind speed [turner ref.] is assumed: F stability for wind speeds of 2 and 3 m/s, E stability for wind speed of 4 m/s; D stability for wind speed of 5 m/s.

The wind scooping effect is expected to be different for different sump dimensions, particularly as the aspect ratio (defined as the ratio of streamwise length over depth) changes. In order to evaluate the implications on vapor dispersion hazard distances, the same wind speed parameterization is performed for two different sumps, with the same total volume but different aspect ratios:

1. Deep sump: 10 m by 10 m by 3.3 m deep (L/D = 3);
2. Shallow sump: 12.5 m by 12.5 m by 2.1 m deep (L/D = 6).

Figure 6 shows the vapor generation rate for the two sump geometries. The different sump floor areas result in different vapor generation rates between the two geometries: a higher vaporization rate is initially obtained from the shallow sump, due to its larger floor area. The contribution to vapor generation from the sump walls becomes more significant in the deeper sump, as the liquid depth increases at a faster rate, resulting in a slight increase in vapor generation at later times.

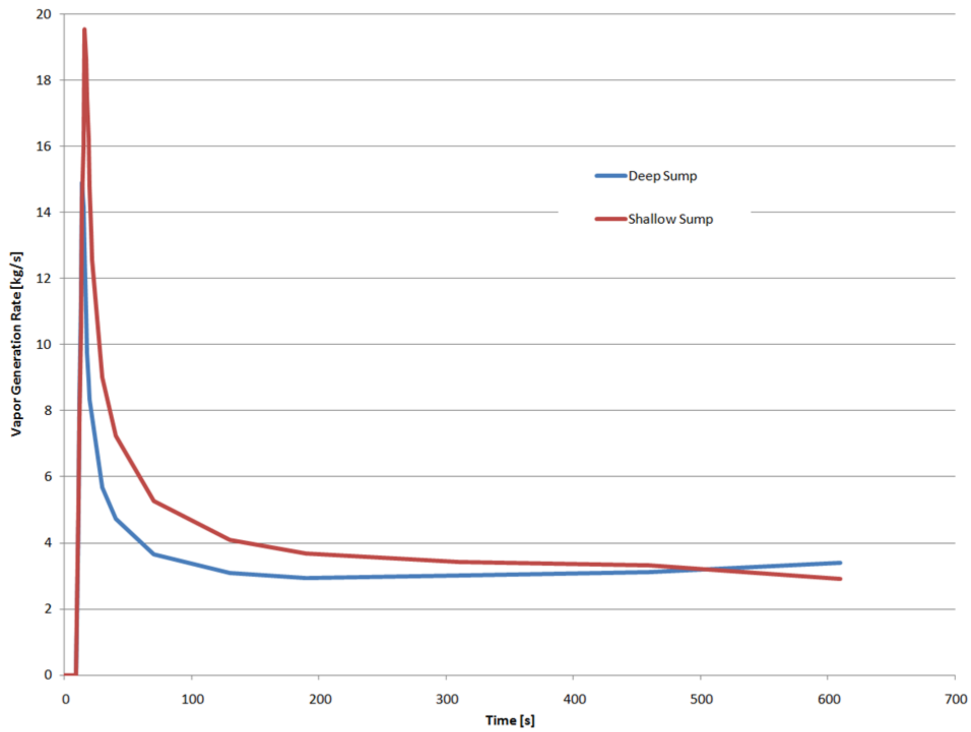

Figure 6. Vapor generation rates for the two sump geometries.

Figure 7 and Figure 8 show the ½-LFL vapor cloud isocontours for the deep sump scenario, respectively 60 and 340 seconds into the spill (which started at time = 10 s). Even though the vapor generation rates are the same for all four cases, higher wind speeds "scoop" a larger volume of LNG vapors (at concentrations equal to or greater than ½-LFL) from the sump, as shown in Figure 7. However, the LNG vapors emerging from the sump are dispersed to below ½-LFL at a faster rate as the wind speed increases, resulting in progressively shorter hazard distances, as shown in Figure 8.

Figure 9 and Figure 10 show the ½-LFL vapor cloud isocontours for the shallow sump scenario, respectively 45 and 320 seconds into the spill (which started at time = 10 s). Similar conclusions can be drawn for this scenario as for the deep sump: higher wind speeds "scoop" a larger volume of LNG vapors (at concentrations equal to or greater than ½-LFL) from the sump (see Figure 9), but the LNG vapors emerging from the sump are dispersed at a faster rate, resulting in progressively shorter hazard distances as the wind speed increases (Figure 10).

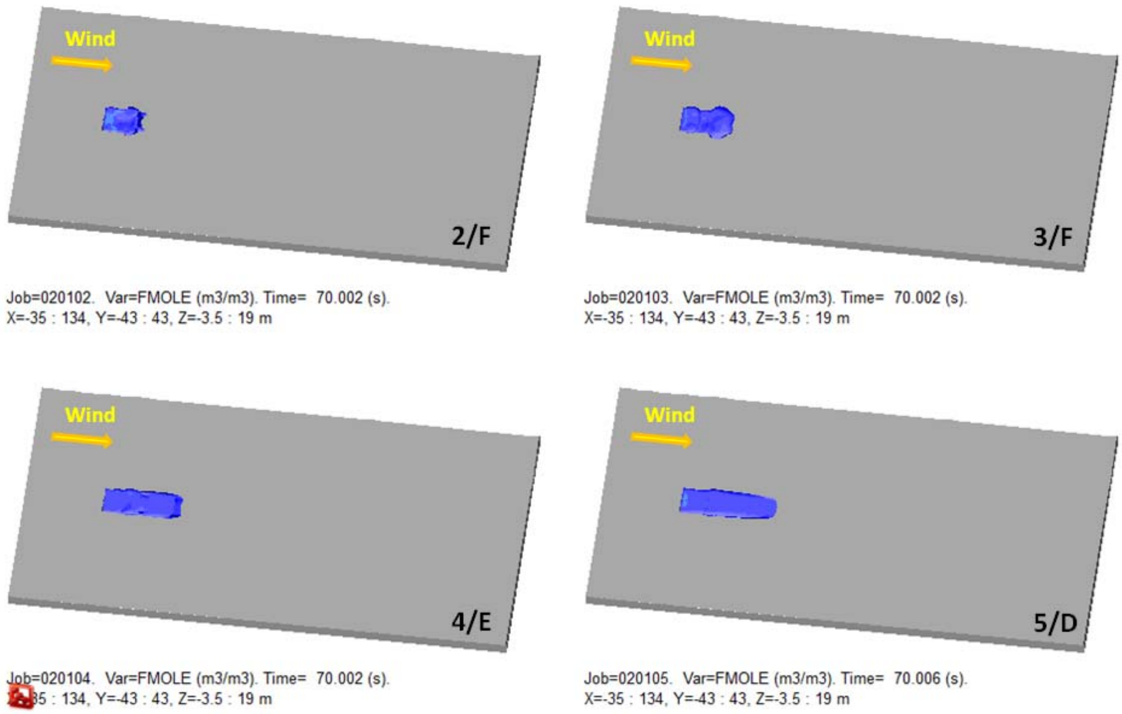

Figure 7. ½-LFL gas concentration isocontours for the deep sump under different wind / stability conditions, approximately 60 s into the spill.

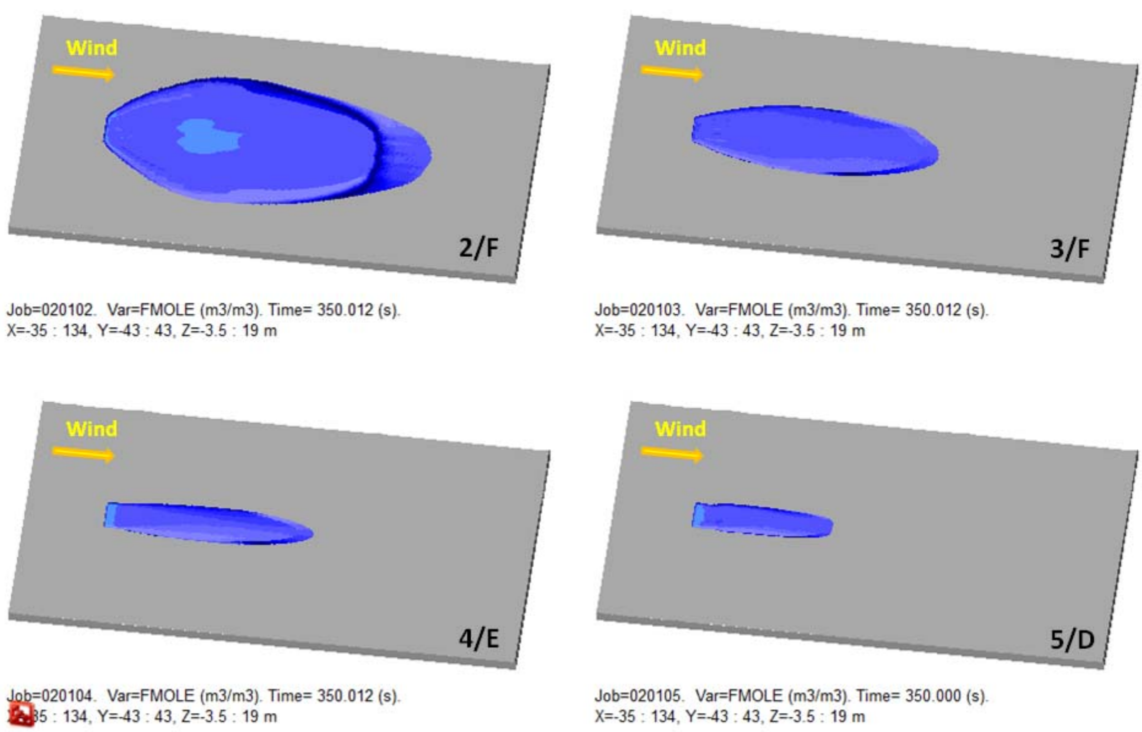

Figure 8. ½-LFL gas concentration isocontours for the deep sump under different wind / stability conditions, approximately 340 s into the spill.

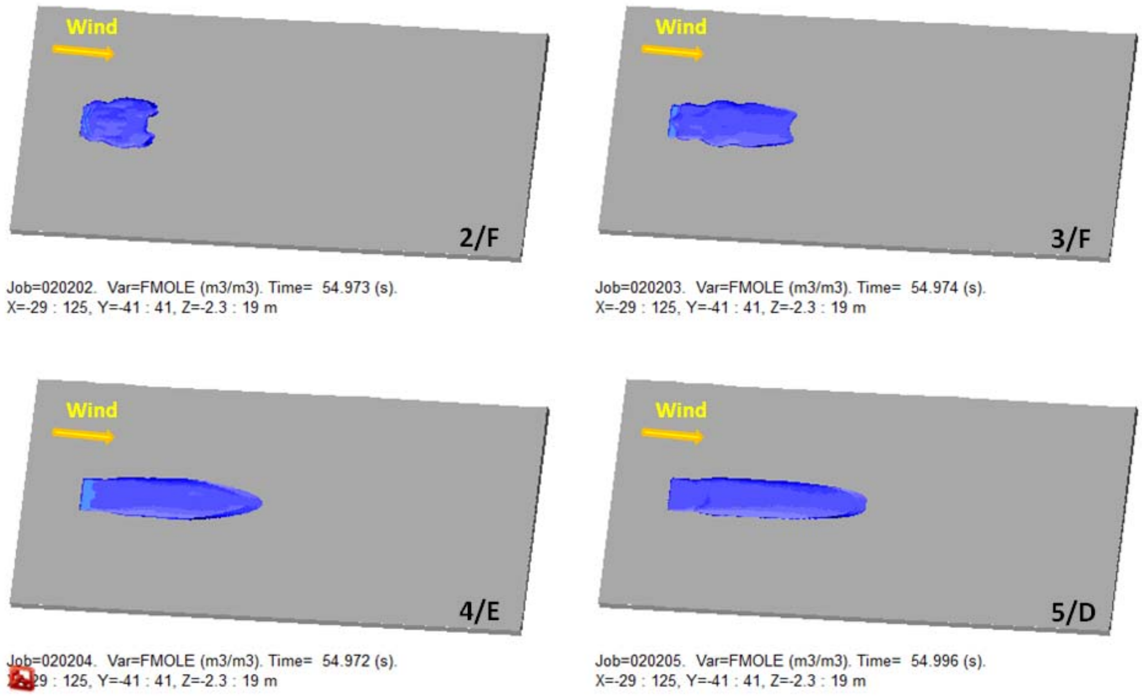

Figure 9. ½-LFL gas concentration isocontours for the shallow sump under different wind / stability conditions, approximately 45 s into the spill.

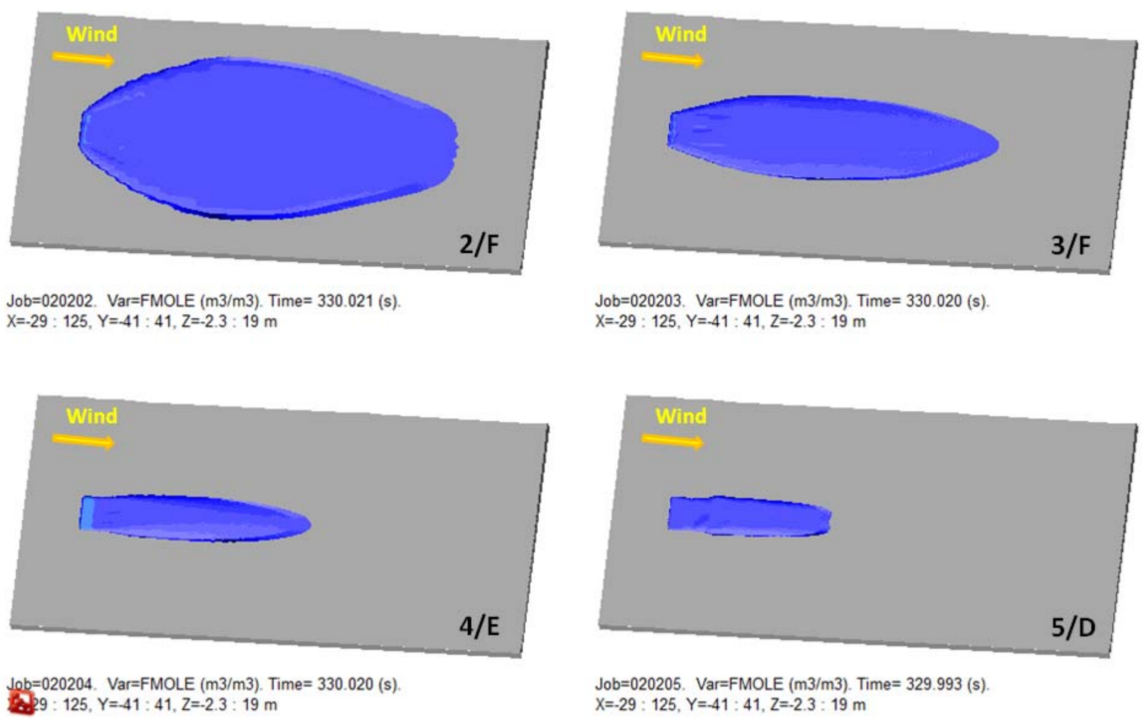

Figure 10. ½-LFL gas concentration isocontours for the shallow sump under different wind / stability conditions, approximately 320 s into the spill.

The maximum vapor dispersion hazard distances, calculated for an LNG spill under different wind conditions and different sump geometries, are summarized in Table 2. The results indicate that the maximum distance to ½-LFL for the vapor cloud from an LNG spill into a sump is reached for the low wind speed case of 2 m/s, as currently required by the NFPA 59A standard and by U.S. federal regulations.

Table 2. Maximum distances to 1/2-LFL for different wind speeds and sump geometries

Wind Speed / Atmospheric Stability	Deep Sump (L/D = 3.0)	Shallow Sump (L/D = 6.0)
2 / F	115 m	123 m
3 / F	86 m	97 m
4 / E	71 m	79 m
5 / D	46 m	66 m

Conclusions

This paper presented an alternative tool to the traditional SOURCE5 / DEGADIS method to calculate the vapor generation and vapor cloud dispersion from an LNG spill into a sump. GexCon's computational fluid dynamics (CFD) model FLACS, which provides an unified source term (i.e., pool spreading and vapor generation) and dispersion model, was applied to the typical scenario that all LNG facilities are required to simulate: a 10-minute LNG spill into a sump.

The pool spreading and vapor generation rate calculated by FLACS for an LNG spill into a sump were compared with the SOURCE5 calculations. Due to the assumptions built into the SOURCE5 model, the vapor generation rate calculated using SOURCE5 appears unphysically large during the sump floor wetting phase of the spill. The effect of heat transfer from the sump walls to the LNG pool was also demonstrated to be significant, at least for sumps with a relatively small footprint and tall walls. This finding should be taken into consideration when evaluating pool spreading and vaporization models in alternative to SOURCE5.

Finally, the wind scooping effect was then evaluated, by simulating the dispersion of an LNG spill into a sump, under increasing wind speeds (from 2 to 5 m/s). The results indicated that increasing wind speed above the current regulatory requirement of 2 m/s does not lead to longer vapor dispersion hazard distances. The CFD simulations showed that while "wind scooping" increases with wind speed, turbulent mixing also increases and at a faster rate, such that the net effect of an increasing wind speed is a reduction in vapor dispersion hazard distances.

References

[1] Title 49 Code of Federal Regulations, Part 193.2059

[2] Havens, J. & Spicer, T., "*United States regulations for siting LNG terminals: Problems and potential*", Journal of Hazardous Materials, *140 (3)*, 2006, pp. 439-443.

[3] Fire Protection Research Foundation, "*LNG Source Term Models for Hazard Analysis: a Review of the State-of-the-Art and an Approach to Model Assessment*", March 2009.

[4] 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.

[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., Ichard, M. and Davis, S.G., "*Validation of FLACS for Vapor Dispersion from LNG Spills: Model Evaluation Protocol*", Proc. 2009 Mary Kay O'Connor Process Safety Symposium

Speaker's Bio

Dr. Filippo Gavelli is the Head of the Dispersion Consulting group at GexCon US, Inc. and specializes in the analysis of heat transfer and fluid flow phenomena, including multiphase flows and cryogenic fluids. He applies his engineering and CFD modeling expertise to the atmospheric dispersion of hazardous gaseous releases, and has extensive experience modeling Liquefied Natural Gas (LNG) vapor cloud dispersion. Dr. Gavelli is responsible for GexCon US' dispersion modeling activities, which include risk assessments and consequence modeling for chemical and petrochemical facilities, offshore installations, hazardous materials transportation and various other applications. He is the lead author of several LNG safety-related papers and has contributed to numerous LNG-related technical committees and expert panels.