Modeling the Vapor Source Associated with the Spill of LNG into a Sump or an Impoundment Area

Nicolas Ponchaut and Harri K. Kytomaa
Exponent®
9 Strathmore Road
Natick, MA 01760

ABSTRACT

In the process of getting a prospective on-shore LNG terminal permitted, the applicant must demonstrate that spills into a sump or into an area of impoundment do not result in the dispersion of ignitable vapors beyond the property boundary (49 CFR 193, NFPA 59A). Numerous models exist for modeling the spreading spill into an impoundment such as PHAST, GASP and many others.

The analysis of spills into an area of impoundment consists of two physical elements, namely the hydraulics of the spreading LNG, and the generation of vapors that then spread downwind as a dense gas. The resulting transient vapor source is then typically entered into a vapor dispersion model, such as DEGADIS, or a model that is able to capture the three-dimensional geometry of the terrain and structures proximate to the spill, such as a CFD model (for example Fluent, CFX, Star-CCM+ and others). To date, the spread of the LNG within the area of impoundment has mostly been modeled using simplifying assumptions regarding the speed of the leading edge, some of which will be described in this paper.

This paper presents a new model for the hydraulics of flowing LNG on flat ground. The model is based on the “shallow water equations” and captures the unsteady complexities of a constant flow rate spill as the LNG spreads within the impoundment. This paper compares the results with validation cases and also with predictions that are based on common simplifying assumptions. The rate of evaporation of LNG is based on the amount of heat that is available from the underlying concrete which is initially at ambient temperature. The model quantifies the amount of LNG that evaporates from the spreading pool based on the transient one-dimensional Fourier conduction law within the concrete.