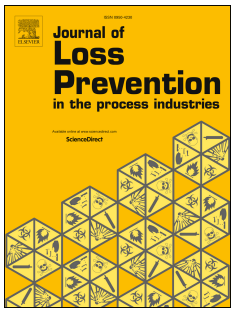
Accepted Manuscript



Numerical simulation of LNG release and dispersion using a multiphase CFD model

Tianpei Luo, Chuangchuang Yu, Ruimin Liu, Mao Li, Jiaxian Zhang, Sheng Qu

PII: S0950-4230(17)30751-9

DOI: 10.1016/j.jlp.2018.08.001

Reference: JLPP 3750

To appear in: *Journal of Loss Prevention in the Process Industries*

Received Date: 25 August 2017

Revised Date: 31 July 2018

Accepted Date: 1 August 2018

Please cite this article as: Luo, T., Yu, C., Liu, R., Li, M., Zhang, J., Qu, S., Numerical simulation of LNG release and dispersion using a multiphase CFD model, *Journal of Loss Prevention in the Process Industries* (2018), doi: 10.1016/j.jlp.2018.08.001.

Numerical Simulation of LNG Release and Dispersion using a multiphase CFD model

Tianpei Luo [a,b], Chuangchuang Yu [b,*], Ruimin Liu [b], Mao Li [b], Jiaxian Zhang [b] and Sheng Qu [b]

[a] State Key Laboratory of Technologies in Space Cryogenic Propellants, Beijing 100028, China

[b] Beijing Institute of Aerospace Testing Technology, Beijing 100074, China

Abstract: Rapid development of Liquefied Natural Gas (LNG) industry and LNG vapor's flammable nature have brought up many safety issues. Computational fluid dynamics (CFD) modeling provides a powerful tool for studying the LNG spill, vapor dispersion, as well as other related safety assessments; however, traditional single phase model is unable to describe the LNG pool formation and phase change, which are essential to risk management. In this paper, we propose an integrated multiphase CFD model to simulate the complete spill process (including LNG release, LNG-water interaction, LNG evaporation and vapor dispersion) in open-obstructed environment based on the Falcon-1 test. The computed results are compared with the experimental data and other simulation results. It is demonstrated that the multiphase CFD model provides better results than other approaches in fully describing the LNG spill process and predicting the peak value of vapor concentration. In addition, the effect of an impoundment on LNG spill and dispersion mitigation is also investigated. The results show that impoundment can significantly control the vapor's dispersion and shorten the hazard area, especially on lateral direction.

Keywords: LNG dispersion; CFD; multiphase model; Falcon test; impoundment

1. Introduction and Motivation

The demand for natural gas supply has been significantly growing over the past few years due to the intense pursuit of clean energy resources worldwide, leading to a rapid development of the Liquefied Natural Gas (LNG) industry. Growing interests in LNG and pressing demand for new LNG terminals to be located close to densely populated area have caused many safety issues on LNG transportation and regasification (Gavelli et al., 2008). Once the containment of LNG on soil, concrete or water is failed during transportation or storage, cryogenic liquid will spread to boil vigorously and form a vapor cloud. This is caused by its higher density relative to air (Flates et al., 2016). The vapor cloud will stay close to the ground and have limited mixing with ambient air. Considering the flammable nature of the cloud, study on its dispersion characteristic and time-dependent plume flow behavior will be very important to the risk assessment and management in LNG application.

Early research focused on large field trials. In 1980 eight LNG spill tests on water were conducted under a variety of meteorological conditions, known as the Burro series (Burro Series Data Report, 1982). These were followed by 10 combustion, rapid transition and

* Corresponding author. Tel. : +86-010-68374824.
E-mail address: chch_yu@foxmail.com(Chuangchuang Yu)