LNG Vapor Dispersion Consequence Modeling with CFX

Ruifeng Qi, William J. Rogers, and M. Sam Mannan
Mary Kay O’Connor Process Safety Center, Department of Chemical Engineering,
Texas A&M University, College Station, Texas 77843-3122
ruifeng.qi@chemail.tamu.edu

Abstract

Regulation 49 CFR 193 and Standard NFPA 49A require the use of validated consequence models to determine vapor cloud dispersion exclusion zones for accidental LNG releases. One tool that is being developed in industry for exclusion zone determination and LNG vapor dispersion modeling is computational fluid dynamics (CFD) codes. The main advantages of CFD codes are that they have the capability of well defining the source term and allowing for the representation of complex geometry and its effects on vapor dispersion.

This paper addresses CFD codes (CFX) for modeling LNG vapor dispersion in the scenarios involving barriers. Discussed are important parameters that are essential inputs to CFD simulation, including atmospheric condition, LNG evaporation rate and pool area, turbulence intensity, source term vapor temperature, ground heat flux, and the effects of obstacles. Sensitivity analysis is being conducted to illustrate the impact of each parameter on the simulation results. In addition, a medium-scale LNG spill test was performed at the Brayton Fire Training School to collect data for validating CFX prediction results. A comparison of test data with simulation results demonstrated that CFX well describes the physical behavior of LNG vapor dispersion, and its prediction results of distances to the Lower Flammable Limit (LFL) shows a good fit to test data.