Using Multiphysics CFD to Model Water Spray Curtain Effects on Dense Gas Dispersion

Sahil Shah and Bharat Tulsyan
Atkins
920 Memorial City Way, Houston TX, USA

Abstract
Accidental release of dense gases such as heavy hydrocarbons and LNG pose a safety hazard as observed during the Buncefield vapour cloud explosion [1]. Difficulties arise in risk mitigation as these releases sink and spread along the ground, and are therefore less conducive to natural dispersion by wind [2]. Water spray curtains are an effective means in overcoming this issue as forced dilution through air entrainment and mixing can push the concentration limits below LEL. While a range of open-field spill tests [3] and scaled wind tunnel experiments [4] have been performed to study this effect, a lesser focus has been placed on numerical simulation using Computational Fluid Dynamics (CFD). In this investigation, multiphysics CFD simulation will be performed using the general purpose software STAR CCM+. RANS turbulence models coupled with Lagrangian sprays will be used to capture water spray curtain effects on atmospheric heavy gas dispersion. The proposed model will be validated against experimental data by comparing spray and no-spray concentration ratios at points both upstream and downstream of the water curtain. Sensitivity to mesh resolution will also be examined. The validated model will then be used to determine the dilution performance for varying release densities and source-to-spray separation distances. CFD modeling poses an effective yet inexpensive platform for designing water spray curtains for hydrocarbon processing and handling facilities. This approach will also provide a framework upon which thermal and smoke shielding effects of water curtains can also be investigated.

Keywords
Dense Gas Dispersion, LNG, CFD, Water Spray Curtains

References