MARY KAY O'CONNOR PROCESS SAFETY CENTER
TEXAS A&M ENGINEERING EXPERIMENT STATION

19th Annual International Symposium
October 25-27, 2016 • College Station, Texas

Multiscale Modeling of Buoyant Fires

Niveditha Krishnamoorthy[1], Alex Graham[1], Alex Read[1], Marco Buonfiglioli[2]

[1]CD-adapco, Melville, NY, USA, 2Atkins Ltd, Surrey, UK Corresponding Author Email:
niveditha.krishnamoorthy@cd-adapco.com

Abstract

The use of computational fluid dynamics (CFD) for understanding the fire hazard potential, can be an alternative to experimental studies for the process safety industry. Detailed experimental measurements are available for some lab-scale and medium-scale fires, however, large-scale fire experiments are significantly more difficult and expensive to conduct and availability of detailed experimental data is scarcer. Furthermore, applicability of available test data to industrial applications is sometimes difficult due to differences (geometry, flow conditions etc.) between the scenario under investigation and that considered during the experiments. Therefore, availability of a reliable technique that can overcome the above shortcomings (cost of experiments, and uncertainties about applicability) could bring benefits to the process safety industry.

The performance of the CFD models in predicting the characteristics of buoyant fires depends on how adequately the fluid dynamics, chemical kinetics, and heat transfer mechanisms in these systems are modeled. In this study, the commercial software package STAR-CCM+ is used for modeling fires of different sizes. The CFD models used in this study are first validated by comparing numerical results against the available experimental data for these fires. The validated models are then used to demonstrate a large-scale fire scenario.

First, axial distribution of temperature and velocity from the Lab-scale experiments of purely buoyant diffusion flames of different heat release rates are compared between CFD simulations and experiments. The CFD model is able to capture the three regimes seen in these flames: (a) a continuous flame region, (b) an intermittent regime, and (c) the plume region. Next, a medium-scale 1m diameter pool fire is modeled, where the mean species concentrations, temperature, and velocity fields are compared with experimental measurements. Finally, a large-scale pool fire scenario is demonstrated using the same modeling approach. Detailed temporal and spatial information of the fire characteristics such as temperature and species concentration distribution, and radiative fluxes from this fire can be easily extracted from the CFD model, which would otherwise be difficult to obtain from field data. CFD simulations can thus be considered as a valuable tool for understanding the fire dynamics under various scenarios and be used to help devise appropriate safety strategies.