Jet Fire Computational Fluid Dynamics Simulations: Validation from an Industrial and Consultancy Perspective

Sean P. Malkeson, Tim Jones, Robert English
MMI Engineering
Warrington, Cheshire, United Kingdom

1 Abstract

A jet fire is a turbulent diffusion flame that results from the ignition and sustained combustion of a continuously released fuel with significant momentum in one or numerous directions. Jet fires can originate from gaseous releases, two-phase mixture releases (i.e. liquid and gas) and releases from pure liquid inventories. The properties of a jet fire depend on numerous factors including fuel composition, release conditions, release rate, release direction, ambient conditions and geometry. Jet fire behaviour can vary significantly depending on the relative effects of these factors. Jet fires represent a significant element of risk associated with major accidents on off-shore facilities. Jet fires often occur in areas of these off-shore facilities which house process equipment and, therefore, the probability of the flames impinging on vessels, pipework and structural supports is high. Due to the nature of the flame-turbulence interaction in jet fire phenomena, the heat fluxes to the impinged objects can be sufficiently high to cause pipework or vessel failure, structural failure or further escalation. The accurate prediction of jet fire behaviour (i.e. prediction of the flame-turbulence interaction, flame shape, propagation speeds, heat release and temperature) is essential in the identification, quantification and mitigation of risk to structures and personnel following the ignition of accidental releases.

The use of Computational Fluid Dynamics (CFD) has become an integral tool for combustion studies in an industrial and consultancy context to understand the effect on plant and personnel. Currently, there are a number of commercially-available CFD software packages that can be used for such combustion analyses. ComputIT KFX and its recently released competitor GexCon AS FLACS-Fire are developed specifically for fire studies. KFX and FLACS-Fire are both fully 3D transient finite-volume solvers with standard, industry-accepted turbulence, combustion and radiation models. Currently, both KFX and FLACS-Fire run single-processor calculations on structured Cartesian grids only. However, there are a number of commercially-available general-purpose CFD software packages (e.g. ANSYS Fluent, ANSYS CFX and CD-Adapco Star-CCM+) that can model combustion phenomena with a large range of turbulence, combustion and radiation models available. These general-purpose solvers can run on unstructured grids, on multiple processors, and both for steady-state and transient calculations.

The current study intends to compare the performance of CFD software packages against available experimental data of propane gas jet fires [1] to assess (i) the accuracy (i.e. prediction of the flame

behaviour) and, (ii) efficiency (i.e. physical run-times) for the considered codes. To achieve this, the current study uses KFX, FLACS-Fire and ANSYS CFX to model a number of propane gas jet fires with varying release conditions. The flame shapes, temperature, velocity profiles and heat release for the jet fires are assessed. The computational costs associated with each code are recorded providing a reference of the potential benefits and shortcomings of each CFD package. Future work will build upon this study to provide a more comprehensive reference of the performances of these codes for the accurate and efficient modelling of multi-phase jet fires and pool fires.

References

[1] Johnson AD, Shirvill LC, Ungut A. (1999). CFD Calculation of Impinging Gas Jet Flames. Health & Safety Executive Offshore Technology Report – OTO 1999 011.