Reliability Engineering and System Safety 143 (2015) 19–29

Contents lists available at ScienceDirect

Reliability Engineering and System Safety

journal homepage: www.elsevier.com/locate/ress

LNG pool fire simulation for domino effect analysis

Muhammad Masum Jujuly, Aziz Rahman, Salim Ahmed, Faisal Khan *

Safety and Risk Engineering Group, Department of Process Engineering, Faculty of Engineering & Applied Science, Memorial University of Newfoundland, St. John's, NL, Canada A1B 3×5

ARTICLE INFO

Available online 3 March 2015

Keywords:
Pool fire
Liquefied natural gas (LNG)
Computational fluid dynamics (CFD)
Domino effect

ABSTRACT

A three-dimensional computational fluid dynamics (CFD) simulation of liquefied natural gas (LNG) pool fire has been performed using ANSYS CFX-14. The CFD model solves the fundamental governing equations of fluid dynamics, namely, the continuity, momentum and energy equations. Several built-in sub-models are used to capture the characteristics of pool fire. The Reynolds-averaged Navier–Stokes (RANS) equation for turbulence and the eddy-dissipation model for non-premixed combustion are used. For thermal radiation, the Monte Carlo (MC) radiation model is used with the Magnussen soot model. The CFD results are compared with a set of experimental data for validation; the results are consistent with experimental data. CFD results show that the wind speed has significant contribution on the behavior of pool fire and its domino effects. The radiation contours are also obtained from CFD post processing, which can be applied for risk analysis. The outcome of this study will be helpful for better understanding of the domino effects of pool fire in complex geometrical settings of process industries.

1. Introduction

Fire and explosion are among the most dangerous accidents in process facilities; especially pool fire is the most frequent incidents. Several catastrophic accidents e.g. Buncefield, UK (2005), Puerto Rico, USA (2009), Sitapura, India (2009) and Bucheon LPG filling station, Korea (1998) were caused by pool fire [1,2]. Pool fire is an uncontrolled combustion of vapor generated from a flammable liquid pool such as, liquefied natural gas, gasoline, jet fuel and so on. The chain of accidents, termed as 'domino effect' may lead to extremely severe consequences. Analyzing past accidental scenarios it is observed that more than half of the industrial domino accidents involved fire as a primary event. Pool fire is responsible for triggering 44% of all physical accidental scenarios which escalates domino effect [8,43]. The direct flame engulfment and steady radiation from the pool fire is the reason for the escalation of this kind of accidents. In order to avoid such calamity a detail study on pool fire is required to save human lives and prohibit the destruction of a facility. To quantify the risk involved with pool fire, it is important to understand its characteristics. Pool fire characteristics largely depend on the fuel mass burning rate which is a function of the fuel properties, pool diameter and the wind speed. Several methods are available in the literature to calculate surface emitting power of a pool fire [3,4].

There are two major types of models available to calculate pool fire characteristics, analytical models and numerical models, such as computational fluid dynamics (CFD) models. The point source model and the solid flame model are two examples of analytical models which have been used to analyze fire radiation hazard for a long time. The point source thermal radiation models are based on the assumptions that the flame is a single point source of thermal energy and the thermal radiation intensity varies inversely with the square of the distance [5]. The point source model can predict radiation in larger distances from the flame but in closer distances it underestimates the thermal radiation. The reason behind this is that the thermal radiation is considers a single point source where as in closer distances flame radiation depends on the size, shape and the orientation of the flame [19]. Another major limitation of the point source model is that it does not consider the effect of smoke. This model also does not consider the wind velocity and direction. For these limitations, point source model is not recommended for modeling large pool fire [6]. Solid flame models and the modified solid flame models are widely used as alternatives of the point source model. In the solid flame model a cylindrical shaped flame zone is considered as a radiating object. In the modified solid flame model two zones are considered: a clear zone and a soot zone with different irradiance power.

Although solid flame and modified solid flame models are well established and validated by experimental results, there are still some drawbacks of using these models. These models assume similar irradiance of fire throughout the solid circle zone. Advanced turbulence model is not used in these models to capture the full dynamics of pool fire in eddy scale. During the wind scenario the tilt

* Corresponding author. Tel.: +1 709 864 8939.
E-mail address: fikhan@mun.ca (F. Khan).

http://dx.doi.org/10.1016/j.ress.2015.02.010
0951-8320/

of the flame as a solid cylinder is practically not valid. In case of complex geometries these models cannot predict the exact behavior of pool fire [7].

Analytical methods are very convenient to calculate the radiation hazard because of their simplicity and accuracy. However, analytical methods are case specific and cannot be applied to complex geometries. Moreover, with analytical methods the domino effect cannot be fully captured. Although numerical methods are relatively complex, they can reliably predict radiation hazard. Few studies have been performed using computational fluid dynamics (CFD) for numerical investigation of fire related hazard [7,10–17]. CFD models have much better temporal and spatial fidelity than point source or solid flame models. However, valid assumptions and boundary conditions are required to analyze pool fire using numerical approach at the preprocessing stage. The simulation time of CFD for a complex geometry may be high. Apart from these constraints, CFD is the most reliable and realistic method for fire simulation. Detailed assessment of the domino effect scenarios require advanced three dimensional fire and explosion or dispersion scenarios and their interaction with structures [8]. CFD codes give the advantage to simulate such scenarios. Khan and Abbasi [9] suggested a mechanism to calculate the probability of occurrence of domino effects and forecast the impacts of such chain accidents. The probability of domino effect occurrence depends not only on the damage potential of the primary accident, but also on a number of other factors of the secondary unit. The post processing results obtained from a CFD simulation can accurately predict the probability of domino effect occurrence.

Several studies have modeled the pool fire and the consequences involved in case of the release of hydrocarbons. Hyunjoo et al. [10] used ANSYS CFX-11 to predict the instantaneous and time-averaged flame temperature and thermal radiation intensity of organic peroxide pool fire. Alireza et al. [11] performed a similar study with organic pool fire using ANSYS FLUENT to predict the safety distance from the pool. Schälike et al. [12] simulated LNG pool fires using ANSYS FLUENT-12: Three different diameters ($d=1$ m, 6.1 m, 30 m) were used to simulate the flame temperature and thermal radiation intensity. In their study, large eddy simulation (LES) is used as the turbulence model. For modeling combustion, the laminar flamelet approach was taken. The discrete ordinates (DO) model is used for radiation and the Moos–Brookes model is used to model soot formation. The objective of their study is to predict the mass burning rate of LNG. Some consequence analysis studies were also performed to predict and quantify the probability to cause serious injury to personnel, major damage to equipment and structure and disruption of operations. Pula et al. [13] used a grid based approach to analyze the consequences for fire and explosion. Mohammad et al. [14] proposed an integrated approach to model the entire sequenced involved in a potential accident; an integrated accident scenario of liquid and gas release was modeled using FLACS and FDS codes. Hansen et al. [15] used FLACS codes in order to simulate the release and dispersion of LNG and compared the result with experimental data to confirm that FLACS is suitable for modeling LNG dispersion. Gavelli et al. [16] analyzed the consequences resulting from the ignition of LNG vapor cloud dispersion during the offloading process. FLACS CFD codes were used to model the vapor cloud dispersion and ignition. The study showed that the sequences of events led to a pool fire after the release of LNG and ignition. Currently ANSYS CFX and FLUENT are becoming more popular for the numerical investigation of fire, explosion, fluid dispersion and consequence analysis. Ruifeng et al. [17] used ANSYS CFX-11 to perform simulations of LNG vapour dispersion and its consequences; a parametric study was performed to study the effects of atmospheric conditions, LNG pool diameter and turbulence intensity, and the presence of obstacles. Sun et al. [7] conducted a 3-D CFD simulation of LNG pool fire using ANSYS FLUENT-14; an advanced turbulence model large eddy simulation (LES) was used to simulate the pool fire with additional sub-models for combustion and radiation. The model outcomes were then compared with experimental results for validation.

In this work, a CFD study is performed to evaluate the effects of environmental conditions on the domino effects of an LNG pool fire. The most important feature of this study is analysis of the effects of pool fire on the surrounding processing units using the CFD post-processing results. From the effect of local temperature of the processing units the safe distance of the adjunct tank with flammable liquids can be determined. The maximum thermal radiation intensity and the temperature received by the processing units can be used to perform hazard analysis.

2. Theoretical framework for CFD simulation

The numerical simulations in this study are carried out with the commercial computational fluid dynamics (CFD) code ANSYS CFX-14. It uses element based finite volume method (FVM) to discretize computational domain utilizing finer meshing [17]. The mesh creates finite volumes which are used to solve mass, momentum, energy equations. Discretization helps to linearize a large system of non-linear algebraic conservation and transport equations [18]. A general solution strategy of ANSYS CFX-14 solver for a steady-state simulation of combustion–radiation model is given in Fig. 1.

Heat transfer through combustion is complex and consists of various physical and chemical processes. These include buoyancy driven flow, turbulence, fuel evaporation, fuel combustion, radiation heat transfer and the interaction between solid structures and radiant heat. These physical and chemical processes are modeled as a set of partial differential equations with boundary conditions. The theoretical framework of a CFD simulation is based on the

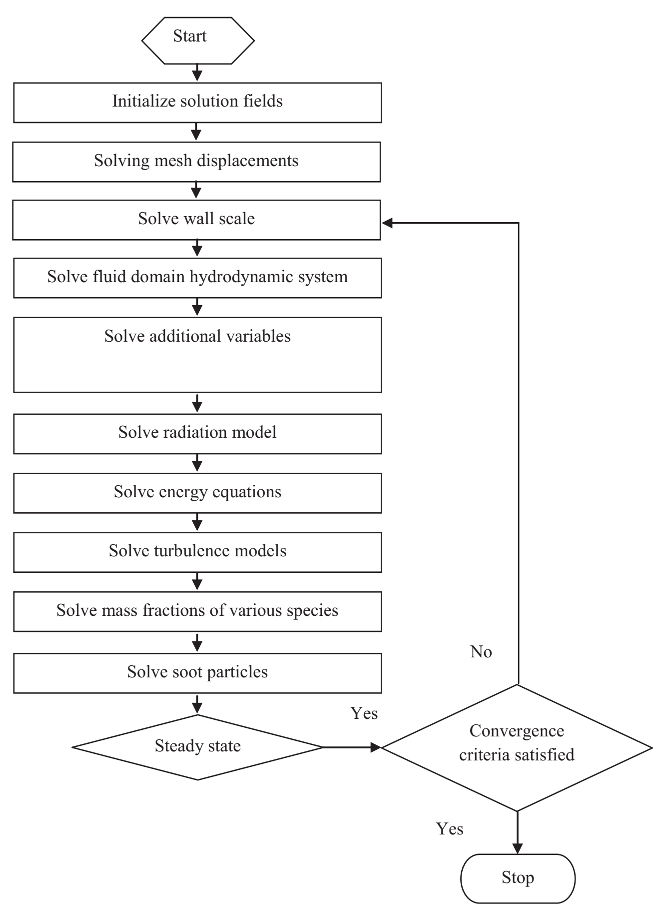

Fig. 1. Solution procedure for a steady-state simulation by ANSYS CFX-14.