Ocean Engineering 157 (2018) 219–233

Contents lists available at ScienceDirect

Ocean Engineering

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

A numerical fire simulation approach for effectiveness analysis of fire safety measures in floating liquefied natural gas facilities

Ruochen Yang [a], Faisal Khan [a,*], Ming Yang [b], Depeng Kong [c], Changhang Xu [c]

[a] Centre for Risk, Integrity and Safety Engineering (C-RISE), Memorial University, St John's, NL, A1B 3X5, Canada
[b] Department of Chemical Engineering, School of Engineering, Nazarbayev University, Astana, 010000, Kazakhstan
[c] Safety Engineering, China University of Petroleum (East China), Changjiang West Road, Huangdao District, 266580, Qingdao, China

ARTICLE INFO

Keywords:
Release simulation
Fire simulation
Offshore processing
Offshore
Process safety
Floating LNG
Computational fluid dynamics

ABSTRACT

Fire remains a serious threat to a floating liquefied natural gas facility. It is of greater concern given the remote locations and limited accessibility of emergency services. This study aims to present a rigorous procedure to study potential accident scenarios in an offshore (floating processing) facility with different ignition source locations and verify the effectiveness of safety measures using computational fluid dynamics code. The uniqueness of the present study is the integration of release, dispersion and fire modeling scenarios, simplifying the fire analysis and increasing its effectiveness from the offshore process system design and analysis perspectives. The first step of the procedure is to identify the range of potential release scenarios and their strength of dispersion in confined and semi-confined spaces. Subsequently, potential fire scenarios are analyzed considering the influence of the location. Computational fluid dynamics models are used to analyze these three steps of the scenarios. Application of the procedure is demonstrated on an offshore facility by analyzing 14 credible scenarios. The ranges of safety measures of these fires are also studied to determine their effectiveness to prevent fires and mitigate their impact. This study provides a simple and efficient way to analyze the impact of key design parameters. In this study, the transition from fire to explosion is not considered and all the environmental factors are assumed to be constants in the simulation.

1. Introduction

Global energy demand is continuously rising. Natural gas being one of the cleanest sources of energy, its demand is sharply rising. Because of the growing demand, many oil companies are currently increasing their investment in floating liquefied natural gas (FLNG) facilities such as Floating Storage and Regasification Units (LNG FSRU) and Floating Production Storage and Offloading (LNG FPSO). With the development of shipbuilding and offshore industries, the concept of the FLNG was recently proposed (Xie et al., 2014). An FLNG facility uses various types of technologies developed for conventional land-based LNG, offshore oil and gas, and marine transport industries (Aronsson, 2012). An FLNG facility can implement gas extraction, gas pre-treatment, natural gas liquefaction, condensate treatment, water treatment, LNG storage, LNG offloading and combined technologies in one offshore facility, which creates a congested and complicated layout (Bunnag et al., 2011).

Fire and explosion accidents such as the Piper Alpha disaster (Ramsay et al., 1994), the BP Texas City disaster (Kalantarnia et al., 2010), the BP Deepwater Horizon explosion (Sammarco et al., 2013), the Cleveland explosion (De Angelis et al., 2012) and Buncefield oil depot fire (Ottemöller and Evers, 2008) have demonstrated the importance of safety in oil and gas operations. In a typical onshore oil refinery or chemical plant, hazardous facilities are usually separated from other parts of the plant. However, on an FLNG, facilities have to be arranged in a congested layout. While this layout brings economic and environmental benefits (Lee et al., 2014), it has a higher fire risk compared to a conventional natural gas processing unit (Xin et al., 2015). In addition, offshore and remote operations usually have limited infrastructure and resources support. All these make it more challenging to assure fire safety in offshore facilities.

Fire accidents caused by flammable hydrocarbons' leakage have been well studied by many researchers (Dadashzadeh et al., 2013; Dan et al., 2014; Darbra et al., 2010; Fay, 2003; Hansen et al., 2007; Hissong, 2007; Ichard et al., 2010; Johnson and Cornwell, 2007; Luketa-Hanlin, 2006; Pitblado et al., 2005; Sun et al., 2014; Yun-sheng and Hua-gang, 2008). The overpressure resulting from a flammable gas explosion is not

---

* Corresponding author.
E-mail address: fikhan@mun.ca (F. Khan).

https://doi.org/10.1016/j.oceaneng.2018.03.052
Received 17 December 2017; Received in revised form 15 March 2018; Accepted 17 March 2018

significant in open areas, while places with confined layouts are dangerous (Dadashzadeh et al., 2013). Luketa-Hanlin (2006) studied the behaviour of LNG spills and pool formations on water and discussed the modeling of LNG spills, taking combustion events such as pool fires and vapour cloud fires into consideration. Fay constructed a model to predict the dynamics of spills from LNG and oil tankers. The pool fire area, duration, and heat release rate were determined using this model (2003). Jet fires, explosions and flash fires occurring on the topside of LNG-FPSOs were analyzed considering different leakage hole sizes. It can be concluded that even though the LNG is safe enough under ALARP criteria, there is a need to select independent protection layers to meet a higher standard (Dan et al., 2014). Sun validated the CFD model of fire radiation by comparing the simulation results and experimental data, followed by a hazard analysis of an LNG Satellite station. The distance between dike walls and AVV banks was suggested to be enlarged by the author (2014). Hissong described the key factors used to model an LNG spill on water; the results from pool fires on land were compared with the results of pool fires on water (2007).

Using computational fluid dynamics (CFD) tools to model the consequences of fire accidents and conduct an analysis has been well validated by many studies and experiments. Dadashzadeh et al. proposed an integrated approach for fire and explosion simulation; FLACS was used to simulate the evaporation and dispersion of flammable gas and delayed ignition, while the Fire Dynamics Simulator (FDS) code was used to model the ignition of the rest of the fuel over the liquid pool (2013). Baalisampang et al. used the FDS code to study fire occurring on a typical FLNG processing facility and its impact on personnel and assets. In his study, a water deluge system was applied to mitigate the impact of fire (2017). In another study conducted by Baalisampang et al. three credible scenarios were identified, and the impact of fire on personnel and assets was determined by combining the FDS code and Probit method (2017). Baalisampang et al. also proposed a method to determine an inherently safe layout design and highlight the importance of improved layout design and passive control strategies (2016). In the study conducted by Hansen et al. (2007), FLACS was used to develop a CFD model to validate the studies of LNG-vapour dispersion; humidity and other effects were considered in this study to design a pool-spread model. PHAST was used by Pitblado et al. to predict the hazard zone caused by an accident or deliberate attack; a range of credible scenarios was developed in this study (2005). Berg et al. identified an optimal safety design for an FPSO by using a CFD model to quantify the overpressure an explosion can cause and also assessed the risk reduction measures using a quantitative method. The effects of barrier walls, separation gaps and other influencing factors were discussed in this study (2000). Van Hees listed many previous validation studies and also conducted several simulations to validate the FDS model. The results of these studies show good correspondence between FDS simulation and experiment results (2013). Binbin conducted a comparative analysis by fire simulation using FLUENT and FDS, and found that although Fluent and CFX have more extensive simulation areas and other advantages in terms of meshing, the result of FDS has high consistency with measured results in some situations (2011). Table 1 shows the main differences of FDS, FLUENT and CFX. In this study the fire analysis is mainly based on the FDS code-a specialty CFD tool developed to study fire dynamics.

Many previous studies have explored natural gas fire accidents (Baalisampang, 2017a, 2017b; Fay, 2003; Hansen et al., 2007; Hissong, 2007; Jin and Jang, 2015; Luketa-Hanlin, 2006; Pitblado et al., 2005) with focus on leakage parameters such as the leakage point, leakage probability, release rate and environmental parameters such as wind speed and direction. However, these studies ignore the impact of the location of the ignition source. In this study, the influence of the effect of the ignition source location is mainly considered.

The objective of this study is to present a rigorous procedure to study potential accident scenarios in an offshore floating processing facility with different ignition source locations and verify the effectiveness of safety measures based on the consequences of potential FLNG fire accident scenarios. The scenarios with different ignition source locations are modeled using the FDS code under the assumption that all the environmental factors are constants in the simulation. Another unique aspect of the present study is consideration of a fire's impact using areas of influence and temperature distribution. Safety measures (firewalls and fire suppression systems) are analyzed for their effectiveness in mitigating the effect of fire. A limitation of this study is that it does not consider the transition from fire to explosion during simulation.

Table 1
Comparative analysis of FDS, FLUENT and CFX.

	FDS	ANSYS FLUENT	ANSYS CFX
Discretization method	Finite-volume method: cell-centered method	Finite-volume method: cell-centered method	Finite-volume method: vertex-centered method
Mesh	Rectilinear mesh, all objects need to be represented by cuboids	Various mesh could be selected according to the shape of objects to give higher accuracy than FDS in meshing	Various mesh could be selected according to the shape of objects to give higher accuracy than FDS in meshing
Model	Combustion model using single step, mixing-controlled chemical reaction which uses three lumped species	Abundant physical models	Abundant physical models
Turbulence Model	For low-speed, thermally-driven flow with an emphasis on smoke and heat. Turbulence is treated by Large Eddy Simulation (LES) or Direct Numerical Simulation (DNS)	A wide range of turbulence models can be selected according to different situations including k-ε model, k-w model, LES model	A wide range of turbulence models can be selected according to different situations
Application	Specialty tool developed to study the fire dynamics	Used widely	Used widely

2. The proposed methodology

This study focuses on the simulation of fires caused by ignition sources located in different places on an FLNG and the verification of the effectiveness of safety measures. It incorporates the release and dispersion modeling of an LNG for the development of various credible scenarios, and employs CFD simulations for each scenario to analyze the fire's impact on the FLNG to determine the most dangerous scenario. This is followed by the implementation of safety measures such as a firewall and automatic fire suppression system to mitigate the fire's impact on human beings, adjacent assets and structures. Fig. 1 demonstrates the procedure of this study.

2.1. Step 1: Scenario development

When developing a credible scenario, many parameters which could contribute to a fire must be considered, such as miscellaneous parameters (pressure, temperature and humidity), fuel parameters (chemical and physical properties), leak parameters (rate, location, duration and direction), wind parameters (speed and direction) and so on. In order to study the impact of locations of ignition sources on safety measure design, all the parameters except the location of the ignition source remain unchanged in all scenarios developed in this study. The users of this approach may choose other parameters that are not limited to those used in this study.