Thin-Walled Structures 105 (2016) 16–28

Contents lists available at ScienceDirect

Thin-Walled Structures

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

Full length article

Numerical simulation of internal gaseous explosion loading in large-scale cylindrical tanks with fixed roof

Ke Hu, Yang Zhao*

Space Structures Research Center, Zhejiang University, Hangzhou 310058, China

A R T I C L E I N F O

Article history:
Received 1 January 2016
Accepted 28 March 2016
Available online 8 April 2016

Keywords:
Cylindrical tanks
Flammable gas
Explosion loading
CFD
Overpressure
Anti-explosion

A B S T R A C T

As a common and important type of thin-walled structure, large-scale cylindrical tanks are widely used for the storage of hazardous petrochemicals. The detonation of those flammable substances generates destructive explosive shock waves in an instant, which may cause serious damage to tanks and even bring economic losses and casualties. Accurate and rational determination of explosion loading on tanks is the foundation for structure damage analysis and explosion resistance design of tanks. For simulating explosion flow field in closed containers, a CFD model was established based on conservation equations, realizable k–ε turbulence models, EDC (Eddy-Dissipation-Concept) combustion models and SIMPLEC (Semi-Implicit Method for Pressure-Linked Equations with Consistent) algorithms. Compared with experimental data reported in existing literatures, the effectiveness and accuracy of the numerical methods and the CFD model were validated. And then on this basis, a series of numerical simulations have been carried out to ascertain the magnitude and distribution of internal explosion loading on tanks, taking into account the influences of tank capacity, H/D (height-to-diameter ratio), roof form, flammable gas species and height of underlying oil level. These indicate that maximum explosion loading on tanks increases along with increase of tank capacity, H/D and reactivity of flammable gas in general. And increases of the underlying oil volume and inert gas content in tanks are conducive to lowering the value of explosion loading. Moreover, tanks having dome roofs and rounded arc junctions show a better anti-explosion performance from the aspect of structural design.

1. Introduction

With the rapid development of the petrochemical industry, large-scale cylindrical tanks are now being widely used for the storage of LNG (Liquefied Natural Gas), LPG (Liquefied Petroleum Gas), oil and other petrochemicals. Fig. 1 shows steel tanks in practical engineering. As is known, these hazardous substances are prone to volatilizing flammable gas in tanks, risking fire and explosion accidents when ignition conditions are present. Once an accident occurs, the destructive explosive shock waves generated in a very short time will cause serious damage to the tank, as shown in Fig. 2, and the spreading fire may bring cascading faults (such as thermal buckling) to surrounding tanks [1], as shown in Fig. 3. Therefore, the knowledge of the fundamental explosive parameters of flammable gas explosion in tanks, such as the explosion loading (the maximum overpressure), the explosion time (the presentation time of the maximum overpressure) and the variation of explosion flow fields, play a significant role in structural damage analysis and anti-explosion design of tanks.

The main research methods used to study gaseous explosions in closed containers are experimental investigation and numerical simulation. In view of the complexities associated with the development of reliable numerical simulation techniques, it is necessary to study explosions in simplified geometries to develop a basic understanding of explosion processes. Experimental investigations on gaseous explosions have been performed [2–8]. Phylaktou et al. investigated the flame speeds and rates of pressure rise for gaseous explosions in a closed cylindrical vessel of large length-to-diameter ratio [2]. The highest rates of pressure rise and flame speeds were observed very early in the explosion evolution in a long vessel; therefore, the authors suggested the initial fast phase is very important and should dominate considerations in pressure relief vent design for vessels of large L/D. Amyotte et al. studied explosions of a stoichiometric ethylene-air mixture in a closed and vented 26 L vessel at various initial turbulence levels and various initial pressures [3]. Their experiments enabled a determination of the impact on explosion development of burning velocity enhancement due to initial turbulence, cellular flame formation and venting-induced turbulence. Gieras et al. experimentally measured explosion pressure and rate of explosion

* Corresponding author.
E-mail address: ceyzhao@zju.edu.cn (Y. Zhao).

http://dx.doi.org/10.1016/j.tws.2016.03.026
0263-8231/

Fig. 1. Steel tanks in practical engineering.

Fig. 2. Accident scene of tank explosion.

Fig. 3. Tank that was affected by a fire [1].

pressure rise as a function of molar methane concentration in a mixture with air in the 40 dm3 explosion chamber [4]. It was stated that the increase in the initial temperature of the methane-air mixture caused a significant increase in the explosion range. Cammarota et al. carried out experimental explosion tests in a 5 L cylindrical tank reactor with stoichiometric methane-air mixtures at non-atmospheric initial conditions [6]. The authors found that an increase in the initial temperature of the methane-air mixture positively affects the burning velocity, the maximum pressure rise and the deflagration index K_G. As the calculation speeds of CPUs rapidly increase, computational fluid dynamics (CFD) has emerged as a numerical simulation technique that has a much greater predictive ability than empirical methods and can be applied to large-scale containers and very complex geometries [9]. In recent years, researchers have become increasingly interested in CFD simulations and meaningful results in numerical simulation have been achieved [10–16]. Ref. [10] described an experimental and theoretical study of premixed flame propagation in a cylindrical vessel containing turbulence-inducing rings. Comparisons between measurements and numerical predictions demonstrated that the mathematical model used provides a reasonable simulation of combustion within the vessel. Maremonti et al. investigated the CFD code AutoReaGas for simulating a gas explosion in two linked vessels [11]. It was concluded that the CFD codes appear to be a valuable tool for analyzing gas explosion experiments in linked vessels for designing safe equipment. Ferrara et al. carried out numerical tests to simulate gas explosions vented through ducts [12]. Their results suggest that the severity of ducted explosions is mainly driven by the vigorous secondary explosion occurring in the duct (burn-up) rather than by the duct flow resistance or acoustic enhancement. Huang et al. studied the laws of transmission of flame and pressure waves in pipeline gas explosions based on the simulation software FLUENT [15]. It turned out that the maximum pressure value of the explosion point is not the maximum value of the whole explosion process; the maximum pressure value of the pressure wave first decreases near the explosion point, then rises to a peak, and then drops gradually. Woolley et al. derived predictions of confined, vented methane-hydrogen explosions in a 70 m^3 vessel using a CFD approach [16]. Based on their calculations, they concluded that hydrogen addition can have a significant effect on overpressure generation and identified safe operating limits for hydrogen addition.

Despite these research results, few studies have been devoted to the analysis of internal flammable gas explosion loading in large-scale containers such as oil storage tanks. In this paper, a CFD model was built using the computational fluid dynamics software FLUENT. First, we simulated the explosion of methane-air mixture in a small-scale cylindrical vessel at almost the same initial conditions of the experiment. The effectiveness and accuracy of the numerical methods and CFD model in simulating gaseous explosions in closed containers were validated against the experimental data reported in Ref [17]; some information rarely observed in experiments was also provided. Based on this validation, a series of CFD simulations were carried out to obtain the magnitude and distribution of internal explosion loading on tanks, taking into account the influence of tank capacity, *H*/*D*, roof forms, flammable gas species and height of the underlying oil level.

2. Numerical model and experimental validation

2.1. Conservation equations and theoretical models

An internal flammable gas explosion in a closed container (e.g., explosion vessels, spherical or cylindrical tanks) is a fast combustion reaction which can be described by the group of 3D unsteady hydrodynamics equations. Because the computational demand of large eddy simulation (LES) or even direct numerical simulation (DNS) is still very high, the RANS (Reynolds-averaged Navier-Stokes) approach has been used in the following group of conservation equations [18,19].