Journal of Loss Prevention in the Process Industries 29 (2014) 13–21

Contents lists available at ScienceDirect

Journal of Loss Prevention in the Process Industries

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

Influence of the shape of mitigation barriers on heavy gas dispersion

V. Busini*, R. Rota

Department of Chemistry, Materials and Chemical Engineering "G. Natta", Politecnico di Milano, Piazza Leonardo da Vinci 32, Milano, Italy

ARTICLE INFO

Article history:
Received 25 September 2013
Received in revised form
19 December 2013
Accepted 8 January 2014

Keywords:
LNG
Mitigation barriers
Heavy gas dispersion
Obstacles

ABSTRACT

Regasification plants have become an emerging risk because their numbers are increasing and concern from the general population towards these systems has grown. Consequently, there is increased interest in investigating the effect of mitigation measures to limit the impact of large accidents on the population living close to the plant. Among the various possible mitigation measures, physical barriers present several advantages; however, it is known that the necessary barrier height can became impractically large to be effective in mitigating the consequences of a large LNG release. Therefore, computational fluid dynamics models were used in this work to analyze the performance of mitigation barriers with different shapes to investigate the possibility of increasing mitigation barrier efficiency by simply changing the main geometrical characteristics of the barrier such as roughness, battlements, or even holes.

1. Introduction

Regasification plants have become an emerging risk because their numbers are increasing and concern from the general population towards these systems has grown. Thus, the risks associated with the storage and transportation of liquefied natural gas (LNG) have been a highly discussed topic in the literature in recent years. In particular, the need to assess this risk has given rise to several studies carried out with both simulation models and experiments on large-scale spills of LNG (Luketa-Hanlin, 2006). A good collection of potential hazards related to handling LNG and techniques to model and analyze the consequences of these hazards is contained in a recent book by Woodward and Pitblado (2010) and in a paper by the same authors (Pitblado & Woodward, 2011; Woodward & Pitblado, 2010).

Moreover, accident histories compiled by Delano (2003), Bainbridge (2003), and LNG World Shipping (2006) reveal that most accidents over the last 50 years happened during operation of the LNG carrier at the dock or inside the plant; these locations are complex environments characterized by the presence of large obstacles (Bainbridge, 2003; Delano, 2003; LNG World Shipping, 2006). This poses the problem of how to define the adequacy of available models for the study of LNG dispersion in real conditions (Ivings, Jagger, Lea, & Weber, 2007) and, more generally, of the dispersion of dense gases in the presence of large obstacles such as real and complex industrial geometries. Several works indeed stress that the common practice is to use integral models, which, on the other hand, are intrinsically unable to include the presence of obstacles because their predictions are realistic and reliable only under open-field conditions (Witlox, Harper, & Pitblado, 2013). Neglecting the effect of large obstacles (such as physical barriers) to the dispersion of dense gases can lead to macroscopic errors (Britter, 1998; Nielsen, 1998). To evaluate the dispersion of dense gases in complex environments, it is therefore necessary to use models developed in the frame of computational fluid dynamics (CFD) as discussed in several works in the literature (Busini et al., 2011; Gavelli, Bullister, & Kytomaa, 2008; Gavelli, Chernovsky, Bullister, & Kytomaa, 2010; Koopman & Ermak, 2007; Luketa-Hanlin, Koopman, & Ermak, 2007; Pontiggia, Busini, Gattuso, Uguccioni, & Rota, 2012; Pontiggia et al., 2010; Tauseef, Rashtchian, & Abbasi, 2011; Zhang, Ning, & Ma, 2009). These models allow the evaluation of obstacle effects (e.g., the size and shape of eddies or the interaction between vortices caused by nearby obstacles) in order to implement simplified formulas in integral models (Scaperdas & Hebden, 2003) or to evaluate the effect of mitigation barriers on the expected hazardous distance (Busini, Lino, & Rota, 2012). In particular, the influence of mitigation barriers on atmospheric dispersion has been an active topic of research in the field of street canyons (Hagler et al., 2012).

In this work, computational fluid dynamics models were used to analyze the performance of mitigation barriers with different shapes. In particular, we employed a case study similar to another recent work that analyzed the effect of simple mitigation barriers on the dispersion of an LNG gas cloud in a regasification terminal (Busini et al., 2012). Our goal was to probe the effects of different barrier characteristics (e.g., roughness, battlements, and holes). The

---

* Corresponding author. Tel.: +39 02 2399 3186.
E-mail address: valentina.busini@polimi.it (V. Busini).

0950-4230/$ – see front matter © 2014 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.jlp.2014.01.001

final aim was to improve dense gas dispersion by increasing the turbulence level behind the barrier and, therefore, the mixing rate between air and the gas cloud.

2. Materials and methods

Computational fluid dynamics codes solve numerically and simultaneously the Navier–Stokes equations of motion, the energy balance and the equation arising from turbulence modeling (Launder & Spalding, 1972; Luketa-Hanlin, Koopman, & Ermak, 2007). The domain is discretized through the use of a calculation grid that allows transformation of the partial differential equations into a system of algebraic equations.

In this work, the k–ε model was used to represent the effects of the turbulence. This model was complemented with an Atmospheric Stability sub-Model (ASsM) that ensures the consistency of the CFD results with the Monin–Obukhov theory (Pontiggia, Derudi, Busini, & Rota, 2009).

The reliability of the CFD model used in all computations reported in this work has been previously verified by comparison with experimental measurements both in free-field conditions and in the presence of large obstacles (Pontiggia et al., 2009; Pontiggia et al., 2011).

The commercial package Fluent 12.1.2 (ANSYS Inc., 2009) was used for all computations together with the boundary conditions summarized in Table 1.

For the sake of comparison, the Process Hazard Analysis Software Tools (PHAST) software was also used (DNV, 1999). PHAST can examine the progress of a potential accident from the initial release to the far-field dispersion including modeling pool spreading and evaporation through integral models, which are unable to account for the presence of large obstacles as previously discussed.

To size the mitigation barrier, a previously developed criterion was used (Derudi, Bovolenta, Busini, & Rota, 2014); here, it suffices to mention that the dimensionless parameter R^* allows the characterization of different types of obstacle. Such a parameter is defined as the minimum between two other parameters: the ratio between the height of the frontal face of a given obstacle, h_{obs} (or width w_{obs}), and the cloud height h_{cld} (or cloud width w_{cld}) evaluated under free-field conditions (that is, without any obstacles):

$$R_h = \frac{h_{\mathrm{obs}}}{h_{\mathrm{cld}}} \tag{1}$$

$$R_w = \frac{w_{\mathrm{obs}}}{w_{\mathrm{cld}}} \tag{2}$$

$$R^* = \min(R_h, R_w) \tag{3}$$

It has been shown that an obstacle's influence on the hazardous distance can be disregarded for $R^* < 0.25$ while it must be considered for $R^* > 1$. The range $0.25 < R^* < 1$ represents a sort of transition zone where the influence of the obstacle cannot be foreseen (Derudi et al., 2014). Therefore, an effective mitigation barrier should be characterized by a value of $R^* \geq 1$.

Table 1
Boundary conditions.

Ground	Wall @ 300 K, roughness = 0.06 m
Walls	Adiabatic wall, roughness = 0.005 m
Pool	During atmospheric stabilization: Wall @ 300 K, roughness = 0.01 m During pool evaporation: mass flow inlet After the end of pool evaporation: adiabatic wall
Wind inlet, domain sides, sky	Velocity inlet
Wind outlet	Pressure outlet

3. Results and discussion

As a case study, a release of LNG deriving from the full-bore rupture of a pipeline was selected. The characteristics of both the pipeline and storage are reported in Table 2.

The modeling of the LNG dispersion was performed for a 5D stability class and 5 m/s wind speed at 10 m above the ground with the suite package PHAST to define the pool dimensions deriving from the spill and the evaporating mass flow; the results of this simulation in terms of vaporization rate are shown in Fig. 1 while the lower part of Fig. 2 illustrates the maximum distance at which the LNG concentration reaches the lower flammability limit (LFL). This distance is not representative of a specific time after the start of the release; rather, it shows the area where hazardous concentration values larger than the LFL are expected. According to Fig. 2, PHAST predicts that the cloud takes an elongated shape typical of dense gas releases with a fair amount of spreading in the initial part and a progressive narrowing up to dissipation; the maximum distance reached by the cloud is approximately 570 m from the center of the pool. These results are expected to be reliable in the absence of large obstacles because PHAST has been successfully validated in comparison with experimental data obtained in the open field.

This pre-modeled source term was used in the CFD simulations by considering a pool with a radius of 5 m and a mesh built using GAMBIT (ANSYS Inc., 2004) size functions to make the grid denser in critical areas; the size of the domain of integration was 1000 × 50 × 800 m.

To simulate the initial expansion and the subsequent shrinkage of the pool, the boundary conditions of the surface, which were initially set as *wall* with the same characteristics as the terrain during the first phase of wind stabilization, were changed to *mass flow inlet* and then *adiabatic walls*.

The CFD results obtained under open field conditions are compared with those of the integral model in Fig. 2 where the upper part shows projections of the LFL contour on the ground obtained with two different grid sizes ($7 \cdot 10^4$ cells and $4 \cdot 10^6$ cells). We can see that the results obtained with the larger number of cells are in fair agreement with those obtained with the smaller number of cells; therefore, proving that the computed hazardous distance is reasonably grid independent. As in the PHAST simulation, the cloud develops mainly along the wind direction even if the spreading is less significant and the narrowing takes place less gradually, which leads to a more abrupt edge of the cloud. The maximum distance reached by the cloud is approximately 520 m from the center of the pool, which is in reasonable agreement with the value predicted by PHAST.

The open-field CFD simulation also evaluated the height of the cloud. Based on the results, the necessary height of the mitigation barrier can be estimated as approximately 6–7 m tall using the aforementioned constraint $R^* = 1$ (see Fig. 2B).

For all the shapes investigated, the mitigation barrier was positioned at 150 m from the pool's center and was 450 m wide (in

Table 2
Characteristics of both pipeline and storage.

Pipe diameter	1 m
Total inventory	45,000 kg
Temperature	111 K
Pipeline length	20 m
Density	450 kg/m^3