Ocean Engineering 111 (2016) 524–532

Contents lists available at ScienceDirect

Ocean Engineering

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

CFD analysis of natural gas dispersion in engine room space based on multi-factor coupling

Xiao Jian Li [a,*], Rui Ping Zhou [a], Dimitrios Konovessis [b]

[a] School of Energy & Power Engineering, Wuhan University of Technology, Wuhan, China
[b] School of Mechanical & Aerospace Engineering, Nanyang Technological University, Singapore

ARTICLE INFO

Article history:
Received 10 May 2015
Accepted 17 November 2015
Available online 8 December 2015

Keywords:
CFD
Natural gas dispersion
Engine room space
Gas detector

ABSTRACT

Gas supply systems typically located at the engine room space of LNG gas-fueled ships, present potential threats for fire and explosion due to gas leakage and dispersion. A validated Computational Fluid Dynamics (CFD) model is proposed for natural gas dispersion analysis in an engine room space under multi-factor coupling. Results demonstrate that gas dispersion depends on leakage rate, position and direction of release, temperature gradient, ventilation and the machinery equipment located in the engine room. Under the impact of air flow, temperature gradient and gas-buoyancy, natural gas tends to accumulate on the top of the engine room space, which can be pumped out through air-outlets. Natural gas is likely to concentrate at areas where vortex flows generate. Effective arrangement and locations of gas detectors are discussed considering the variation of gas concentration at several selected point within the engine room space.

1. Introduction

Design of LNG gas-fueled ships currently receives considerable research and development focus within the maritime industry, as they provide a robust solution platform for compliance with strict emission regulation and economic effectiveness (Stanley et al., 2007). Fuel storage tanks are filled with LNG through bunkering, which is then vaporized to natural gas when passing through the evaporator and transported into the engine room space. In this process, pipelines in engine room space may perforate, crack or break owing to several factors like material aging, mechanical vibration, or corrosion. As a result, accidental leakage and dispersion of natural gas may occur, leading to fire and explosion (Salem, 2010).

Modeling leakages and dispersion of natural gas is the most critical step in assessing these fire and explosion risks in engine room space. Many traditional models and methods used for gas dispersion in atmosphere are available to simulate the dispersion of hazardous gas (Rigas et al., 2003; Labovsky and Jelemensky, 2011). These models are simplifications of the conservation equations for mass, momentum and energy, and are only suitable for gas dispersion in the open air. However, CFD models have been extensively used for more complex environments within buildings or manufacturing plant that should consider the impact of barriers, ventilation and the location of detectors. Currently, CFD is used and validated for a great number of analyses of dense gas dispersion in indoor and outdoor environments (Luketa-Hanlin et al., 2007; Paik et al., 2010; Meroney, 2012; Siddiqui et al., 2012; Sun et al., 2013). As for buoyant gas, like hydrogen and natural gas (Prasad et al., 2011), have merely assessed the capability of a CFD software package to simulate the dispersion of buoyant gases in a partially confined geometry, aiming at the dispersion behavior of highly buoyant gases. (Ivings and Santon, 2009) have made great contributions on area classification of natural gas installations, then focusing on the effectiveness of the ventilation for preventing the buildup of natural gas. They have validated a CFD model using tracer gas, as discussed in Section 2.2 of this paper, and developed a method for the assessment of ventilation effectiveness by considering the average gas concentration across the air-outlets. Newton et al. (2014) assessed the use of CFD to model the ventilation of a working marine vessel. This represents a significant undertaking when utilized in the design stage of optimized ventilation systems for engine rooms. However, this research did not consider the hazardous gas dispersion.

This paper presents research on the modeling of natural gas dispersion in engine room spaces of LNG gas-fueled ships, considerably more complex indoor environments, with focus on the impact of large volume, leakage rate, direction and position of release, presence of complex obstacles (machinery equipment), temperature gradient and ventilation. Effective arrangement and location of gas detectors is required in ventilating such enclosed spaces. The application of a validated commercial CFD code, CFX, is described in illustrating diffusion law in the engine room space of

* Corresponding author. Tel.: +86 15072433854; fax: +86 27 86560042.
E-mail address: tobeloyalty@163.com (X.J. Li).

http://dx.doi.org/10.1016/j.oceaneng.2015.11.018
0029-8018/

Nomenclature

ρ	density of mixed gas, kg/m^3
$\overline{U}$	velocity vector
μ_t	turbulent viscosity, Pa s
ρ_a	density of air, kg/m^3
$\overline{T}$	averaged temperature, K
K_t	turbulence coefficient of thermal conductivity
C_p	constant-pressure specific heat for mixed gas, J /(kg K)
C_{pv}	constant-pressure specific heat for air, J/(kg K)
$\overline{c_i}$	component concentration
Y_i	local mass fraction of each species
S_i	rate of creation by addition, kg/(m^3 s)
$\overline{J_i}$	diffusion flux of species i, kg/(m^2 s)
$D_{i,m}$	mass diffusion coefficient for species i in the mixture
$D_{T,i}$	thermal diffusion coefficient
W_n	theoretical release rate associated, kg/s
d_n	specific release hole diameter, mm
P_s	normal operating pressure, kPa
K	specific heat capacity ratio
MW	molecular weight, kg/mol
R	universal gas constant, J/(mol K)
T_s	storage or normal operating temperature, K
α_1	constant of the function of $k-\omega$
α_3	constant of the function of α_3
α_3	constant in the ωequation of $k-\omega$ model
β_1	constant of the function of β_3
β_2	constant of the function of β_3
β_3	constant in the ω equation of $k-\omega$ model
σ_{k1}	constant of the function of σ_{k3}
σ_{k2}	constant of the function of σ_{k3}
σ_{k3}	constant in the K and ω equation of $k-\omega$ model
$\sigma_{\omega1}$	constant of the function of $\sigma_{\omega3}$
$\sigma_{\omega2}$	constant of the function of $\sigma_{\omega3}$ and ω equation
$\sigma_{\omega3}$	constant in the ω equation of $k-\omega$ model
ϕ	heat emission from main engines and generators

LNG gas-fueled ships. A validated CFD model is detailed concerning the influence of buoyancy and thermal force, mainly emphasizing the $k-\omega$ based shear-stress transport (SST) model. Gas leakage and dispersion is then simulated under multi-factor coupling. A case study on the effective location arrangement of gas detectors is presented aiming to demonstrate the mitigation of accidental release consequences.

2. CFD model for natural gas dispersion

2.1. CFD modeling approach

The Navier–Stokes equations are applied to this case, as dispersion in engine room space is a viscous flow, with transport phenomena of friction and thermal conduction included. In addition, dispersion is a turbulent flow bringing about fluctuating flow variables. Reynolds time-averaging procedures are used to smooth out these fluctuations adding Reynolds-averaged Navier–Stokes (RANS) equations and turbulence model. Eqs. (1) and (2) are the continuity and the momentum conservation equations, respectively.

$$\frac{\partial}{\partial t}(\rho)+div(\rho\overline{\mathbf{U}})=0 \tag{1}$$

where ρ is the density of mixed gas, and $\overline{\mathbf{U}}$ is the velocity vector.

$$\frac{\partial(\rho u_i)}{\partial t}+\frac{\partial}{\partial x_j}(\rho\overline{u_i u_j})=-\frac{\partial p}{\partial x_i}-\frac{\partial}{\partial x_j}\left(\rho\overline{u_i^{'}u_j^{'}}\right)+\frac{\partial}{\partial x_j}\left(\mu_t\frac{\partial\overline{u_i}}{\partial x_j}\right)+\frac{\partial}{\partial x_i}\left(\mu_t\frac{\partial\overline{u_j}}{\partial x_i}\right)+(\rho-\rho_a)g \tag{2}$$

where μ_t is the turbulent viscosity and ρ_a is the density of air. Additional terms for the Reynolds stress $-\rho\overline{u_i^{'}u_j^{'}}$ are included, representing the effects of turbulence, which vary highly and their spatial and temporal dependence cannot be expressed in simple algebraic terms. These Reynolds stresses must be modeled for Eq. (2) to be representative. The Boussinesq hypothesis is a common method to relate the Reynolds stresses to the mean velocity gradients, as Eq. (3) illustrates.

$$-\rho\overline{u_i^{'}u_j^{'}}=\mu_t\left(\frac{\overline{u_i}}{x_j}+\frac{\overline{u_j}}{x_i}\right)-\frac{2}{3}\left(\rho k+\mu_t\frac{\partial u_k}{\partial x_k}\right)\delta_{ij} \tag{3}$$

The Boussinesq hypothesis is used in the Spalart–Allmaras model, $k-\varepsilon$ and $k-\omega$ models. According to Ivings et al. (2010), the shear-stress transport (SST) $k-\omega$ model is accurate in simulating the indoor dispersion of natural gas.

The SST model has the characteristic of effectively blending robustness in the near-wall region with the free-stream independence of the $k-\omega$ model in the far field. This model gives highly accurate predictions of the onset and the amount of flow separation under adverse pressure gradients. Eqs. (4) and (5) are the transport equations for the SST $k-\omega$ model.

$$\frac{\partial(\rho k)}{\partial t}+\nabla\cdot(\rho Uk)=\nabla\cdot\left[\left(\mu+\frac{\mu_t}{\sigma_{k3}}\right)\nabla k\right]+P_k+P_{kb}-\beta'\rho k\omega \tag{4}$$

$$\frac{\partial(\rho\omega)}{\partial t}+\nabla\cdot(\rho U\omega)=\nabla\cdot\left[\left(\mu+\frac{\mu_t}{\sigma_{\omega3}}\right)\nabla\omega\right]+(1-F_1)2\rho\frac{1}{\sigma_{\omega2}\omega}\nabla k\nabla\omega+\alpha_3\frac{\omega}{k}P_k+P_{\omega b}-{}^{\omega b}\beta_3\rho\omega^2 \tag{5}$$

where P_k is the turbulence production due to viscous forces, using Eq. (6).

$$P_k=\mu_t\nabla U\cdot\left(\nabla U+\nabla U^T\right)-\frac{2}{3}\nabla\cdot U(3\mu_t\nabla\cdot U+\rho k) \tag{6}$$

When the Boussinesq buoyancy model is being applied, the buoyancy production P_{kb} is modeled as

$$P_{kb}=\frac{\mu_t}{\rho\sigma_\rho}\rho\beta g\cdot\nabla T \tag{7}$$

The additional buoyancy term in the ω equation reads:

$$P_{\omega b}=\frac{\omega}{k}((\alpha_3+1)C_3\max(P_{kb},0)-P_{kb}) \tag{8}$$

where the coefficients of this model, $\alpha_3,\beta_3,\sigma_{k3}$ and $\sigma_{\omega3}$ are a linear combination of the corresponding coefficients shown as

$$\Phi_3=F_1\Phi_1+(1-F_1)\Phi_2 \tag{9}$$

where F_1 is the blending function. The values of the coefficients used are the following (ANSYS, 2009):

$$\beta'=0.09,\alpha_1=5/9,\beta_1=0.075,\sigma_{k1}=2,\sigma_{\omega1}=2,\alpha_2=0.04,$$
$$\beta_2=0.0828,\sigma_{k2}=2,\sigma_{\omega2}=1/0.856$$

Based on the first law of thermodynamics and Fourier law of heat conduction, Eq. (10) represents the energy equation for a fluid element within the flow.

$$\frac{\partial(\rho\overline{T})}{\partial t}+\frac{\partial}{\partial x_i}(\rho\overline{u_i}\overline{T})=\frac{1}{C_p}\frac{\partial}{\partial x_i}\left(K_t\frac{\partial\overline{T}}{\partial x_i}\right)+\frac{C_{pv}-C_{pa}}{C_p}\left[\left(\frac{\mu_t}{\sigma_c}\right)\frac{\partial\overline{c_i}}{\partial x_i}\right]\frac{\partial\overline{T}}{\partial x_i} \tag{10}$$