Fire Safety Journal 63 (2014) 1–8

Contents lists available at ScienceDirect

Fire Safety Journal

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

Modeling and analysis of water discharge trajectory with large capacity monitor

Tatsuya Miyashita [a,*], Osami Sugawa [b], Tomohiko Imamura [b], Kyoko Kamiya [c], Yasuo Kawaguchi [d]

[a] Graduate School of Science and Technology, Department of Mechanical Engineering, Tokyo University of Science, 2641 Yamazaki, Noda 278-8510, Japan
[b] Department of Mechanics Systems Engineering, Tokyo University of Science, Suwa, 5000-1 Toyohira, Chino 391-0292, Japan
[c] Graduate School of Environmental and Information Sciences, Yokohama National University, 79-9 Tokiwadai, Hodogaya-ku, Yokohama 240-8501, Japan
[d] Department of Mechanical Engineering, Tokyo University of Science, 2641 Yamazaki, Noda 278-8510, Japan

ARTICLE INFO

Article history:
Received 19 November 2012
Received in revised form
29 May 2013
Accepted 3 September 2013
Available online 16 November 2013

Keywords:
Full-blown fire
Large capacity monitor
Water discharge trajectory
MPS method
Spreadsheet model

ABSTRACT

In the event that a full-blown fire occurs in a large fuel oil storage tank of over 34 to 100 m in diameter and of over 10 to 20 m in height, the firefighting will be carried out using a set of large capacity monitors. It is important to estimate the discharge trajectory of water and/or fire-foam in order to avoid the thermal updrafts from the burning oil surface, and utilizing the entrainment around the peripheral zone of a tank edge. A large-scale water discharge with flow of 10 to 40 kL/min, pressure of 0.6 to 0.9 MPa was simulated using a 3D simulation model based on the Moving Particle Semi-implicit (MPS) method, and it was confirmed that the flying behavior and the water discharge trajectory correlates with the discharge flow and pressure. The water discharge which was described using this 3D simulation model gave good agreement with the recommended curved lines from the Disaster Management Committee. It is suggested that this model can estimate with precision the percentage of the delivered water mass into a tank, and can be a useful support tool for fighting tank fires.

1. Introduction

On September 2003, in Tomakomai City, Hokkaido, Japan, full-blown fire occurred in a large fuel oil storage tank with floating roof (42.7 m in diameter and 24.4 m in height) caused by the sloshing phenomenon due to the long-term oscillation of the Tokachi offshore earthquake and static electricity. The fire continued burning for about 44 h and the 26,000 kL naphtha burned out. The fire triggered partial revisions of the Fire Service Act and the Act on the Prevention of Disaster in Petroleum Industrial Complexes and Other Petroleum Facilities which has obligated to install a set of large capacity monitors to any petroleum tank of over 34 to 100 m in diameter since November 2008 [1]. The large capacity monitor can discharge fire-foam over 10 to 40 kL/min, which is ten times bigger capability than that of an aerial platform, and can deliver fire-foam into a tank located 50 to 100 m away. Also, the monitor have a proven record of extinguishing in 65 min a full-blown tank fire of 82 m in diameter and 10 m in height in the U.S. in 2001 [2]. However, Japan's petroleum tanks are generally built taller than 20 m in height, and half of the fire-foam discharged from monitor is blocked by the tank walls and drops to the ground. Moreover, the monitor cannot shift easily from its first installed position due to its large-sized discharge system. It is necessary to install the monitor in a suitable position in order to deliver the fire-foam into the tank efficiently by predicting the discharge trajectory—that is, the maximum and/or minimum range and height and also the width of the area where the water is most concentrated, called the “footprint” or “landing zone”. It is difficult to predict discharge trajectories with accuracy, because they depend not only on the discharge flow, pressure and angle (discharge conditions), but also on the wind velocity and direction (wind conditions), due to the discharged water and/or fire-foam being easily blown around by the wind. Therefore, development of support tools which can estimate the discharge characteristics taking account of the effects of wind has been demanded in order to conduct efficient firefighting of tank fires.

To predict the delivery of the fire-foam onto the burning oil surface considering the buoyancy and entrainment at the around the tank edge which are induced by the tank fire is very important for efficient firefighting. Therefore, we divided the analysis of the problem into two regions, that's are (a) discharge performance of large capacity foam monitor, and (b) the spread behavior of fire-foam on the oil tank surface. This paper deals with the water

* Corresponding author. Tel.: +81 266 73 1201; fax: +81 266 73 1230.
E-mail address: tatsuya-mi@hotmail.co.jp (T. Miyashita).

0379-7112/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.firesaf.2013.09.028

Nomenclature

A	amplitude
C_D	coefficient of air resistance [–]
C_w	reduction coefficient of air resistance [–]
d	particle diameter [m]
d_m	mean particle diameter [m]
f	frequency [Hz]
$\vec{F}$	external force [m/s^2]
g	gravitational acceleration [m/s^2]
h	height of water discharge trajectory [m]
h_0	nozzle height [m]
m	mass of particle [kg]
M	proportion (percentage) of water mass delivered into a tank [–]
n	uniformity number [–]
N_i	particle number density [–]
P	discharge pressure [MPa]
P'	dimensionless discharge pressure [–]
Q	discharge flow [kL/min]
Q'	dimensionless discharge flow [–]
r	distance between particles [m]
r_e	interaction radius [m]
R_m	mass fraction of particles [–]
S	projected sectional area [m^2]
t	time [s]
$\vec{u}$	velocity of particle [m/s]
$\vec{U}$	wind velocity [m/s]
U'	dimensionless wind velocity [–]
x	distance from nozzle [m]
θ	discharge angle [rad]
ρ	density [kg/m^3]
ρ_{air}	density of air [kg/m^3]

and/or fire-foam discharge trajectory as the primary problem, but the second item (b) including the effect due to the buoyancy is not concerned. The authors constructed previously a three-dimensional simulation model [3,4] based on the Moving Particle Semi-implicit (MPS) method [5], and verified the validity of the simulation results through comparison with medium-scale water and fire-foam discharge experiments with flow of 1.5 kL/min [6,7]. The purpose of this study is to simulate a large scale water discharge with flow of more than 10 kL/min as a case study used for the 3D simulation model, and to make clear the flying behavior and water discharge trajectory correlate with the discharge conditions. Furthermore, this paper presents a new model which can predict simply the trajectory of straight water discharge based on 3D simulation results, called the "spreadsheet model". In order to serve as a utility tool which can support the planning of disaster prevention, the discharge trajectory described using the spreadsheet model is to be evaluated by comparing it with a recommended curved line.

2. Simulation

2.1. Simulation model

The 3D simulation model of water discharge was constructed based on the MPS method, which was developed for incompressible fluid dynamics. MPS method is one of the particle methods or meshless methods, as well as Smoothed Particle Hydrodynamics (SPH) method. The MPS method has been applied to free surface flows and multi-physics problems: for example, wave breaking, sloshing, shipping water, micro flow and solid dynamics. In the MPS method, the governing equations are the equation of continuity and the Navier–Stokes equation, and particle interaction models are prepared with respect to differential operators: gradient, divergence and Laplacian. All the interactions are limited among the neighboring particles within a finite distance. Mass, momentum and energy conservation equations, which involve such differential operators, are transformed to particle interaction equations. The particles move in Lagrangian description, so that the convection terms are not calculated. Thus, numerical diffusion does not occur. This is preferable to keep the interfaces clear. Incompressibility is calculated by a semi-implicit algorithm where the pressure field is implicitly solved using the Poisson equation, while the other terms are explicitly calculated [8,9].

For analysis of the flying behavior in a straight water discharge, surface tension is calculated as force that operates among neighboring particles in the MPS method. Droplet break-up behavior is classified by the Weber number W_e, and viscosity effects on the droplet breakup are correlated with the Ohnesorge number O_n [9]. However, these calculations need to employ the high computation cost with the large size calculation domain such as water discharge of large capacity monitor. Therefore, we constructed new models – a water stream model, a water mass break-up model and a wind turbulence model – instead of the previous MPS method. This model focused on the water particle movement and interaction between them but not on the air flow in order to reduce the process of numerical simulation, and considering only fluid flow. Thus, the water particles are affected by the air resistance as external force $\vec{F}$ of Eq. (1). However, the particles in the central portion of the water mass (assembly of particles) are only slightly affected by the air resistance, because only the particles of surface portions of water mass are affected by air resistance. Therefore, we defined a reduction coefficient of air resistance C_w which is determined as Eq. (2). N_i of Eq. (3) is the particle number density, can be calculated by the position and velocity of other particles (neighboring particle) in the hemispherical interaction radius of each fluid particles, shown in Fig. 1. This model is applied the weight function (function for searching neighboring particle) of MPS method where the mass of particle m, coefficient of air resistance $C_D(=0.5)$, density of air ρ_{air}, projected sectional area S, velocity of particle $\vec{u}$, wind velocity $\vec{U}$, distance between particles r and interaction radius r_e (is 12 times as large as the diameter of particle).

$$\vec{F}=\frac{1}{2m}C_w C_D \rho_{air} S(\vec{u}-\vec{U})|\vec{u}-\vec{U}| \tag{1}$$

$$C_w = 1/N_i \quad (N_i \neq 0) \tag{2}$$

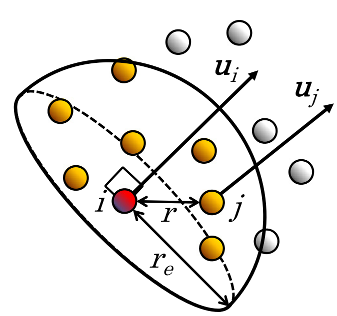

Fig. 1. Definition of a particle interaction model for a calculation of air resistance considering to the other particles.