Applied Ocean Research 68 (2017) 77–90

Contents lists available at ScienceDirect

Applied Ocean Research

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

Hydrodynamics of oil jets without and with dispersant: Experimental and numerical characterization

F. Gao[a], L. Zhao[a], M.C. Boufadel[a,*], T. King[b], B. Robinson[b], R. Conmy[c], R. Miller[d]

[a] *Center for Natural Resources Development and Protection, Department of Civil and Environmental Engineering, New Jersey Institute of Technology, Newark, NJ, United States*[1]
[b] *Bedford Institute of Oceanography, Department of Fisheries and Oceans, Dartmouth, NS, Canada*
[c] *National Risk Management Lab, United States Department of Environmental Protection, Cincinnati, OH, United States*
[d] *Department of Mechanical Engineering, Clemson University, Clemson, SC, United States*

ARTICLE INFO

Article history:
Received 27 March 2017
Received in revised form 1 August 2017
Accepted 22 August 2017

Keywords:
CFD
Experimental testing
Oil jet
Dispersant

ABSTRACT

In this paper, we present the analysis of an underwater horizontal oil jet experimental measurement and Computational Fluid Dynamics (CFD) using the Reynolds Averaged Navier Stokes (RANS) equations. Two oil subsurface releases were conducted: one with crude oil and another with crude oil premixed with dispersant at the dispersant to oil ratio (DOR) of 1:20. The jet profile was captured by a camera at moderate resolution, and the instantaneous velocity was measured by a Vectrino Profiler. The velocity components, turbulence kinetic energy, and turbulence dissipation rate from the experiment agreed well with those from the CFD simulation using the k-epsilon turbulence model. The spread angle of the jet was found to be around 21° and 24° from the experiment measurement, for oil without dispersant and oil with dispersant, respectively. The latter is close to the angle of miscible jets at 23°. The jet profile of oil with dispersant had a smaller buoyancy than that without dispersant, which is probably due to the large water entrainment for the oil with dispersant jet. The cross sections of the jet for both cases gradually became flattened with distance, as the plume turned upward.

1. Introduction

Jet and plume are commonly encountered in natural environment and engineering application, including underwater hydrothermal-vents [1], wastewater discharge from outfalls [2], oil-leakage/blowout during off-shore drilling [3], production and transportation [4]. Underwater oil jets have gained extensive attention recently following the Deepwater Horizon oil spill in 2010 with its disastrous impact on the natural environment [5,6].

The characteristics of turbulent oil jet/plume are affected by many factors, such as nozzle shape [7], Reynolds number range [8,9], temperature difference [10], to name a few.

In the absence of cross flows, vertical jet/plumes show symmetrical and self-similar characteristics at cross sections [11] and the detailed self-similarity depends on the initial condition [12,13]. In the presence of buoyancy (e.g., due to a light fluid discharging in a denser fluid), Chu and Lee [14] presented a general integral method based on conservation laws of mass and momentum. In order to integrate the partial differential equations, empirical profile shapes (i.e. Gaussian distribution) for velocity and concentration were assumed. Empirical equations for entrainment of ambient fluid into the turbulent buoyant jet were used to ensure equation closure.

The initial stage of a jet is subjected to the strong influence of shear force and instability at the edge of the discharged jet which causes the formation of ligaments of fluid. For multiphase flow, this is known as primary breakup [15,16]. The subsequent entrainment of the ligaments leads to secondary breakup that results in the formation of smaller oil droplets [17]. In the process of jet breakup, the mixing and entrainment can be further complicated at the interface during jet breakup as this process is dominated by small-scale turbulent structure [18].

Cross flow has been also a factor in vertical jets [1,19,20]; Smith and Mungal [21] showed that the entrainment process could affect the jet penetration into the cross-flow. New, et al. [22] used Particle Image Velocimetry (PIV) techniques to observe the large scale vortices structure and showed that a shear layer was stretched by cross-flow. Murphy et al. [20] conducted detailed experiments of oil released from a vertical jet in cross flow. They also considered

---

[*] Corresponding author at: Department of Civil and Environmental Engineering, New Jersey Institute of Technology, Newark NJ 07102, United States.
E-mail address: boufadel@gmail.com (M.C. Boufadel).
[1] http://nrdp.njit.edu.

http://dx.doi.org/10.1016/j.apor.2017.08.013
0141-1187/© 2017 Elsevier Ltd. All rights reserved.

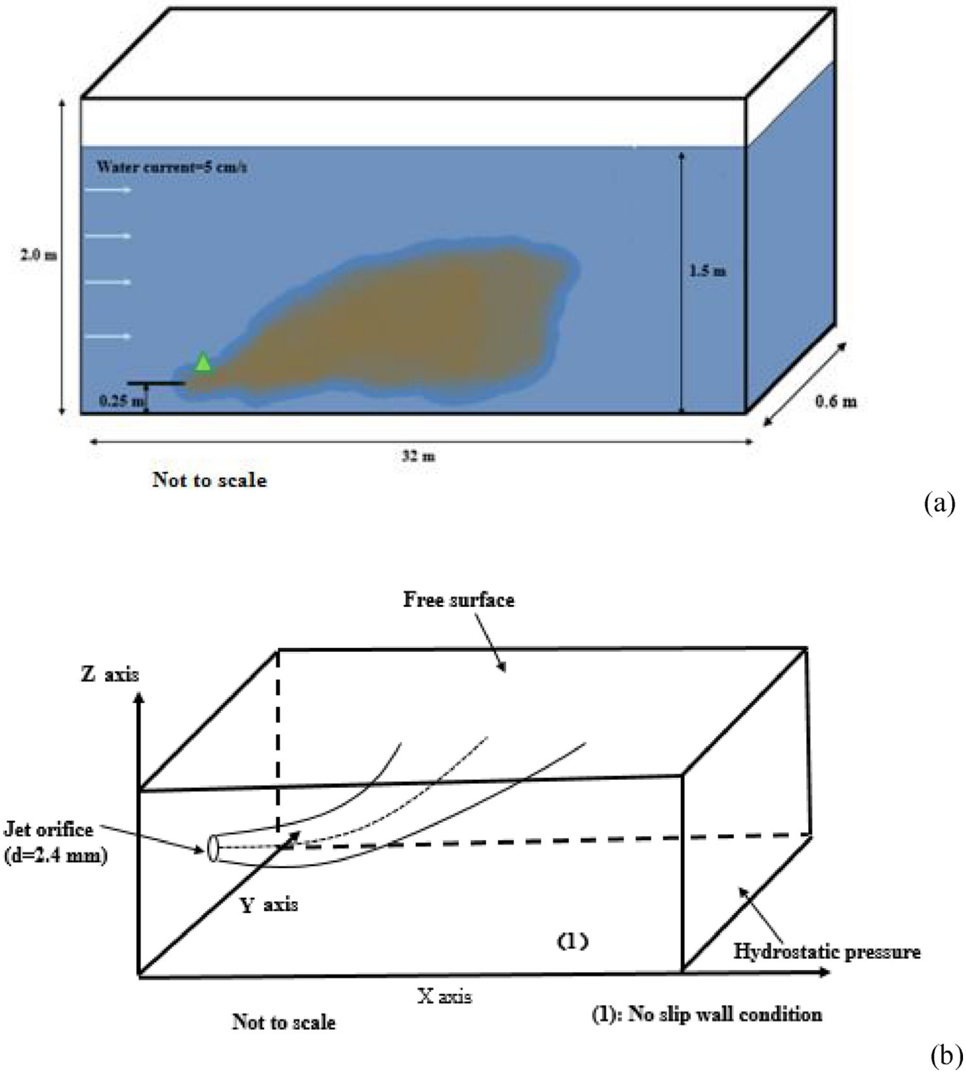

Fig. 1. A schematic describing the release of the horizontal oil plume (a) the dimensions of Bedford Institute of Oceanography Tank (with the green triangle denotes the position of the Vectrino Profiler, which was as (x = 0.5 m, y = 0.3 m, and z = 0.304 m) (b) the boundary condition implemented in the present study. (For interpretation of the references to colour in this figure legend, the reader is referred to the web version of this article.)

situations where dispersant was premixed with the oil. They also investigated the migration of oil droplets, and their entrapment in countercurrent vortices. Based on these observations and theoretical arguments, they developed trapping functions to explain the behavior.

The plume trajectory of miscible plumes is commonly modeled by a Lagrangian approach [23], which predicts the average hydrodynamic behavior along the centerline of the plume. However, the approach cannot reproduce the separation of droplets from horizontal plumes due to individual buoyancy, as noted by Zhao, et al. [24]. Computational Fluid Dynamics (CFD) seems to be a good alternative. Within the CFD approach, the Direct Numerical Simulation (DNS) solves the Navier-Stokes equations and provides the most accurate solution to the flow field. DNS is thus able to provide turbulence quantities at the interface of two fluids along with information on detailed entrainment processes [25]. However, DNS is computationally expensive and thus only limited to low to moderate (orifice-based) Reynolds number flow (e.g., Reynold <5000, typically transition between laminar and turbulent flow) [12,26]. Reynolds Averaged Navier-Stokes Equation (RANS) model [27] have been successfully used to capture the overall dynamics even at very large Reynolds numbers [28]. However, they cannot predict small-scale turbulence structure [29–31]. Large Eddy Simulation (LES) emerged as a compromise between DNS and RANS; it solves the large-scale turbulent flow and leaves the small scale to sub-grid modeling. However, it remains relatively computationally expensive, and the sensitivity of the far-flow field to the near orifice conditions is still unknown along with the effect of subgrid scale (SGS) modeling on flow characterization [32]. For these reasons, we used RANS in this work to investigate the engineering properties of horizontal underwater oil jets.

Numerous experimental works reported the measurement of the water velocity from jets. Kotsovinos [33] found that the axial velocity from a vertical plume becomes stable and follows a Gaussian distribution across the diameter at a distance of more than six diameters from the orifice and persists for a while before it deteriorates at larger than 80 diameters from the orifice. Mih [34] developed a jet radial velocity expression based on Gaussian axial velocity distribution in axisymmetric flow. Falcone and Cataldo [35] measured the mean radial and turbulent radial velocity profiles in a circular jet at up to 40 jet diameters downstream of the jet exit using an Laser Doppler Anemometry (LDA)and established a relationship between mean radial velocity and jet entrainment. In general, most velocity measurements were taken within 80 diameters of