Marine Pollution Bulletin 95 (2015) 200–206

Contents lists available at ScienceDirect

Marine Pollution Bulletin

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

Experimental and numerical investigations on reliability of air barrier on oil containment in flowing water

Jinshu Lu *, Zhenfeng Xu, Song Xu, Sensen Xie, Haoxiao Wu, Zhenbo Yang, Xueqiang Liu

School of Maritime, Zhejiang Ocean University, Zhoushan 316002, China

ARTICLE INFO

Article history:
Available online 25 April 2015

Keywords:
Air barrier
Oil spilling
Oil containment boom
Model experiment
Numerical simulation

ABSTRACT

Air barriers have been recently developed and employed as a new type of oil containment boom. This paper presents systematic investigations on the reliability of air barriers on oil containments with the involvement of flowing water, which represents the commonly-seen shearing current in reality, by using both laboratory experiments and numerical simulations. Both the numerical and experimental investigations are carried out in a model scale. In the investigations, a submerged pipe with apertures is installed near the bottom of a tank to generate the air bubbles forming the air curtain; and, the shearing water flow is introduced by a narrow inlet near the mean free surface. The effects of the aperture configurations (including the size and the spacing of the aperture) and the location of the pipe on the effectiveness of the air barrier on preventing oil spreading are discussed in details with consideration of different air discharges and velocities of the flowing water. The research outcome provides a foundation for evaluating and/or improve the reliability of a air barrier on preventing spilled oil from further spreading.

1. Introduction

Air barriers or (Pneumatic barriers) generate bubble plumes, which significantly modify the flow field and typically generate significant vortex around them. It eventually generates a backward surface flow which suppresses the existing environmental surface flow. As a result, the oil spreading dominated by the convection behavior can be prevented and the spilled oil is expected to be contained in a controlled area. This initiates the development of a new type of oil containment boom. The air barriers are usually installed fully submerged. Therefore, compared to the traditional oil booms installed on the water surface, the oil boom based employing the air barrier is less affected by the environmental wind/wave conditions and generally does not influence the ship navigation during the operations (Milne, 1970). However, the real on-site applications of this type of oil booms has indicated that the reliability of the air barrier on preventing spilled oil from further spreading is frequently not as high as expected due to a lack of well-established theories to guide the design and the installation. This limits wider applications of this technology.

So far, the evaluation of the reliability of the air barrier on oil containments largely relies on the theoretical hydrodynamic studies on the effects of the air barrier on the surrounding flow field. In the earlier stage, it has been reported that the air plume generated by the air bubbles in still water is closely related to the air discharge (Taylor, 1955; Evans, 1955; Hensen, 1955; Laurie, 1955; Hussain and Siegel, 1976) and the configuration of the air aperture, e.g. the size and the spacing arrangement, plays a less important role (Bulson, 1961; Baines and Hamilton, 1959; Ince, 1964; Abraham and Burgh, 1964; Basco, 1971). By analyzing the flow generated by the air plume and its spatial variation (e.g. the decay in vertical direction), researchers (e.g. Jones, 1972) initiated and developed the oil containment boom adopting the air barrier. However, the above-mentioned hydrodynamic studies associated with the air barriers were not designed for the purpose of exploring the behavior of the spilled oil subjected to an air plume without considering the distinguishing issues on the oil spreading. One typical example is that the oil film may be splashed, transports downward from the water surface following the downward water flow caused by the circulation/vortex. Eventually, a portion of spilled oil may escape from the control area following the uplift part of the circulation/vortex. This phenomenon is often observed when the air discharge is large (e.g. Lau and Engel, 1980; Lo, 1996, 1997). Although solutions to avoid such problem may be found based on field observations (e.g. Fannelop, 1983; McClimans et al., 2012; Eidnes et al., 2013), they have not been studied quantitatively, leading to great uncertainty in the real applications. Another big gap is that a shearing current (a free surface flow), water waves and winds often observed in the real

* Corresponding author. Tel./fax: +86 580 2554315.
E-mail address: ljs_ljs@zjou.edu.cn (J. Lu).

http://dx.doi.org/10.1016/j.marpolbul.2015.04.020
0025-326X/

application but has not been systematically investigated in the theoretically researches. Qualitative observations have revealed that the effectiveness of the air barriers on the oil containment boom is significantly affected by the wind and waves (Basco, 1971; Fang and Johnston, 2001a, b) as well as the current (Basco, 1971; Lau and Engel, 1980). However, systematically quantitative researches addressing such issues are rarely found in the literatures.

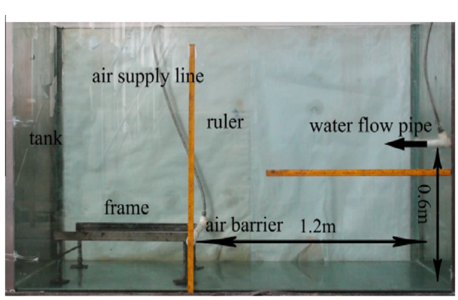
In this paper, systematic experimental and numerical studies on the reliability/effectiveness of air barriers on oil containment boom with the consideration of the effect due to the current will be presented. Considering the fact that the current may be represented by a shearing flowing water in most area requiring the installation of the oil boom, it is generated through a narrow inlet on the still free surface in this study. A submerged pipe with air apertures is installed near the bottom of the water tank to generate the air bubbles and the air plume. To quantitatively assess the reliability or the effectiveness of the air barrier on the oil containment, the horizontal distance from the location of the air barrier to the forefront of the spreading oil film will be used here. Larger distance means better effectiveness or higher reliability. The effects of the configuration of the air aperture and the location of the pipe on the effectiveness of air barrier subjected to different air discharges and velocities of the flowing water will be the focus of this paper.

2. Experimental configuration and numerical models

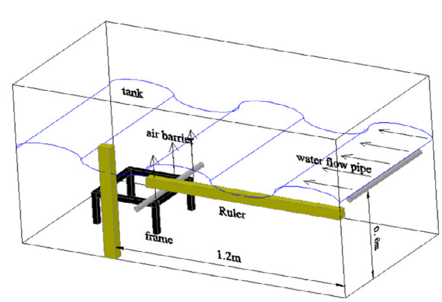
The experiment is carried out at the Zhejiang Ocean University, China. A sketch and photograph of the tank are illustrated in Fig. 1. The length, width and depth of the tank are 2.0 m, 1.0 m and 1.0 m, respectively. The pipe generating air bubbles is installed in an adjustable frame with distance from the left end (downstream) of the tank of 0.8 m. The submerged depth (h) can be adjusted to study the effect of the immersion depth on the effectiveness of the air barriers on the oil containment. Apertures are evenly distributed along the pipe. In order to explore the effects of the aperture size (D) and the separation interval (spacing), different aperture sizes and separation intervals (d) will be considered in the experiment. As observed by Eidnes et al. (2013), the air barrier developed from multiple air pipes may perform better than a single pipe; the cases with dual pipes are also investigated for comparison. The environmental water flow is introduced by using a narrow inlet on the mean water depth, i.e. 0.6 m from the bottom of the tank. It is achieved by using a horizontally positioned pipe installed on the right end of the tank. It generates a water jet near the free surface, generating a shearing flow. From the laboratory observation and the numerical simulations (presented below), the shearing flow generated by the inlet pipe shows two typical features: (1) the flow velocity decays dramatically from the water surface to the bottom of the tank; (2) an vortex appears from the free surface to the bottom of the tank. Although such vortex may not be seen in reality, it magnitude is significantly smaller compared to both the tidal current on/near the surface and the flow introduced by the air barriers. As a result, the present way to introduce the shearing flow is acceptable for a deep water scenario where tidal current or wind-driven current is only significant in thin area near the free surface. An outlet pipe is installed in the downstream (left end) of the tank to maintain the mean water depth. A Reynolds similarity law is adopted in this research to select the type of the oil and the inlet flow velocity. Based on this, the rapeseed oil with the viscosity of 97 cSt and inlet flow velocity V_m = 0.1 m/s are considered in the experiment. This configuration corresponds to a real scenario in which spilled heavy oil with the viscosity of 3500 cSt is subjected to a typical surface current of 2 Kn. During the experiment, 10 L oil is split into the upstream near the inlet pipe in a short time at a level. Initially, the surface shearing flow generated by the inlet jet drives the spreading of the oil toward the outlet of the tank. However, the bar barrier leads to vortices on both sides of the air pipe. Near the free surface upstream, the fluid moves toward the inlet. This suppresses the inflow and eventually leads to a quasi-steady state, as illustrated in Fig. 2, in which the forefront of the oil film almost remains at the same position where the flow velocity is approximately zero (referred to as zero-velocity surface). Be noted that during this stage, there will be an insignificant spreading of the oil film due to the diffusion. The horizontal distance from the location of the air barrier to the forefront of the spilled oil film is defined as the effective containment distance (S), reflecting the effectiveness or the reliability of the air barrier on the oil containment. It should be noted that when the oil film reaches the downstream of the air barrier, the value of S becomes negative implying the failure of the air barrier. Two high-speed camcorders simultaneously record the entire process.

As indicated above, different aperture sizes, separation intervals, submerged depth of the pipe will be used. For the purpose of comparison, both a dual-pipe and a single-pipe system to generate the air bubbles will also be considered. A list of test conditions have been designed and summarized in Table 1. All cases have been repeatedly carried out at least three times on different days and by different persons to minimize the possibility of measurement error. The average values of the measured data are presented in this paper.

In the numerical simulation, the commercial software FLUENT is used. The same computational domain as shown in Fig. 1 is used. A multiple-phase Navier–Stokes solver, in which air, water and oil are treated as three different phases using the volume of fluid (VOF) technology, is solved using the finite volume method. The k–ε model is employed to model the turbulence. On the top of the computational domain, i.e. open to the air, a pressure out-let is used. A wall boundary condition is used on the bottom of the tank. A velocity inlet boundary and a flow-split outlet boundary



(a)



(b)

Fig. 1. Sketch of the tank (a) and experimental facility (b).