A Numerical Study on the Wind Effect on the Oil Containment by Boom

Kwangu Kang*, Moonjin Lee, and Jung-Yeul Jung
Maritime and Ocean Engineering Research Institute
KIOST
Daejeon, Republic of Korea
kgkang@kiost.ac

***Abstract*— We numerically investigate the effect of wind on the oil containment by boom. The numerical simulations are carried out by using commercial computational fluid dynamics software, FLUENT with Volume of Fluid method. A boom containment of high viscosity oil in real scale is simulated, where the lengths of boom and simulated domain size are practical. It is found that the effect of wind becomes significant above the threshold velocity of 10 m/s. At wind velocity of 35 m/s, the leakage rate is twice larger than the leakage rate at 0 m/s.**

Keywords— Oil Spill, Oil Containment by Boom, Wind Effect, Numerical Simulation, FLUENT

I. INTRODUCTION

Oil spills can do significant harm to the marine ecological environment, so it is very important to prevent them from being transported to open area. An oil slick is mainly transported by the combined effects of winds, waves, and currents. Oil booms have been widely deployed to prevent the spread of oil spill in marine or inland waters. In spite of their popular usages, oil booms have suffered many failures. Generally, the oil is leaked underneath the barrier by following seven basic mechanisms: splash-over, submergence, planing failure, structural failure, entrainment failure, drainage failure and critical accumulation. Many previous studies had identified that drainage failure, droplet entrainment failure, and critical accumulation failure are most dominant failure mechanisms. [1, 2]

A lot of numerical studies have been carried out on the problem of oil containment. Goodman et al.[3] successfully modeled three different failure mechanisms such as drainage failure, droplet entrainment, and critical accumulation. Their result implies that numerical simulation can depict the major boom failure phenomena and more importantly the loss rate. Brown et al.[4] numerically investigated the effects of current velocity on the containment by the oil barrier and their results agreed with the experimental data except deviation from ignoring the turbulent effect. Chebbi [5] provided an analytic treatment for oil circulation by using both boundary layer model and constant drag coefficient model. His model is simply compared to full CFD model and useful to estimate the minimum length of draft required for the prevention of drainage failure. Amini and Schleiss [6] modeled single and double barriers containing oil over water surface by using commercial CFD simulation tool, FLUENT with VOF method. The numerical results were in good agreement with LSPIV and UVP experimental measurement. They studied the influence of the distance between two rigid barrier and slick shape contained by the barrier.

Most of previous researches have been studied in calm water in laboratory scale. However, oil boom is deployed under open-sea conditions, where waves and winds exist. It is generally anticipated that the performance of booms is reduced in open-sea conditions. Therefore it is very crucial to understand the effect of variation of hydrodynamic conditions on the effectiveness and failure of booms. A few studies have taken the effect of winds and waves. Fang and Johnston [7~9] developed a local two-phase nonlinear hydrodynamic numerical model to simulate oil containment by a fixed boom and investigated effects of current, wave, and wind condition. Violeau et al.[10] applied the Smoothed Particle Hydrodynamics (SPH) Lagrangian numerical method to modeling of oil containment by boom and quantitatively present relation between the oil boom failure velocity and wave height. Xing et al.[11] recently investigate the relation between loss rate and wave steepness by using commercial CFD software, FLUENT.

However, few studies investigate the relation between wind and loss rate of oil containment. Hence, the present investigation focuses on the oil containment by rigid floating boom under various wind conditions, based on the Navier-Stokes equations for viscous, incompressible fluid and Volume of Fluid (VOF) method, utilizing the commercial CFD software FLUENT. The size of the boom is same with the boom in practical use in Korea, which is larger than the sizes of booms used in previous research based on lab scale experiment. In other words, the numerical platform established here is close to the open sea condition.

978-0-933957-40-4 ©2013 MTS

II. MATHMATICAL MODEL

A. Governing equations

For time-dependent turbulent flow in real ocean conditions, we solved continuity equations, Navier-Stokes equations and realizable k-ε equations. VOF method is adopted to compute the multi-phase. In the simulation, it is assumed that the fluid is incompressible[12].

Continuity equation for the mass conservation is follows.

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = S_m \tag{1}$$

Incompressible viscous fluid Navier-Stokes equations is

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \nabla \cdot \left(\bar{\bar{\tau}}\right) + \rho \vec{g} \tag{2}$$

where p is the static pressure, $\rho \vec{g}$ is the gravitational body force. The stress tensor is given by

$$\bar{\bar{\tau}} = \mu \left[\left(\nabla \cdot \vec{v} + \nabla \cdot \vec{v}^T \right) - \frac{2}{3} \nabla \cdot \vec{v} I \right] \tag{3}$$

where, $\mu$is the molecular viscosity, I is the unit vector, and the second term on the right hand side is the effect of volume dilation. To calculate the turbulent viscosity, we adopt realizable k-ε model[13], which is found to be substantially better than that of standard k-ε model as follows,

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon - Y_M \tag{4}$$

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_j}(\rho \varepsilon u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \rho C_1 S \varepsilon - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{v \varepsilon}} + C_{1\varepsilon} \frac{\varepsilon}{k} C_{3\varepsilon} G_b \tag{5}$$

In these equations, G_k and G_b represent the generation of turbulent kinetic energy created by average velocity gradient and buoyancy, respectively. Y_M stands for the contribution of the fluctuating dilation in compressible turbulent kinetic energy to the overall dissipation rate. C_2 and $C_{1\varepsilon}$ are constants. σ_k and σ_ε are the turbulent Prandtl numbers for k and ε, respectively. Like other k-εmodel, the eddy viscosity is computed from

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \tag{6}$$

The main difference between the realizable k-εmodel and the standard k-ε model is that C_μ is no longer constant. (Shih and Liou, 1995)

In the VOF model, the tracking of the interfaces between phases is accomplished by the solution of a continuity equation for the volume fraction of two phases, oil and water phases. For the qth phases, this equation has the following form with the assumption of 2-dimension, incompressible and no mass transfer between phases.

$$\frac{\partial \alpha_q}{\partial t} + \frac{\partial (\alpha_q)}{\partial x} + \frac{\partial (\alpha_q)}{\partial y} = 0 \tag{7}$$

q = water and oil

The air volume fraction will be computed based on the following constraint, i.e. the volume fraction equation for air will not be solved.

$$\sum_{q=1}^{3} \alpha_q = 1 \tag{8}$$

α_q is volume ratio of the q-phase to the unit.

B. Simulation domain

In this study, a containment of bunker C in real scale is simulated. The schematics of simulations are shown in Fig. 1. The size of domain is two-dimensional 100 m × 10 m channel with a water level of 5 m. A rigid barrier with 0.3 m floating part and 0.4 m submerged part is located in the center as shown in Fig. 1, which is a practical boom size. The bunker C has a density of 993 kg/m^3, and a viscosity of 30 kg/m·s. The interfacial tension of bunker C and water is 0.04 N/m. We generated 108,070 rectangular meshes and the meshes near boom area are refined.

Moloney [14] suggested that the distance between the upstream boundary and the boom should exceed 15 times the boom draft to ignore the pressure changes over this distance. Also Fang and Johnston [8] suggested that the water depth in the numerical simulation needs to be > 5 times the draft of boom so that the flow around the boom is not influenced or distorted by the bottom boundary. The geometries of our two models satisfy Monoley's distance criterion and Fang and Johnston's water depth criterion.

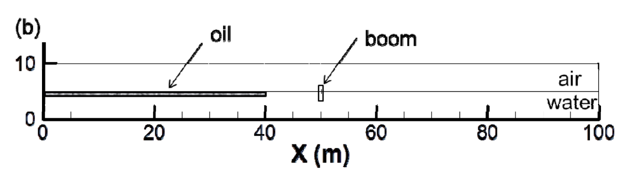

Fig. 1. Schematics of numerical simulation of Bunker C containment by boom

TABLE I. SUMMARY OF SIMULATED WIND AND WATER VELOCITIES

Water velocity	Wind velocity
0.3 m/s	0 m/s
0.4 m/s	0 m/s
0.5 m/s	-35 m/s -5 m/s 0 m/s 0.5 m/s 5 m/s 10 m/s 20 m/s 35 m/s
0.6 m/s	0 m/s

C. Boundary conditions

For the simulation with bunker C oil, following boundary conditions were applied.

- Top and bottom ace: Neumann boundary condition
- Inlet : uniform velocity condition
- Outlet: volume balance condition.

We assume that velocity profile at top and bottom boundary face is fully developed and applied Neumann boundary condition at top and bottom faces to depict the deeper water depth than numerical domain and opened air system

III. RESULTS AND DISCUSSION

To investigate the effect of winds on the containment of spilt bunker C oil, we varied the sea water velocity and wind velocity and list in Table. 1. Figure 2 shows the contour of fraction of bunker C. In initial condition, the bunker C is assumed as thin film with thickness of 2 cm as shown in Fig. 1. However as a film approaches the barrier, the film is broken into large droplets as shown in Fig. 2. We attribute the formation of large droplet from thin film to extremely high viscosity of bunker C, since the viscosity of liquid represents the level of cohesive force between molecules. And the mechanism of containment failure is critical accumulation, that is common in high viscosity liquid (> 100,000 cSt).

Figure 3 shows the accumulated leaked oil volume under through the barrier for four different water velocities. As the water velocity increases, the velocity of bunker C oil should increase accordingly. We note that the arrival times of bunker C to barrier are larger than the times calculated by simple velocity distance relation. For example, in the case of 0.5 m/s flow velocity, the expected arrival time of bunker C to barrier is 20 seconds that is calculated distance between initial bunker C position and barrier and flow velocity, i.e. 10 m over 0.5 m/s. This implies that the formation of droplet retards the movement of oil. The retarded velocity of oil is approximately 1.2 times higher than water velocity.

Figure 4 represents the accumulated leaked oil volume for various wind velocities ranges from -5 m/s to 35 m/s. The 35 m/s is typical wind velocity of very strong typhoon. As shown in Fig. 4, the leaked volume does not show much difference below 10 m/s wind velocity. However, the leaked volumes start to increase at 10 m/s wind velocity and the 35 m/s wind velocity case shows significant increases in leaked volume. The leaked volume at 35 m/s wind velocity is about twice larger than the leaked volume at wind velocity of 0 m/s. Moreover, at -35 m/s wind velocity, the oil slick could not arrive at the barrier until $t = 50$ s. Above discuss indicates that the wind effect on the movement of oil slick becomes important at higher velocity than threshold wind velocity of ~10 m/s.

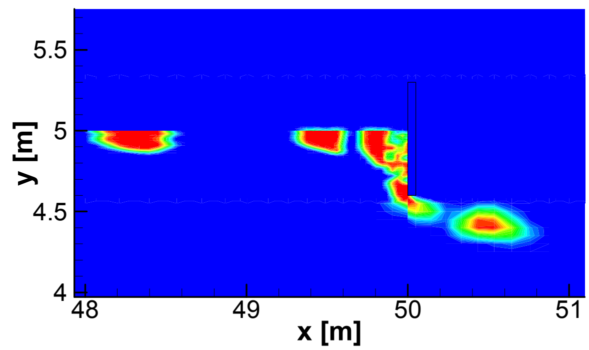

Fig. 2. Distribution of bunker C around rigid boom at flow velocity of 0.5 m/s and wind velocity of 0 m/s at $t = 30$ s

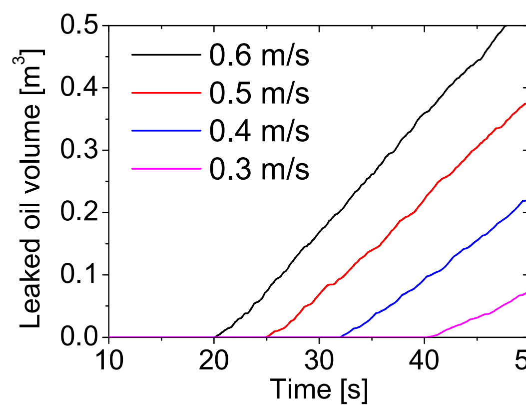

Fig. 3. Leaked oil volume as a function of time at 4 different water velocities

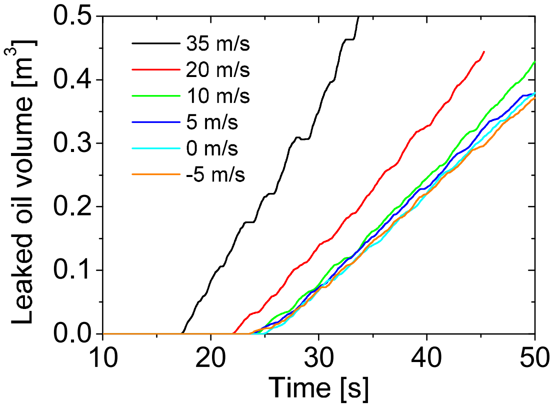

Fig. 4. Leaked oil volume as a function of time at different wind velocities

IV. CONCLUSIONS

In this paper, we investigate the effect of wind velocity on the failure of oil boom. The simulation is performed under the real open-sea condition, in other words, the length scales are much large than laboratory scale. Below the threshold wind velocity of ~10 m/s, the effect of wind on the movement of oil slick is negligible. However, above the threshold wind velocity, the shear force from wind velocity enhances the movement significantly.

ACKNOWLEDGMENT

This work is carried out as a part of the study on “Development of Oil Spill Response Supporting System based on 3-D Oil Dispersion Modelling” and “Development of HNS spill response information supporting technologies based on HNS dispersion model” under the support by the Korea Institute of Ocean Science and Technology.

REFERENCES

[1] G. A. L. Delvigne, “Barrier failure by critical accumulation of viscous oil”, presented at the Oil Spill Conf., Oil Spill Conf., 1989.

[2] C. M. Lee, K. H. Kang, and N. S. Cho, “Trapping of keaked oil with tandem oil fences with Lagrangian analysis of oil droplet motion”, *Journal of Offshore Mechanics and Arctic Engineering*, vol 120, No. 1, pp 50–55, 2 1998.

[3] R. H. Goodman, H. M. Brown, C.-F. An, and R. D. Rowe, “Dynamic modelling of oil boom failure using computational fluid dynamics”, *Spill Science & Technology Bulletin*, vol 3, No. 4, pp 213–216, 1996.

[4] H. M. Brown, R. H. Goodman, C.-F. An, and J. Bittner, “Boom failure mechanisms: Comparison of channel experiments with computer modelling results”, *Spill Science & Technology Bulletin*, vol 3, No. 4, pp 217–220, 1996.

[5] R. Chebbi, “Profile of oil spill confined with floating boom”, *Chemical Engineering Science*, vol 64, No. 3, pp 467–473, 2 2009.

[6] A. Amini, and A. J. Schleiss, “Numerical modeling of oil-water multiphase flow contained by an oil spill barrier”, *Engineering Applications of Computational Fluid Mechanics*, vol 3, pp 207–219, 1 2009.

[7] F. Fang, and A. J. Johnston, “Oil containment by boom in waves and wind. I: Numerical model”, *Journal of Waterway, Port, Coastal, and Ocean Engineering*, vol 127, No. 4, pp 222–227, 8 2001.

[8] F. Fang, and A. J. Johnston, “Oil containment by boom in waves and wind. II: Waves”, *Journal of Waterway, Port, Coastal, and Ocean Engineering*, vol 127, No. 4, pp 228–233, 8 2001.

[9] F. Fang, and A. J. Johnston, “Oil containment by boom in waves and wind. III: Containment failure”, *Journal of Waterway, Port, Coastal, and Ocean Engineering*, vol 127, No. 4, pp 234–239, 8 2001.

[10] D. Violeau, C. Buvat, K. Abed-Meraim, and E. de Nanteuil, “Numerical modelling of boom and oil spill with SPH”, *Coastal Engineering*, vol 54, No. 12, pp 895–913, 12 2007.

[11] F. Xing, W. Wanqing, and W. Wenfeng, “Numerical Simulation Technology of Oil Containment by Boom”, *Procedia Environmental Sciences*, vol 8, No. 0, pp 40–47, 2011.

[12] Fluent Inc., *FLUENT 6.3 User’s Guide*. 2006.

[13] T.-H. Shih, and W. W. Liou, “A new k-□ eddy viscosity model for high Reynolds number turbulent flows”, *Computers & Fluids*, vol 24, No. 3, pp 227–238, 1995.

[14] B. L. J. Moloney, “Oil spill containment using booms”, James Cook University, 1996