Proceedings of the 7th International Conference on Asian and Pacific Coasts (APAC 2013) Bali, Indonesia, September 24-26, 2013

A MODEL TO SIMULATE THE SPREADING OF OIL AND GAS IN UNDERWATER OIL SPILLS

X. Niu [1], X. Li [1] and X. Yu [1]

ABSTRACT: The rapid growth of offshore oil production and undersea oil delivery pipelines increases the risk of underwater oil spill. In this study, a model based on the Lagrangian particle tracking method is developed to simulate the spreading of oil and gas in an underwater oil spill, which is helpful to estimate the environmental impact and to find effective measures for preventing the spreading of oil. The oil droplets and gas bubbles released from the leakage point are modeled by a large number of representative particles, which are divided into several groups to simulate different components of oil and gas leaked from the underwater blowout. The movement of each particle in one time step includes two components, a mean movement and a random walk. The mean movement is computed by combining the effect of surrounding marine hydrodynamic, the buoyant jet flow near the leakage point and the rise velocity of representative oil droplets or gas bubbles.The random walk method is used to simulate the turbulent diffusion. The compressibility and dissolution of gas are also considered, which play an important role in deepwater. Comparing with the previous models for underwater oil spill based on the integral Lagrangian control volume method, the present model is more flexible in simulating the crude oil which has complex components. The model is validated by several experiment cases and successfully applied to simulate the DeepSpill field expreiment, and good agreement between the calculation and the observation is obtained. The fractionation of different gas bubbles or oil droplets is considered and significant differences in the underwater distribution of oil droplets and gas bubbles with different sizes are clearly shown in the simulated results.

Keywords: Oil spill, underwater process, particle tracking method, hybrid model.

INTRODUCTION

The rapid growth of offshore oil production and undersea oil delivery pipelines increases the risk of underwater oil spill, which usually causes extensive damage to marine environment and wildlife habitats and also harms the fishing and tourism industries. Study on the underwater oil spill is helpful to estimate the damage to marine environment and to find an effective measure for preventing the spreading of oil.

Underwater oil spill usually behaves as a multiphase buoyant jet of oil droplets and gas bubbles near the leakage point and is dominated by the exit momentum. When oil droplets and gas bubbles move far away from the leakage point, they are dominated by the advection and diffusion of the ambient current. Therefore, the whole underwater oil spill process is often artificially divided into two stages, the buoyant jet stage and the advection diffusion stage. In the earlier studies, researchers focused on the dynamic process of the buoyant jet caused by oil spill and a number of achievements were obtained. Yapa and Zheng (1997) and Zheng and Yapa (1998) had developed a model based on the integral Lagrangian control element method to simulate the buoyant jet in an underwater oil spill. Thereafter this model has been improved in many aspects and successfully applied to simulate many problems (e.g. Chen and Yapa 2004; Johansen 2000; Yapa et al. 2010). In the subsequent models, the advection diffusion stage was involved to complete the underwater oil spill processes by introducing a large number of Lagrangian particles at the end of the buoyant jet stage. In previous studies, the terminal level of buoyant jet dynamics is usually adopted as a transition point from the buoyant jet stage to the advection diffusion stage, and several types of criteria mentioned by Dasanayaka and Yapa (2009) for the terminal level of buoyant jet dynamics were usually used.

One of the important improvements to the original model of Yapa and Zheng (1997) is to involve the separation of gas bubbles from the main buoyant jet in ambient cross-flow. Several studies had observed that gas bubbles and large oil droplets could separate from the main buoyant jet in certain ambient cross-flow (Socolofsky et al. 1999). The separation occurs when horizontal ambient current drives the entrained fluid away from the dispersed phase (Socolofsky and Adams 2002). As another point of view, the separation of dispersed phase from the main buoyant jet occurs when

[1] Department of hydraulic Engineering, Tsinghua University, Beijing, 100084, CHINA

the dispersed phases (gas bubbles or big oil droplets) move much faster in vertical direction than the entrained fluid and then run away from the main buoyant jet. Socolofsky and Adams (2002) and Chen and Yapa (2004) had taken the separation of gas bubbles from the main buoyant jet into consideration. In their studies, they focus on the dynamics of the buoyant jet, so the separation of gas bubbles mainly results in reduced buoyancy of the main buoyant jet. In those improved models the separation of gas bubbles starts at a separation height given by empirical formulae. However, the separation of large oil droplets from main buoyant jet was ignored in the previous studies. It is because that the separation of oil droplets may not affect the main buoyant jet as notable as the separation of gas bubbles, owning to the smaller density difference. Gas bubbles and oil droplets with different sizes have different rise velocity. And they separate from the main buoyant jet in a cross flow asynchronously, which is called fractionation as mentioned by Socolofsky and Adams (2002). Big bubbles and droplets will separate from the main buoyant jet rapidly and move individually to water surface, while some small bubbles and droplets may remain moving within the main buoyant jet for a long time. The fractionation affects the trajectories of gas bubbles and oil droplets, which finally affects the time, location and distribution of the oil film emerging on water surface after the underwater oil spill happened.

In the previous studies, researchers mainly concerned the dynamics of the buoyant jet in oil spill. So it is acceptable to ignore the separation of oil droplets and to directly introduce a large number of particles uniformly distributed at the end of the buoyant jet stage without considering the fractionation of different gas bubbles and oil droplets. However, to simulate the evolution and fate of the oil spill from an underwater blowout is the final destination, we try to find out the time, location and distribution of the oil film emerging on water surface from where it starts.

In this study, we focus on the underwater process of a blowout and a model based on the Lagrangian particle tracking method is developed to track oil and gas leaked out. The oil droplets and gas bubbles released from the leakage point are modeled by a large number of particles. The particles are initially divided into several groups to represent bubbles or droplets of different sizes, so as to describe their behaviors more accurately. Owing to the simplicity of particle tracking method, it has been widely used in many research fields and has been well developed to simulate water surface oil spill including the complex physico-chemical processes (e.g. Guo and Wang 2009; Korotenko et al. 2004; Nagheeby and Kolahdoozan 2010). To develop a model based on Lagrangian particle tracking method for the underwater process will provide a direct way to link the underwater process with the surface spreading process in modeling an underwater oil spill.

The detailed information of the present model is introduced in Section 2. In Section 3 the model is applied to simulate the laboratory experiment carried out by Socolofsky et al. (1999). In Section 4 the model is applied to DeepSpill, a field experiment on oil and gas blowout in deep water, in order to validate the applicability of the present model in deepwater condition.

MATHEMATICAL MODEL

Framework of the Present Model

Particles are introduced into the computational domain at the leakage point and tracked by a random walk model. To consider different behavior of bubbles and droplets of different sizes, particles are divided into *N* groups according to the original size distribution of bubbles and droplets. The terminal rise velocity of particles in each group is the same. And the change in size and density of gas bubbles owning to the compressibility and dissolution are considered, which should not be ignored especially in case of deepwater. So the terminal rise velocity of each group is also changing according to the position of particles.

The movement of each particle in one time step includes two components, a mean movement and a random walk, which is calculated by the following equation.

$$\mathbf{x}_p\left(t+\Delta t\right)=\mathbf{x}_p\left(t\right)+\mathbf{u}\left(\mathbf{x},t\right)\Delta t+\mathbf{r} \tag{1}$$

In which $\mathbf{x}_p$ is the position vector of the particle, Δt is the time step; $\mathbf{u}\left(\mathbf{x},t\right)$ is the mean velocity of the particle, which depends on its terminal rise velocity, the buoyant jet hydrodynamics and the ambient current; $\mathbf{r}$ is the random walk step for one time step, which accounts for the dispersive phenomena caused by turbulence. The random walk step component in each direction is assumed as $\mathbf{r} \equiv r_i = \sqrt{2D_i\Delta t}r_0$, following Korotenko et al. (2004), in which the subscribe *i* indicates *x*, *y*, *z* directions; r_0 is a random number drawn from a normal distribution with mean 0 and standard deviation 1; D_i is the diffusion coefficient.

The key issue in the simple model is to estimate the mean velocity and the random walk step of each particle. Since gas bubbles and oil droplets may separate from the main buoyant jet in a cross-flow, the movement of each particle may have two states, i) within the main buoyant jet and ii) separated from the main buoyant jet, as shown in Figure 1.

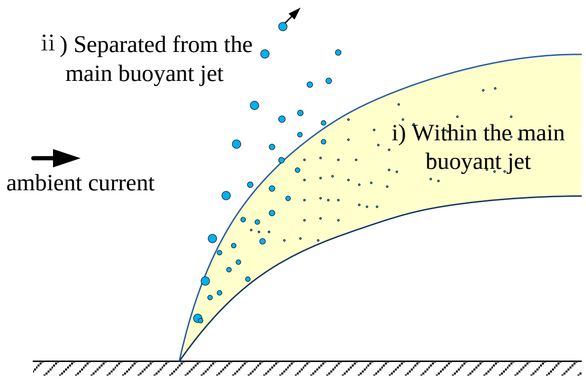


Fig.1 Sketch of the physical problem

i) Within the main buoyant jet

The mean velocity of the particle is assumed to be the same of cross-section avarage velocity of buoyant jet if the particle is within the main buoyant jet. The velocity of the main buoyant jet is calculated by an integral Lagrangian control volume method following Chen and Yapa (2004), considering the decrease in buoyant force owing to the separation of oil droplets as well as gas bubbles. And if the particle is within the main buoyant jet, the diffussion coefficient is estimated by $D_i \propto |\mathbf{V}| B$, referring to Rodi (1982), in which $|\mathbf{V}|$ is the magnitude of avarage velocity of the buoyant jet and B is the radius of the cross-section of the buoyant jet.

ii) Separated from the main buoyant jet

After separated from the main buoyant jet, the mean velocity of the particle is assumed to be the velocity of the ambient current plus the terminal rise velocity.

$$\mathbf{u}(\mathbf{x},t) = \mathbf{U}_a(\mathbf{x},t) + w_b\mathbf{j} \tag{2}$$

In which w_b is the terminal rise velocity and $\mathbf{j}$ is the unit vector in vertical direction. The velocity of ambient current $\mathbf{U}_a$ and the diffusion coefficient D_i are provided by a hydrodynamic model if no measured data is available.

While a packet of particles are introduced into the computational domain, a Lagrangian element containing all those particles is also defined to simulate the main buoyant jet. The average velocity and expansion of the Lagrangian element is calculated to provide the information for tracking particles. It is assumed that a group of particles begin to separate from the main buoyant jet when the vertical velocity of the main buoyant jet reduces to the terminal rise velocity of the group.

The flow chart of present model is shown in Figure 2. The model includes several interrelated sub-modules. Details on the sub-modules are described in the next section.

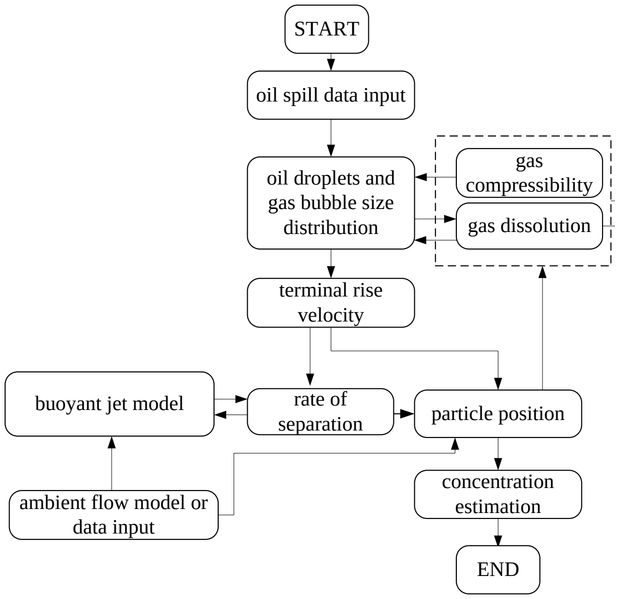


Fig.2 Flow chart of the present model

Sub-modules in the Present Model

Buoyant jet model

An integral Lagrangian control element method following Chen and Yapa (2004) is used to simulate the dynamics of the buoyant jet near the leakage point. The average velocity and expansion of the control element are then provided for estimating the mean velocity and random walk step of the particle within the main bouyant jet. Since the released oil and gas are modeled as N groups of particles with different properties, the model is modified as following. For a Lagrangian control element, we have

1) Mass conservation

$$M = m_w + \sum_{i=1}^{N} m_i \tag{3}$$

$$\frac{\mathrm{d}m_w}{\mathrm{d}t} = \rho_a Q_e \tag{4}$$

$$\frac{\mathrm{d}m_i}{\mathrm{d}t} = D_{sep,i} + D_{dis,i} \tag{5}$$

In which, M is the total mass of the Lagrangian control element; m_i is the total mass of the i-th group of particles, m_w is the mass of the entrained water. ρ_a is the density of ambient water, Q_e is volume flux of entrainment in the buoyant jet, which is computed following the method of Yapa and Zheng (1997). $D_{sep,i}$ is the rate of separation for the i-th group of particles. $D_{dis,i}$ is the rate of dissolution of the i-th group of particles.

2) Momentum conservation

$$\frac{\mathrm{d}MU}{\mathrm{d}t}=U_a\frac{\mathrm{d}m_w}{\mathrm{d}t}+\sum_{i=1}^{N}\frac{\mathrm{d}m_iU}{\mathrm{d}t} \tag{6}$$

$$\frac{\mathrm{d}MV}{\mathrm{d}t}=V_a\frac{\mathrm{d}m_w}{\mathrm{d}t}+\sum_{i=1}^{N}\frac{\mathrm{d}m_iV}{\mathrm{d}t} \tag{7}$$

$$\frac{\mathrm{d}MW}{\mathrm{d}t}=W_a\frac{\mathrm{d}M}{\mathrm{d}t}+\sum_{i=1}^{N}\frac{\mathrm{d}m_iW}{\mathrm{d}t}+\sum_{i=1}^{N}\frac{m_i}{\rho_i}\Delta\rho_i g \tag{8}$$

In which U, V, W are the velocity components of the control element in x, y, z direction, U_a, V_a, W_a are the velocity components of ambient current. ρ_i is the density of the i-th group of particles, $\Delta\rho_i$ is the density difference between the i-th group of particles and the ambient water, $\Delta\rho_i=\rho_a-\rho_i$.

Terminal rise velocity and rate of separation

Bubbles or droplets of different properties, such as density, diameter, surface tension, have different terminal rise velocities. The terminal rise velocity is very important in simulating the fate of oil and gas. It can be approximately calculated by the formula given by Zheng and Yapa (2000), which can be widely applied to solid particles, liquid droplets, or gas bubbles and can cover a broad range of bubble or droplet size.

The terminal rise velocity is also used as a very important parameter in calculating the rate of separation of bubbles and droplets from the main buoyant jet. When the separation occurs, the rate of separation for the i-th group of particles $D_{sep,i}$ is approximately estimated as

$$D_{sep,i}=m_{i,0}\frac{\mathrm{d}S_i}{\mathrm{d}t} \tag{9}$$

In which $m_{i,0}$ is the initial total mass of the i-th group of particles; S_i is the proportion of the overlapped cross-section area between the i-th group of particles and the main buoyant jet to the cross-section area of the i-th group, which is calculated by the method of geometry. Here, both the cross-section area of the i-th group of particles and the cross-section of the main buoyant jet are assumed as a circle, as shown in Figure 3.

The broken line circle shows the cross-section of the i-th group of particles, and the solid line circle shows the cross-section of the main buoyant jet. l_i is the deviation of the centrode of the i-th group of particles from the centrode of the main buoyant jet. The increasing quantity of l_i in one time step can be estimated by Eq. 10.

$$\Delta l_i=(w_{b,i}-W)\Delta t\cos\phi \tag{10}$$

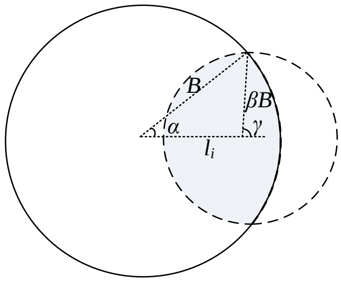


Fig.3 Calculation of the separation

In which Δl_i is the increasing quantity of l_i in one time step; $w_{b,i}$ is the terminal rise velocity of the i-th group of particles; ϕ is the angle between the main buoyant jet trajectory and the horizontal plane.

The radius of the cross-section of the main buoyant jet is B, and the radius of the cross-section occupied by the i-th group of particles is assumed as βB. It is clear that S_i=1 if $l_i\le B(1-\beta)$, and S_i=0 if $l_i\ge B(1+\beta)$. When $B(1-\beta)<l_i<B(1+\beta)$, the rate of overlapped cross-section area equals

$$S_i=1-\frac{\beta^2(2\gamma-\sin 2\gamma)-(2\alpha-\sin 2\alpha)}{2\beta^2\pi} \tag{11}$$

In which

$$\gamma=\arccos\frac{B^2(1-\beta^2)-l_i^2}{2l_i\beta B} \tag{12}$$

$$\alpha=\arccos\frac{l_i^2+B^2(1-\beta^2)}{2l_iB} \tag{13}$$

Gas compressibility and dissolution

In the case that gas bubbles are released in deep water, the variation of density and diameter of gas bubbles according to the compressibility and dissolution can not be ignored. Here, the compressibility of gas is considered. The density of gas varies according to the temperature and pressure, which can be given as follows.

$$PV=nZRT \tag{14}$$

In which P is the gas pressure, which approximately equals ambient pressure; n is the number of moles; V is the volume of gas, T is the absolute temperature, R is the universal gas constant, R=8.31J/mol.K, Z is compressibility factor. If gas is ideal gas, the compressibility factor Z=1. In this study, gas is approximately assumed as ideal gas.

Another important phenomenon is the gas dissolution into ambient water. The rate of gas dissolution in a bubble is estimated by

$$D_{dis,i} = -K_{dis} \quad HP - C \quad m_b A_b N_i \tag{15}$$

In which K_{dis} is the mass transfer coefficient [m/s]; H is the Henry's constant, [mol/m^3/bar]; P is the pressure, [bar]; C is the aqueous phase concentration, [mol/m^3], m_b is the molar mass, A_b is the surface area of bubble. N_i is the number of bubbles in the i-th group, which can be calculated as

$$N_i = \frac{m_{i,0}}{\frac{1}{6}\pi d_{e,i}{}^3 \rho_i} \tag{16}$$

Where $d_{e,i}$ is the volume-equivalent diameter of gas bubbles in the i-th group. The mass transfer coefficient K_{dis} can be calculated as following Zheng and Yapa (2002).

Concentration estimation

Each particle stands for a number of oil droplets or gas bubbles of similar physical parameters. After obtained the distribution of particles, the concentration distribution can be estimated by using the density kernels method (de Haan 1999). The concentration C at the position $\mathbf{x}$ can be estimated by n given particles as follows.

$$C(\mathbf{x}) = \frac{1}{h^3}\sum_{j=1}^{n} K\left(\frac{\mathbf{x}-\mathbf{x}_j}{h}\right) m_{p,j} \tag{17}$$

In which $\mathbf{x}_j$ is the position vector of the j-th particle, h is the width of the kernels, $m_{p,j}$ is the mass of the j-th particle, and $K(\mathbf{r})$ is the kernel function. The most widely used kernel function is the Gaussian kernel.

$$K(\mathbf{r}) = \frac{1}{(2\pi)^{3/2}} \exp\left(-\frac{1}{2}\mathbf{r}^T\mathbf{r}\right) \tag{18}$$

MODEL VALIDATION

Socolofsky et al. (1999) carried out a series of laboratory experiments to investigate the behavior of oil and gas buoyant jet in a cross-flow. They observed that gas bubbles and larger oil droplets tended to separating from the main buoyant jet. The laboratory experimental cases are adopted to validate the present model.

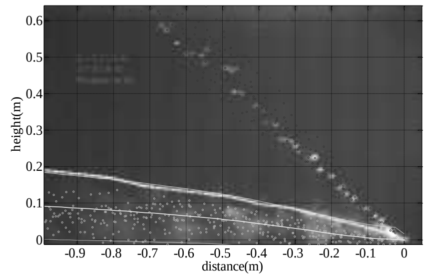
Modeling Gas Released in a Cross-Flow

This series of experiments was carried out to observe the pure gas buoyant jet, and dye was injected at the base of pure gas buoyant jets to show the entrained water.The conditions of the each case are listed in Table 1.

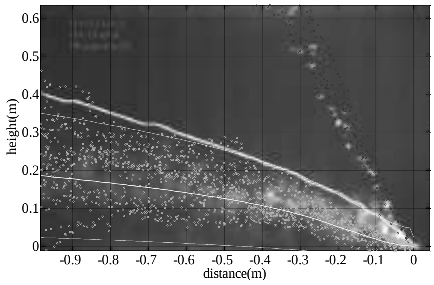
Table 1 Parameters in the experiments for gas bubbles

Case No.	Cross-flow velocity (cm/s)	Gas flow rate at STP (mL/min)
B1	20	200
B2	10	200
B3	5	200

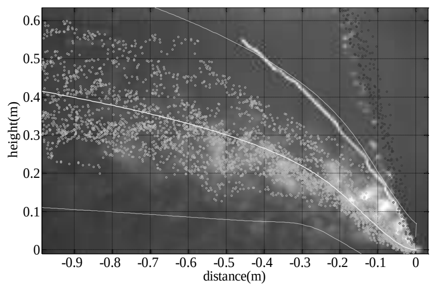
Two groups of particles are used for these cases. One is used to represent gas bubbles, the other is used to represent dye tracer. Ten particles are used for each group. Gas bubbles are assumed to be same size with a diameter 18mm, and the computed terminal rise velocity is about 0.1877m/s.



(a) B1



(b)B2



(c) B3

Fig.4 Compare the simulated results of the buoyant jet of air bubble with the experiment (Socolofsky et al. 1999).

The background of Figure 4 is the experiment photo, red circles show the particles for air bubbles, and green circles show the particles for dye, yellow lines show the range of main buoyant jet. It can be seen that the present model is able to simulate buoyant jets of gas bubbles, and good agreement is obtained between the simulation and observation results.

In present model, the critical condition for the separation of gas bubbles and oil droplets is a basic assumption and is of significant in describing the different behavior of bubbles and droplets. We assume that a group of particles began to separate from the main buoyant jet when the vertical velocity of the main buoyant jet reduces to the terminal rise velocity of the group. In Figure 4 we can see that the separation of gas is well simulated.

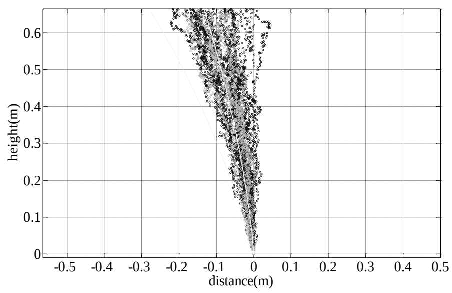
Modeling Oil-Gas Mixture Released in a Cross-Flow

In this section, the present model is applied to simulate oil-gas mixture released at the same time. The conditions are listed in Table 2.

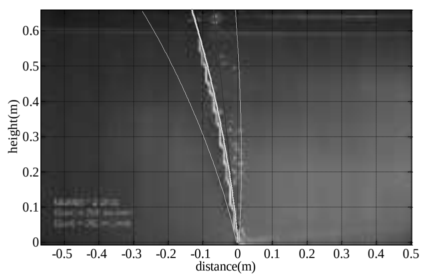
Table 2 Parameters in the experiments for air-oil mixture

Case No.	Cross-flow velocity (cm/s)	Gas flow rate at STP (mL/min)	Oil flow rate （mL/min）
C1	5	250	250
C2	2	250	250
C3	10	250	250

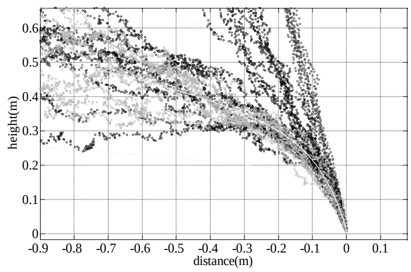
For case C1, C2 and C3, four groups of particles are used to represent gas bubbles, larger oil droplets, fine oil droplets and dye tracer respectively. Gas bubbles are assumed to be same size with a diameter 18mm, and the computed terminal rise velocity is about 0.1877m/s. The diameters of oil droplets are 3mm and 0.5mm.



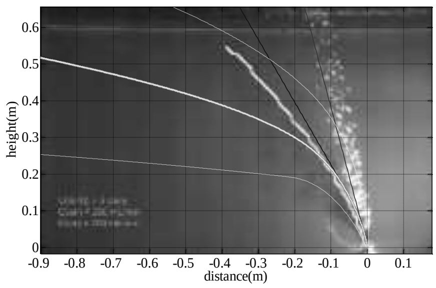
(a) C2 computed result



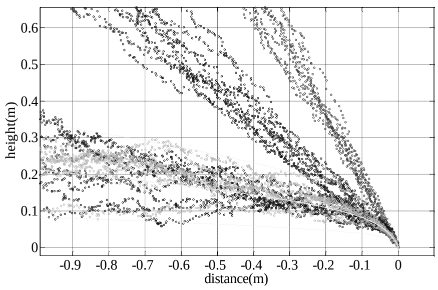
(b) C2 experiment photo



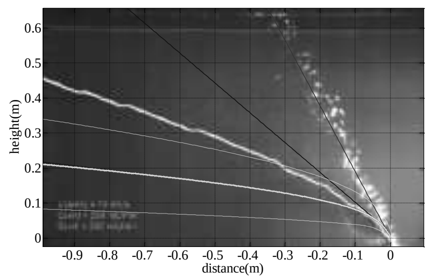
(c) C1 computed result



(d) C1 experiment photo



(e) C3 computed result



(f) C3 experiment photo

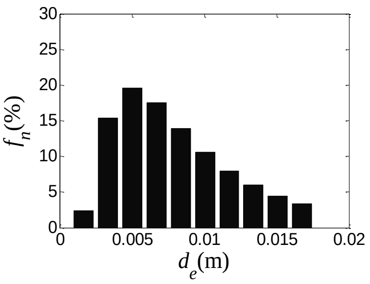
Fig.5 Compare the simulated results of air-oil mixture with the experiment (Socolofsky et al. 1999).

APPLICATION TO THE FIELD CASE

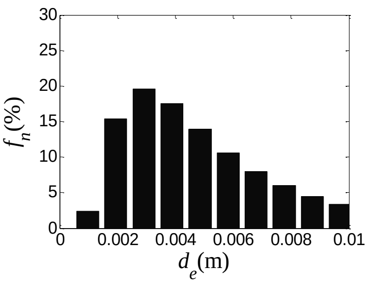
The Deepspill field experiment was conducted in the Norwegian Sea at the Helland Hansen site (Johansen et al. 2003), which is a famous field experiment to investigate the behavior of oil and gas during a deep water release. Gas and oil was released from a water depth of 844m. During the experiment, extensive observation was made including wind, currents, water density, surface and subsurface oil concentration, and so on. This experiment provides very detailed information that can be used to validate the present model.

The model is used to reproduce the diesel/gas case carried out on June 27, 2000. This case started at 6:35 and the release lasted 50min. Diesel discharge rate is 0.01667m^3/min, and Gas discharge rate is 0.6Sm^3/s. The density of diesel is 854.8kg/m^3. The diameter of the orifice is 0.12m. The ambient conditions including the ambient currents and temperature distribution is obtained from the measured actual value, and the variation of salinity and sea water density are ignored in present study. The diffusion coefficient of the ambient current is adopted as 0.1m^2/s in horizontal direction and 0.01m^2/s in vertical direction. Gas dissolution is considered.

Both gas bubbles and oil droplets are modeled by 10 groups of particles, according to the size distribution of gas bubbles and oil droplets estimated by the simplified MEF-based model. Figure 6 shows the estimated size distribution of gas bubbles and oil droplets, based on the method suggested by Chen and Yapa (2007).



(a) gas bubbles

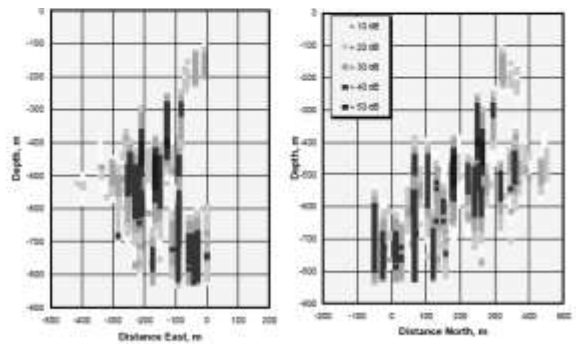


(b) oil droplets

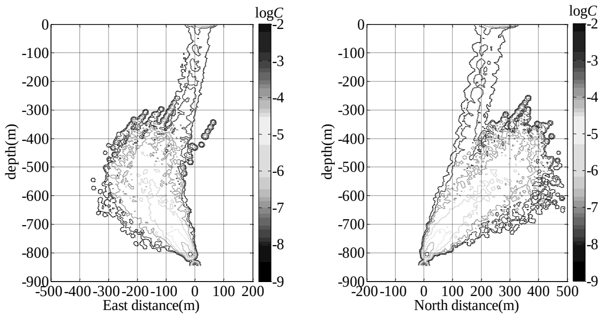
Fig.6 Estimated size distribution of gas bubbles and oil droplets

Figure 7(a) is the averaged echo-sounder data which is used to estimate the concentration profile of the oil and gas mixture in the experiment. Figure 7(b) show the projection of the maximum concentration computed by the present model. It can be seen that the present particle model can well reproduce the deep water release cases.

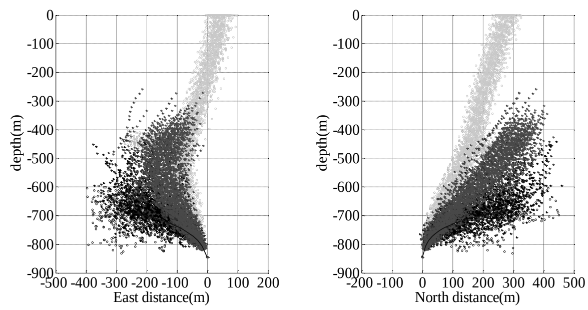
Figure 7(c) shows the side view of the modeled particle distribution, which can be compared with echo-sounder data shown in figure7(a). In Figure 7(c), blue circles represent for gas bubbles d_e<5mm; green circles represent for gas bubbles 5mm<d_e<8mm; cyan circles represent for gas bubbles 9mm<d_e<16mm; black asterisks represent for oil droplets d_e<3mm;red asterisks represent for oil droplets 3mm<d_e<6mm; purple asterisks represent for oil droplets 6mm<d_e<10mm. The different behaviors of bubbles or droplets of different sizes appear clearly in Figure 7(c). While big droplets are separated and rise up to the water surface, smaller droplet may travel underwater for a long distance.



(a) echo sounder data from the Marine Diesel experiment (Start at 06:30, June 27 07:01-08:04)



(b) simulated concentration profile after 1h



(c) simulated particle distribution after 1h

Fig.7 Simulating the Deepspill field experiment

CONCLUSION

In this study, a model based on particle tracking method was developed to predict the distribution of oil and gas released from an underwater blowout. In order to simulate the behaviors of gas bubbles and oil droplets of different properties, including bubble or droplet size, density, surface tensor, etc., the released mass is divided into several groups. Each group has the similar property and is represented by a number of particles. The movement of each particle is simulated by a random walk method. At each time step, the movement of each particle including two parts, one is the mean velocity, and the other is the random walk to simulate the turbulent diffusion. The movement of each particle is divided into two regions, moving within the main buoyant jet or moving individually with the terminal rise velocity and ambient current. The critical condition for the two regions is assumed to be the buoyant velocity of particle equals the vertical velocity of the main buoyant jet. Within the main buoyant jet, the velocity of particle is assumed as the same of the mean cross-section velocity of the main buoyant jet, which is calculated following Yapa and Zheng (1997) with some improvements. In present model, the seperation of big oil droplets as well as gas bubbles in cross-flow can be well simulated. Comparing with the previous models for underwater oil spill based on the integral Lagrangian control volume method, the present model is more flexible in simulating the crude oil which has complex components. The model was applied to the experiment on multiphase buoyant jet carried out by Socolofsky et al. (1999), and the results showed a good agreement with the experiment. The model was also applied to reproduce the DeepSpill field experiment, and the modeled oil-gas cloud agreed well with the echo sounder data.

ACKNOWLEDGEMENTS

The author would like to acknowledge the support by the National Natural Science Foundation of China (51109119), the State key Laboratory of Hydroscience and Engineering of China (2011-KY-1) and Specialized Research Fund for the Doctoral Program of Higher Education (20110002120019).

REFERENCES

Chen, F.H. and Yapa, P.D. (2004). Modeling gas separation from a bent deepwater oil and gas jet/plume. Journal of Marine Systems, 45(3-4): 189-203.

Chen, F.H. and Yapa, P.D. (2007). Estimating the oil droplet size distributions in deepwater oil spills. Journal of Hydraulic Engineering, ASCE, 133(2): 197-207.

Dasanayaka, L.K. and Yapa, P.D. (2009). Role of plume dynamics phase in a deepwater oil and gas release model. Journal of Hydro-environment Research, 2(4): 243-253.

de Haan, P. (1999). On the use of density kernels for concentration estimations within particle and puff dispersion models. Atmospheric Environment, 33(13): 2007-2021.

Guo, W.J. and Wang, Y.X. (2009). A numerical oil spill model based on a hybrid method. Marine Pollution Bulletin, 58(5): 726-734.

Johansen, O. (2000). DeepBlow - a Lagrangian plume model for deep water blowouts. Spill Science & Technology Bulletin, 6(2): 103-111.

Johansen, Ø., Rye, H. and Cooper, C. (2003). DeepSpill––Field Study of a Simulated Oil and Gas Blowout in Deep Water. Spill Science & Technology Bulletin, 8(5-6): 433-443.

Korotenko, K.A., Mamedov, R.M., Kontar, A.E. and Korotenko, L.A. (2004). Particle tracking method in the approach for prediction of oil slick transport in the sea: modelling oil pollution resulting from river input. Journal of Marine Systems, 48(1-4): 159-170.

Nagheeby, M. and Kolahdoozan, M. (2010). Numerical modeling of two-phase fluid flow and oil slick transport in estuarine water. International Journal of Environmental Science and Technology, 7(4): 771-784.

Rodi, W. (1982). Turbulent Buoyant Jets and Plumes. Pergamon Press.

Socolofsky, S., Leos-Urbel, A. and Adams, E.E. (1999). Report Summary: CORMIX 3.2 Analysis of droplet plumes in a cross-flow. U.S. Department of the Interior Minerals Management Service Contract No. 1435-01-98-CT-30964.

Socolofsky, S.A. and Adams, E.E. (2002). Multi-phase plumes in uniform and stratified crossflow. Journal of Hydraulic Research, 40(6): 661-672.

Yapa, P.D. and Zheng, L. (1997). Simulation of oil spills from underwater accidents I: Model development. Journal of Hydraulic Research, 35(5): 673-687.

Yapa, P.D., Dasanayaka, L.K., Bandara, U.C. and Nakata, K. (2010). A model to simulate the transport and fate of gas and hydrates released in deepwater. Journal of Hydraulic Research, 48(5): 559-572.

Zheng, L. and Yapa, P.D. (1998). Simulation of oil spills from underwater accidents II: Model verification. Journal of Hydraulic Research, 36(1): 117-134.

Zheng, L. and Yapa, P.D. (2000). Buoyant velocity of spherical and nonspherical bubbles/droplets. Journal of Hydraulic Engineering, ASCE, 126(11): 852-854.

Zheng, L. and Yapa, P.D. (2002). Modeling gas dissolution in deepwater oil/gas spills. Journal of Marine Systems, 31(4): 299-309.