2014 INTERNATIONAL OIL SPILL CONFERENCE

A Numerical Model for Oil Droplet Evolution Emanating from Blowouts

Lin Zhao[1], Michel C. Boufadel[1], E. Eric Adams[2], Scott A. Socolofsky[3], Kenneth Lee[4]

1: Center for Natural Resources Development and Protection, Department of Civil and Environmental Engineering, the New Jersey Institute of Technology, Newark NJ.
boufadel@gmail.com; http://nrdp.njit.edu

2: Department of Civil and Environmental Engineering, Massachusetts Institute of Technology, Massachusetts.

3: Department of Civil Engineering, Texas A&M, Texas.

4: Flagship of Wealth from Oceans, Commonwealth Scientific and Industry Research Organization (CSIRO), Perth, Australia

ABSTRACT 300034:

This paper presents the details of a numerical model that is capable of simulating the droplet size distribution emanating from blowouts. The model was obtained as a result of combination of traditional mechanistic models developed in reactors with jet (or plume) models to predict the evolution of the plume away from the orifice. Inputs to the model include the energy dissipation rate (or the mixing energy) and holdup, which is the volume fraction of oil in the control volume. These parameters vary as the plume spreads away from the orifice. They have a maximum value near the orifice and rapidly decrease as moving away from the orifice. The model was validated using experimental data available in the literature. Subsequently, the model was used to predict the evolution of droplets in the Deepwater Horizon incident. The model provides the variation of the mean diameter and the droplet size distribution with depths away from the orifice. The sensitivity of different parameters, such as interfacial tension which could present the addition of dispersants was also evaluated.

Keywords: Oil Droplet, Numerical Model, Jet Dynamics, Oil Spill, Deepwater Blowouts

INTRODUCTION:

Deepwater oil and gas blowout is subject to the release of oil and gas from submerged jets and plumes in marine environment. Due to the interactions of oil and gas with the surrounding water after release, different sizes of oil droplets and bubbles are formed at different depths of the water column. Larger and smaller droplets will have different rising time because of their different buoyancy effects. Ryerson et al. (2012) estimated the rise time of large droplets to the surface to be a few hours, while that of smaller droplets to be on the order of weeks in addressing the Deepwater Horizon spill events. Fine droplets may even be kept in the water column for prolonged periods due to factors of vertical turbulence mixing, density stratification, and cross flow (Brandvik et al., 2013; Johansen et al., 2013). The fate and transport of spilled oil in deep waters are evidently influenced by the droplet sizes. Therefore, in response to deepwater oil spill events, there is an urgent need to study the droplet formation processes and obtain the evolution of droplet size distribution (DSD) in submerged jets and plumes.

Our recent developed numerical model, VDROP is capable of simulating the transient droplet size distribution in turbulent regimes, and has been validated extensively against various experimental data (Zhao et al., 2014a). To extend VDROP model in the simulation of droplet size distributions along jet (and/or plume) trajectories, it is required to understand the jet dynamics and the energy distributions along the jets (and/or plumes), because essentially it is the energy forces that prevail in the formation processes of droplets. Therefore, the jet dynamics of a submerged free round jet was studied based on experimental data from literatures, and an empirical formulation was constructed. The VDROP model was integrated with the empirical jet model. Comparisons with experimental data were conducted. Finally, simulations of Deep Water Horizon blowout events were performed.

METHODOLOGIES:

The transient droplet size distribution model, VDROP model developed by Zhao et al. (2014a), has been expanded to be able to simulate the evolution of droplet size distribution along the trajectory of jets and plumes during experimental studies in a laboratory facilities and field events of deepwater oil spills. VDROP model is a comprehensive numerical model with the ability to simulate the transient droplet size distribution in turbulent regimes. In order to simulate oil spill events from a deepwater orifice, the VDROP model was integrated with an empirical jet model which was developed based on the understanding of jet dynamics of a submerged round jet. The resulting integrated model relies on conceptually moving a given volume of oil downstream of the blowout orifice, and allowing the volume to be subjected to a decreasing mixing energy and dilution (due to water entrainment). Detailed descriptions of the methodologies are presented below.

VDROP Model

The population balance equation has been widely used for predictions of droplet formation processes. Different forms of integral-differential equation of the population balance equation can be found in many previous studies (Bandara and Yapa, 2011; Coulaloglou and Tavlarides, 1977; Tsouris and Tavlarides, 1994). For a discrete droplet size system, the following population balance equation was used in VDROP model:

$$\frac{\partial n(d_i,t)}{\partial t} = \underbrace{\sum_{j=i+1}^{n} \beta(d_i,d_j)g(d_j)n(d_j,t) - g(d_i)n(d_i,t)}_{\text{Droplet Breakup}} + \underbrace{\underbrace{\sum_{j=1}^{n}\sum_{k=1}^{n}}_{v_j+v_k=v_i} \Gamma(d_j,d_k)n(d_j,t)n(d_k,t) - n(d_i,t)\sum_{j=1}^{n}\Gamma(d_i,d_j)n(d_j,t)}_{\text{Droplet Coalescence}} \quad (1)$$

where n is number concentration (number of droplets/m^3) of droplets of diameter d_i (m) at a given time t (second). $\beta(d_i, d_j)$ is the breakage probability density function (dimensionless) for the creation of droplet of diameter d_i due to breakage of droplets of (a larger) diameter d_j. $g(d_j)$, (/s) is the breakage frequency of droplets of diameter d_j. For droplet breakup, the first term represents the birth of droplets d_i resulting from the breakup of droplets d_j, while the second term represents the death of droplets d_i due to breakup into smaller droplets; For droplets coalescence, the term $\Gamma(d_k, d_j)$ is the coalescence rate (m^3/s), the first term represents the birth of droplets d_i as a results of coalescence events occurring between droplets d_k and d_j to form drops with the size of d_i, while the second term represents deaths of droplets d_i due to the coalescence of drops d_i with all other drops (including drops of size d_i themselves) to form larger drops.

Jet Dynamics – Development of an Empirical Jet Model

It is essential to understand the jet dynamics before performing simulations of droplet formations in turbulent jets and plumes. Many fluid discharges into the environment involve a simple structure such as the open end of a submerged pipe. Therefore, a free round jet (a free jet refers to as a jet in an environment in which the boundaries (e.g. walls, surface or bottom) will not affect the development of jet trajectories) is considered in this context. Initial discharge conditions usually control the flow near the releasing point of a jet or plume. The primary factors of jet dynamics include the mass flux, momentum flux, and buoyancy flux. For a round jet, the initial values of these factors can be expressed as:

The volume flux

$$Q_0 = \frac{1}{4}\pi D_0^2 U_0 \tag{2}$$

The momentum flux

$$M_0 = \frac{1}{4}\pi D_0^2 U_0^2 \tag{3}$$

The buoyancy flux

$$B_0 = g(\Delta\rho / \rho)Q_0 \tag{4}$$

where D_0 is the jet diameter, U_0 is the mean outflow velocity, g is the gravitational acceleration, $\Delta\rho_0$ is the difference in density between the receiving fluid and the discharged fluid.

The empirical formulations for the primary parameters describing jet dynamics are established based on many experimental investigations. Figure 1 shows one of the fitting results of energy dissipation rate with two experimental data. Considering a turbulent jet through an orifice whose diameter is D_0, the primary factors in describing the initial conditions of jet dynamics can be obtained from Eq. 2-4.

The characteristic length scales for jets and plumes are given by $l_Q = \sqrt{\pi / 4} D_0$ and $l_M = \frac{M_0^{3/4}}{B_0^{1/2}}$.

Based on these characteristic length scales, the centerline velocity, U_c, is given by:

For $x < x_0$, $U_c = U_0$ (5a)

For $x_0 \le x \le 2l_M$

$$U_c = \min\left(5.4U_0(\frac{D_0}{x - x_0}), U_0\right) \tag{5b}$$

For $x > 2l_M$

$$U_c = \min\left(\frac{4.7B_0^{1/3}}{(x - x_0)^{1/3}}, U_0\right) \tag{5c}$$

The variation of the energy dissipation rate (ε) away from the orifice can be calculated as follows:

$$\varepsilon = \min\left(\frac{0.003U_0^3}{D_0}, \frac{CU_c^3}{b}\right) \tag{6}$$

where C is a constant varying between 0.024 and 0.0326; the half-width "b" is given by Fischer (1979) as:

$$\mathrm{b} = \max\left(0.1\mathrm{x}, \frac{\mathrm{D_o}}{2}\right) \tag{7}$$

The holdup (dispersed phase fraction) at a downstream distance x can be calculated as:

$$\varphi = \frac{\mathrm{Q_{oil}}}{\mathrm{Q}} \tag{8}$$

where Q_{oil} is the flow rate of oil and Q is the plume flow rate given by:

$$Q = \max\left(\pi U_c b^2, Q_0\right) \tag{9}$$

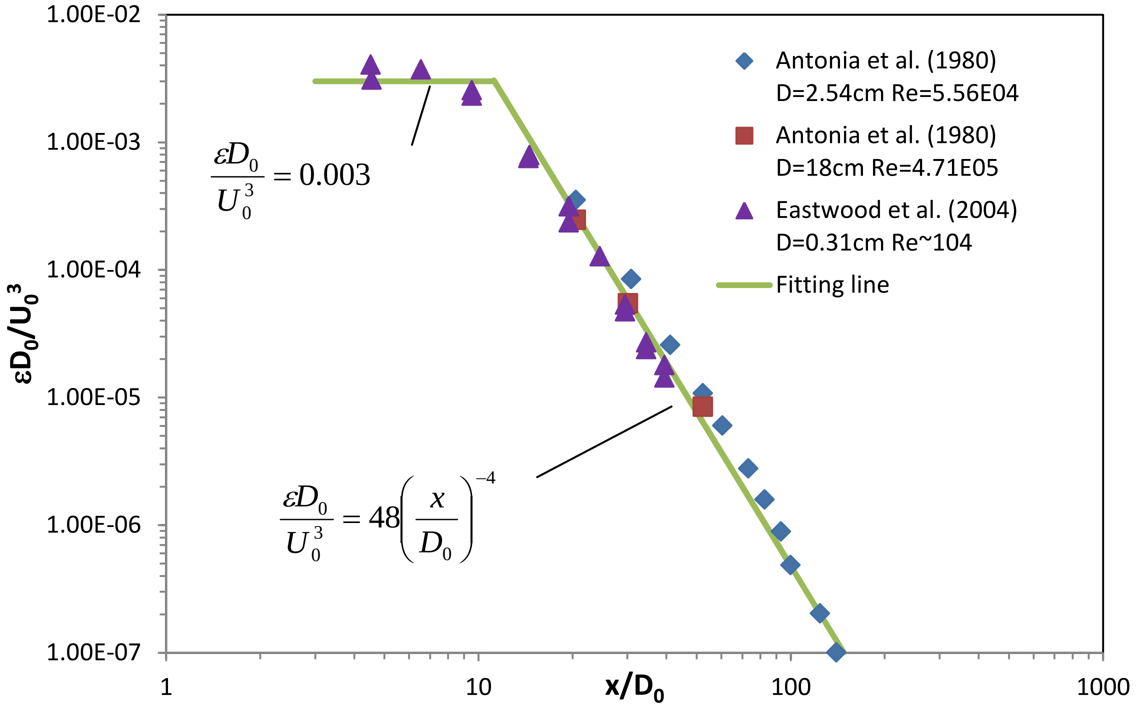

Figure 1 Fitting the turbulent kinetic energy ε, compared with experimentally determined values.

Integration of VDROP and Jet Models

The integration of VDROP and Jet model is based on the concept of Lagrangian method in fluid dynamics, in which a given volume of fluid parcel is tracked while it is moving along the jet trajectory. The fluid parcel being tracked has initial values of volume of V_0, number concentration (numbers of droplets per unit volume) n_0, velocity u_0, energy dissipation rate ε_0, and holdup φ_0. When the parcel is moving from position 0 to position 1, the volume of the parcel becomes V_1 because of the entrainment of surrounding water, and the primary parameters of the fluid parcel in position 1 can be calculated from Eq. 5-9. The droplet size distribution in position P1 can be obtained by updating the energy dissipation rate and the holdup value in position 1. The time step chosen for the simulation is normally very small (less than 0.01 s). After obtaining the droplet size distribution in position 1, the calculation can step to next time step and the procedure repeats. This iterative process yields a droplet size distribution resulting from droplet breakup and coalescence along the jet trajectory.

VALIDATIONS:

The developed model was compared with the experiments conducted by Neto et al. (2008). The experiments were performed in a square glass-walled tank with a single orifice nozzle (the nozzle diameter can be changed) located 45 mm above the bottom. The tap water and air were mixed first and then discharged through the orifice. The temperature of both water and air were fixed at about 20 ºC, and a pressure-regulating valve was used to keep the air pressure at

1 atm. The measurements were taken from the bubbly jet centerline and at a height of 43 cm above the nozzle exit.

Figure 2 shows the comparison of droplet size distribution of modeling results and experimental data for orifice nozzle of 6 mm. Modeling results have good agreement with the experimental data. The predicted d_{50} values of Case 1 and Case 2 are 7.0 mm (observed 7.3) and 2.4 mm (observed 2.9 mm), respectively. The modeling results of Case 2 (the case with larger exit velocity) gave slightly widened droplet size distribution compared to the experimental data.

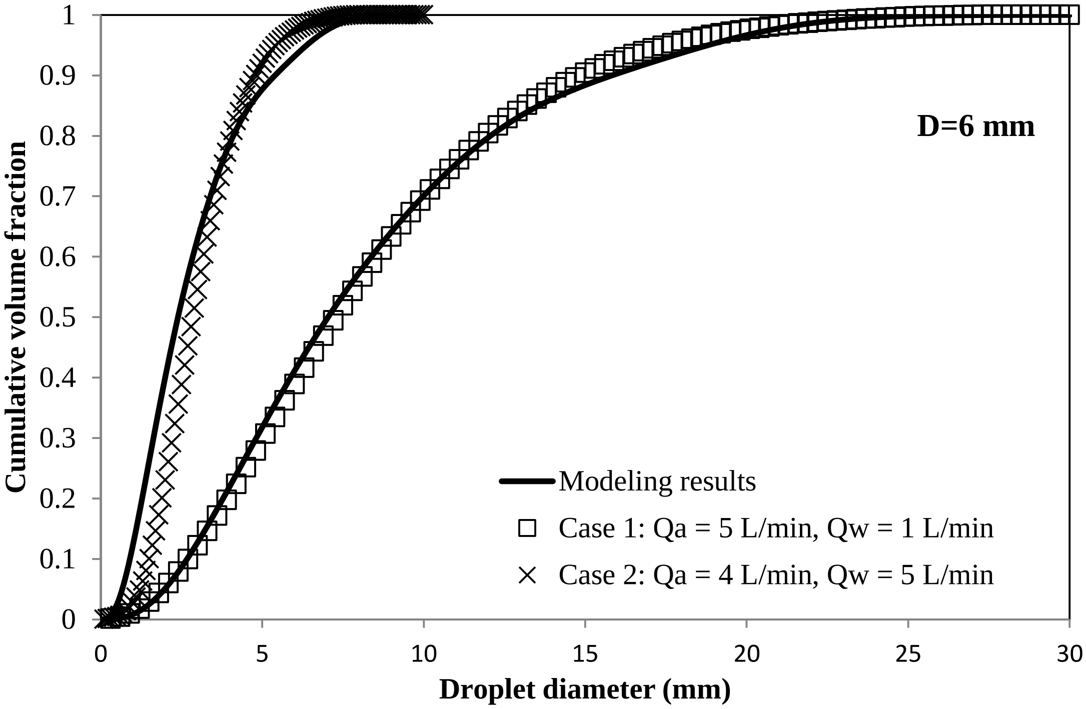

Figure 2 Comparison of oil droplet size distribution (cumulative volume fraction v.s. droplet diameter) with orifice diameter of 6 mm between modeling results and experimental data at 43 cm downstream from the discharge orifice. The experimental measurements were obtained from Neto et al. (2008).

SIMULATION OF DEEPWATER HORIZON EVENTS:

After the explosion and sinking of the Deepwater Horizon mobile offshore drilling unit, crude oil and natural gas from the Macondo well were released into the Gulf of Mexico at ~1.5 km depth of for a period of 87 days (Valentine et al., 2012). The total mass fluxes estimated were 5.4×10^{11} g for liquid oil and $1.8 – 3.1 \times 10^{11}$ g for natural gases. Socolofsky (2012) estimated an oil density at ambient seafloor conditions of 858 kg/m^3. Camilli et al. (2012) indicated that the ambient seafloor pressure and temperature regime at the Macondo well site impose a minimum initial gas phase density of at least approximately 120 kg/m^3. Based on our calculations of methane at 1.5 km depth, the gas density of 131 kg/m^3 was used in the simulation. Combining mass fluxes and densities of oil and natural gases, it gives us an average flow rate of oil of 0.088 m^3/s, and 0.18-0.31 m^3/s for natural gas. The estimated average oil and gas flow rates were consistent with the values of net oil and gas flow rates reported by Camilli et al. (2012). The broken riser source diameter was reported close to 0.5 m (Camilli et al., 2012). The interfacial tension and viscosity of the dispersed oil were 20.9 mN/m and 18.5 cp, respectively.

The model was configured based on the above mentioned discharge information. The largest droplet size was assumed as 10 mm based on the deep spill experiments conducted by Johansen et al. (2000). There are 100 size bins with 0.1 mm interval used for the simulation.

Figure 3 shows the modeling results with different values of interfacial tension at 10 m and 50 m downstream distances from the exit orifice, above the discharge point. The smaller value of interfacial tension may represent the addition of dispersants which is known to reduce oil-water interfacial tension and enhance the formation of smaller droplets. Results of untreated oil are illustrated in Figure 3a. At 10 m downstream from the orifice, most of the oil is in droplets larger than 8.5 mm; the predicted d_{50} at this depth is 9.06 mm. As the downstream distances increase, there is increased dispersion of the larger droplets; smaller sizes of droplets are formed as shown in Figure 3a, with the results for 50 m downstream predicting a d_{50} of 4.66 mm. Figure 3b shows the results with interfacial tension reduced by a factor of 10 to represent the addition of dispersants. The predicted d_{50} at 10 m and 50 m downstream are 3.91 mm and 1.13 mm, respectively. Comparing Figure 3a and 3b, reducing interfacial tension by 10 fold reduces the droplet sizes by 60% to 75%, especially at distances further away from the orifice: at 50 m the oil droplet size is basically between 1 mm to 2 mm. An interesting observation is that a bimodal profile was generated for both untreated oil and with dispersants. As noted in Figure 3b (with 10 fold reduction of interfacial tension), as downstream distance increase, the bimodal profile will eventually becomes uni-modal. Further investigations are still needed for this study, and more scenarios will be performed to present results under different conditions.

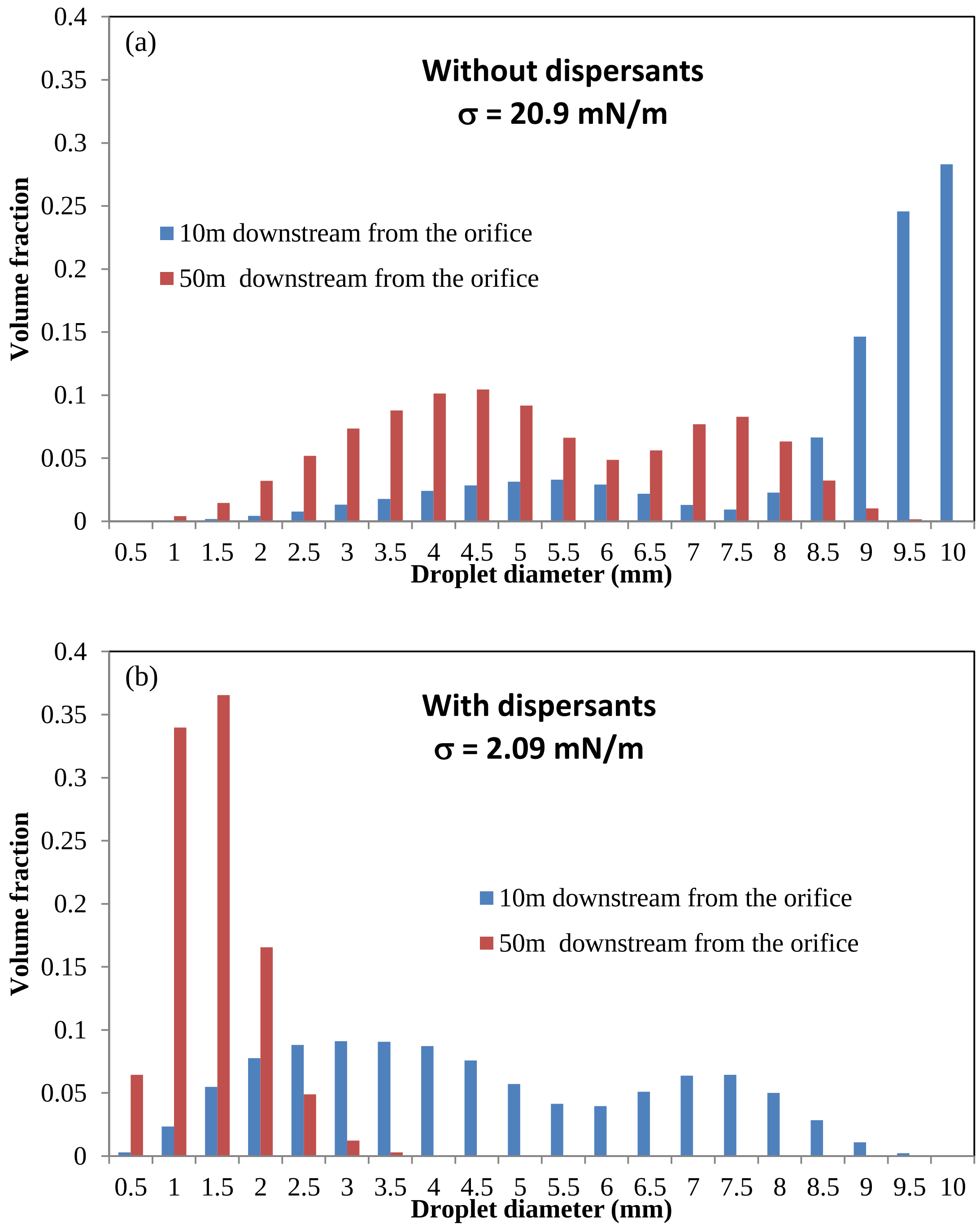

Figure 3 Simulation of droplet volume distribution of Deepwater Horizon Well (DHW) at 10 m and 50 m above the discharge point, a) without dispersant, the oil-saltwater interfacial tension of 20.9 mN/m; b) with dispersant, assuming that the oil-saltwater interfacial tension reduced to 2.09 mN/m

CONCLUSION:

To study the impacts of deep water oil and gas blowouts, it is important to understand the fate and transport of oil droplets and its formation processes. The VDROP model (Zhao et al., 2014a) was integrated with a jet model to produce the droplet size distribution (DSD) resulting from breakup and coalescence of oil droplets at different downstream distances from the exit orifice. Model validations were conducted by comparing with laboratory experimental data, which show good agreement modeling results. Finally, the developed model was used to simulate the Deepwater Horizon well blowouts based on the oil and gas release information and local sea water conditions. Cases with and without dispersants are presented for 10 m and 50 m downstream distance from the exit. Results show that the reduction of interfacial tension (represents the addition of dispersants) significantly promotes the formation of smaller droplets at both distances. Further studies are still required to investigate different discharge conditions.

The model could be further applied to oil spill response. Once the discharge conditions (e.g., orifice diameter and oil/gas flow rate) are estimated during a spill incident, the model could provide the possible droplet size distribution in the near field. Further investigations of the evolution of droplet formation from the reservoir to the seafloor and the fate and transport of oil droplets in the far field are conducted in on-going studies. Also, the VDROP model was used to simulate the droplet formation due to wave effects for a surface spill in a parallel study (Zhao et al., 2014b).

REFERENCES:

Antonia, R.A., Satyaprakash, B.R., Hussain, A.K.M.F., 1980. Measurements of dissipation rate and some other characteristics of turbulent plane and circular jets. Phys. Fluids 23, 695-700.

Bandara, U.C., Yapa, P.D., 2011. Bubble Sizes, Breakup, and Coalescence in Deepwater Gas/Oil Plumes. Journal of Hydraulic Engineering 137, 729-738.

Brandvik, P.J., Johansen, Ø., Leirvik, f., Farooq, U., daling, P.S., 2013. Droplet breakup in subsurface oil releases - Part I: Experimental study of droplet breakup and effectiveness of dispersant injection. Marine Pollution Bulletin 73, 8.

Camilli, R., Di lorio, D., Bowen, A., Reddy, C.M., Techet, A.H., Yoerger, D.R., Whitcomb, L.L., Seewald, J.S., Sylva, S.P., Fenwick, J., 2012. Acoustic measurement of the Deepwater Horizon Macondo well flow rate. PNAS 109, 5.

Coulaloglou, C.A., Tavlarides, L.L., 1977. Description of interaction processes in agitated liquid-liquid dispersions. Chemical Engineering Science 32, 1289-1297.

Eastwood, C.D., Armi, L., Lasheras, J.C., 2004. The breakup of immiscible fluids in turbulent flows. J. Fluid Mech 502, 309-333.

Johansen, Ø., Brandvik, P.J., Farooq, U., 2013. Droplet breakup in subsea oil releases – Part 2: Predictions of droplet size distributions with and without injection of chemical dispersants. Marine Pollution Bulletin.

Johansen, Ø., Rye, H., Melbye, A.G., Jensen, H.V., Serigstad, B., Knutsen, T., 2000. Deep Spill JIP - Experimental Discharges of Gas and Oil at Helland Hansen - June 2000. Technical Report, DeepSpill JIP, SINTEF, Report No. STF66 F01082.

Neto, I.E.L., Zhu, D.Z., Rajaratnam, N., 2008. Bubbly jets in stagnant water. International Journal of Multiphase Flow 34, 12.

Ryerson, T.B., Camilli, R., Kessler, J.D., Kujawinski, E.B., Reddy, C.M., Valentine, D.L., Atlas, E., Blake, D.R., de Gouw, J., Meinardi, S., Parrish, D.D., Peischl, J., Seewald, J.S., Warneke, C., 2012. Chemical data quantify Deepwater Horizon hydrocarbon flow rate and environmental distribution. Proceedings of the National Academy of Sciences.

Socolofsky, S.e.a., 2012. Formation dynamics of subsurface hydrocarbon intrusions following the Deepwater Horizon blowout. Geophys. Res. Lett. 38.

Tsouris, C., Tavlarides, L., 1994. Breakage and coalescence models for drops in turbulent dispersions. AIChE Journal 40, 395-406.

Valentine, D.L., Mezic, I., Macesic, S., Cmjaric-Zic, Hogan, P.J., Fonoberov, V.A., Loire, S., 2012. Dynamic autoinoculation and the microbial ecology of a deep water hydrocarbon irruption. PNAS Early Edition, 1-6.

Zhao, L., Torlapati, J., Boufadel, M.C., King, T., Robinson, B., Lee, K., 2014a. VDROP: A numerical model for the simulation of droplet formation from oils of various viscosities. Chemical Engineering Journal Submitted (CEJ-D-14-00789).

Zhao, L., Torlapati, J., King, T., Davidson, B., Boufadel, M.C., Lee, K., 2014b. A numerical model to simulate the droplet formation process resulting from the release of diluted bitumen products in marine environment, Proceedings of theInternational Oil Spill Conference, Savannah Georgia, USA.