Numerical simulation of deep-water oil blowouts

Daniel Cardoso Cordeiro

大阪大学基礎工学研究科

1. Introduction

In 2010, the largest offshore blowout of all time happened in the Gulf of Mexico, USA. The accident became known as the Deep-water Horizon oil spill in which around 4.9 million oil barrels were spilled into the ocean, and chemical dispersants were applied in a large scale into the subsea as a way to treat the oil blowout plume [1].

Given the possible negative outcomes that subsea injection of chemical dispersants may cause to the environment, it is necessary to assess clearly, whether this technique is specifically adequate for a certain accident scenario before its application. However, since real-scale experiments are unpractical due to their obvious environmental risk, numerical models and laboratory-scale experiments are the only available tools to investigate the blowout phenomenon. Therefore, the development of an accurate numerical blowout model would represent a valuable asset in the decision-making process for the remediation of such accidents with chemical dispersants.

A number of challenges stand in the way of achieving a more exact blowout model. Among them are the sensitivity of model predictions to complex chemical and biological phenomena, the need for a coupling of ocean circulation models with the near-field blowout plume dynamics and the accuracy of the oil/gas droplet size distribution (DSD) [2]. Particularly, the droplet size distribution (DSD) represents a serious threat to the preciseness of the spilled oil path inside the sea as a small variation in the droplet diameter results in a large discrepancy in the distance that the oil could travel both laterally and vertically inside the ocean. This could induce the response team to choose unsuitable remediation techniques with further damage to the environment.

In this research, we propose the use of a LES (Large Eddy Simulation) for turbulence with a hybrid Euler-Euler and volume of fluid (VOF) models to accurately predict the DSD by calculating the actual breakup and coalescence caused by the atomization of the turbulent oil jet into a quiescent water media.

2. Numerical methods

To simulate the oil blowout phenomena, a hybrid Eulerian multifluid solution framework with a volume of fluid (VOF) sharp interface capturing algorithm was used. It improves the accuracy of the traditional Euler-Euler method without the computational burden of coupling an entire interface-capturing model [3]. The present model was validated against a series of laboratory-scale oil blowout experiments. The deterministic approach based on the large-eddy simulation does not rely on the input of any empirical data nor calibration parameters, increasing the reliability to the real-scale prediction of the DSD in deep-water blowouts.

The governing equations solved are the Continuity (Eq. 1) and Navier-Stokes (Eq. 2) equations:

$$\frac{\partial \alpha_k}{\partial t} + \boldsymbol{u_k}.\nabla \alpha_k = 0 \tag{1}$$

$$\frac{\partial(\rho_k \alpha_k \boldsymbol{u_k})}{\partial t} + (\rho_k \alpha_k \boldsymbol{u_k}.\nabla)\boldsymbol{u_k} = -\alpha_k \nabla p + \nabla.(\mu\alpha \nabla \boldsymbol{u_k}) + \rho_k \alpha_k \mathbf{g} + \boldsymbol{F_{D,k}} + \boldsymbol{F_{S,k}} + \boldsymbol{F_{vm,k}} \tag{2}$$

where u is the velocity, is the volume fraction, t is time, ρ is the density, p is the pressure, g is the gravity acceleration, $F_{S,k}$ is the surface tension force, $F_{D,k}$ is the drag force, $F_{vm,k}$ is the virtual mass force and the subscript k indicates the fluid phase. The Euler-Euler method was used to model the multiphase flow coupled with a Schiller-Naumann drag model. The open-source OpenFOAM v4.0 was used for the calculations.

Table 1: Configuration of the cases: dispersants to oil ratio, DOR; surface tension, σ [mN/m]; inlet diameter, D [mm], inlet velocity, U_{in} [m/s]; the jet Reynolds number, Re; the jet Weber number, We; the oil-water viscosity ratio, ν_o/ν_w; the oil-water density ratio, ρ_o/ρ_w ; simulated time interval, ΔT.

	Case	DOR (%)	σ	D	U_{in}	Re	We	ν_o/ν_w	ρ_o/ρ_w	$\Delta TU_{in}/D$
——	1	0%	28.3	14	0.65	1076	179	8.46	0.856	428
- - - -	2	1%	0.6	14	0.65	1076	8438	8.46	0.856	428
······	3	0%	28.3	9	0.65	691	115	8.46	0.856	667
—▲—	4	0%	28.3	14	0.16	264	11	8.46	0.856	114
- -△- -	5	1%	0.6	14	0.16	264	511	8.46	0.856	114
—●—	6	0%	28.3	14	0.65	2151	179	4.23	0.856	428
- -○- -	7	0%	28.3	14	0.65	4302	179	2.11	0.856	428
—■—	8	0%	28.3	14	0.65	1076	125	8.46	0.600	428
- -□- -	9	0%	28.3	14	0.65	1076	84	8.46	0.400	428

The domain was modeled based on the experimental data [4], as a 1.0 m height cuboid tank with a 0.3 m per 0.3 m width. Oil flows through a round inlet at the center of the tank initially filled with water. The DSD was computed a posteroiri by isolating the droplet in the isosurface alpha = 0.1.

３. Results and Discussion

The effects of different parameters over the jet atomization in the DSD were showed using non-dimensional numbers and figures of the jet behavior.

The velocity inlet altered the regime in different cases: 1-2 were atomization and 4-5 were second wind induced breakup. Both the regimes showed very different characteristics regarding the droplet size distribution and the velocity of the particles as per Fig. 1.

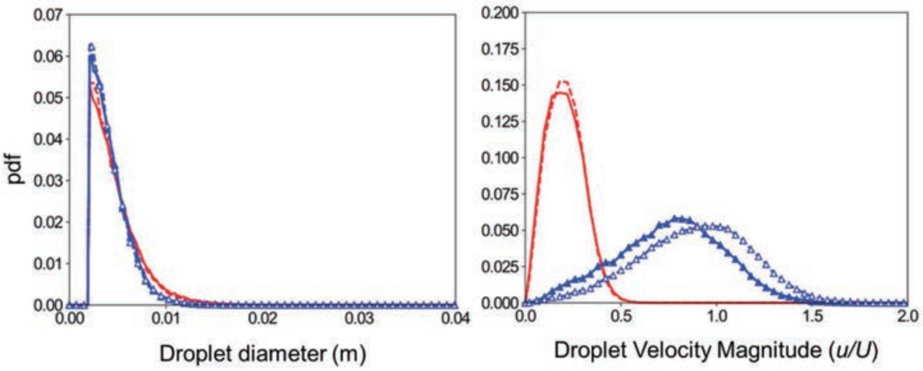

Figure 1: Droplet diameter distribution (left) and droplet velocity distribution (right) for cases 1, 2, 4 and 5.

From the droplets local information, it was possible to conclude that for the case of surface tension reduction, the droplet diameter was slightly reduced while the droplet velocity distribution behaved differently for the cases in different breakup regimes. As Fig. 1 also shows, in the atomization regime (Cases 1 and 2), the droplet velocity was reduced and increased in the second wind induced regime (Cases 4 and 5).

The inlet diameter reduction did not change the flow regime; however, it intensified Kelvin-Helmholtz instabilities near the inlet and narrowed the droplet size distribution (Fig. 2).

Figure 2: Effect of the diameter inlet reduction in the snapshots of the isosurface alpha = 0.1 for cases 1 (left) and 3 (right). The details show the Kelvin-Helmholtz instabilities.

The cases with variation in the density (1, 8, 9) and the viscosity ratio (1, 6, 7), showed in Figs. 5 and 6 respectively, illustrates the impact of the near-field instabilities in both the droplet size distribution and in the droplet velocity distribution. The viscosity reduction decreased the Kelvin-Helmholtz effect while keeping similar DSD, which is caused by the increase in the droplet Re. The density reduction acted to enhance the instabilities near the inlet, inducing breakup much earlier than in the higher density cases.

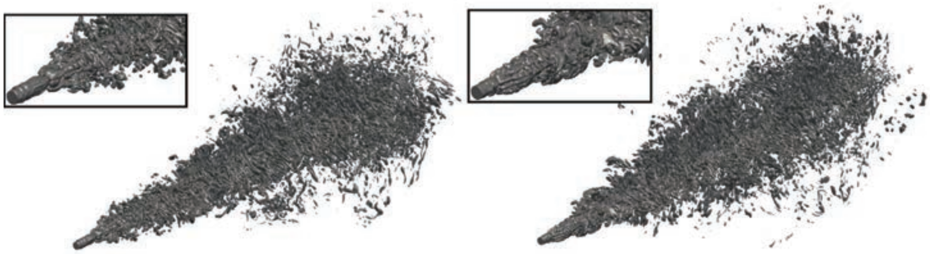

Figure 3: Effect of the viscosity ratio reduction in the snapshots of the isosurface alpha = 0.1 for cases 1 (left) and 6 (right). The details show the Kelvin-Helmholtz instabilities near the inlet.

Figure 4: Effect of the density ratio reduction in the snapshots of the isosurface alpha = 0.1 for cases 1 (left) and 8 (right). The details show the Kelvin-Helmholtz instabilities near the inlet.

4. Conclusion

Using the hybrid volume-of-fluid/Euler-Euler model with a large eddy simulation turbulent model was effective in modeling the laboratory-scale oil blowout in a water tank. The effects of different physical parameters on the atomization were complex and it shows the difficulty in understanding how the distribution of droplets are affected by this phenomenon.

Further research is necessary to create and overarching theory that could explain the jet breakup in a more clear and concise way.

References

[1] Kujawinski et al., Environ. Sci. Technol., vol. 45, pp. 1298-1306, 2011.

[2] Socolofsky et al., Oceanography, vol. 29, pp. 64-75, 2016.

[3] Wardle, Weller, International Journal of Chemical Engineering, vol. 13, 2013.

[4] Miyata et al., JIME, vol. 51, pp.367-374, 2016.