

## **Analysis of the Behavior of Subsea Gas Plume Resulting from a Two-Phase Flow Using Computational Fluid Dynamics**

Maria Fernanda Oliveira<sup>1</sup>, Sávio Souza Venâncio Vianna<sup>2</sup>

### **ABSTRACT**

Accidental subsea gas releases can pose a threat to people, equipment, and facilities since gas can be toxic or flammable at the concentrations in which the leak occurs. The accurate prediction of the behavior of the gas plume formed in the leaks can be fundamental to the development of techniques of accident prevention or, in some cases, remediation measures, avoiding the emergence of more serious consequences. Among the different ways to analyze the behavior of gas plumes formed under water, the Computational Fluid Dynamics (CFD) tool stands out for allowing the study of plume behavior to be done in a safer, simpler, and less expensive way, if compared to experimental studies. Inspired by the accidental release of subsea gas scenario, this work validated a CFD setup of a 2D two-phase air-water flow using the VOF method in Ansys Fluent. The use of the VOF method differs this work from other works that use a hybrid Eulerian-Lagrangian and Eulerian-Eulerian methodology to model such types of flow. In this validation, simulations with a 9 m base tank, and 7 m water depth and 0.050, 0.100, and 0.450 m<sup>3</sup>/s gas flow were performed. The simulated data were compared to experimental results available in Literature. After the validation of the setup, a study was carried out varying the gas flow from 0.0125 to 0.150 m<sup>3</sup>/s to verify how some plume characteristics such as rise time, fountain height and plume horizontal dispersion distance are affected by the changes. The relationship between the flow rates and the fountain heights after 15 s of flow was linear, whereas both initial fountain height and rise time followed a power trendline. Lastly, the plume horizontal dispersion distance for higher flow rates remained practically constant.

### **1. INTRODUCTION**

The accidental subsea gas releases consequences can pose a threat to human life, environment, and oil and gas exploitation facilities. Even with risks involved in the activity, and despite the search for renewable energy resources, it is impossible to deny the growth of activities related to oil and gas exploration over the years. Furthermore, although accidents related to the underwater release of gases continue to happen, which is evidenced by catastrophes such as the Macondo Oil Disaster, the Montara Oil Spill, and others, the understanding of the phenomenon of submerge gas releases and their consequences is still restricted [1]. The study of this phenomenon is, therefore, of paramount importance.

In the search for more information about the consequences of phenomena involving subsea release of gases, the use of experiments in true ocean conditions is still limited. Additionally, traditional integral models are not able to yield acceptable results on the surface behavior [2]. In this context, the Computational Fluid Dynamics (CFD) stands out for offering a possibility to carry out numerical experiments safely and at a relatively low cost, compared to most experimental studies.

The simulation tool has proven to be fundamental in the evolution of studies on subsea gas release, and different works tried to explain the gas plume behavior in water. Within the scope of computer simulation, different authors have studied the phenomenon of gas release in liquid media using hybrid Eulerian-Lagrangian and Eulerian-Eulerian approaches. Cloete et al. (2009) used a hybrid discrete phase model (DPM) and Volume of Fluid (VOF) to study air liberation in water media [2] and, in the following years, the author's works were enhanced [1, 3–8]. However, besides the good bubble behavior modeling, these hybrid models can be restricted to releases with low gas rates and incapable of predict the surface velocities adequately [9].

<sup>1</sup> MS, Chemical Engineer – University of Campinas

<sup>2</sup> PhD, Chemical Engineer – University of Campinas

An Eulerian-Eulerian approach to study underwater bubble plumes was used by Wu et al. (2017), who compared RANS and LES simulations to understand influence of the methods on the accuracy of the simulated results. Yet, the computational cost of the numerical experiments proposed by the authors were significantly higher than the DPM + VOF studies [9].

The need to correctly simulate the water-air interface for different flows is related to the desire to understand how the leaked gas reaches the surface and disperses. Characteristics such as the plume horizontal dispersion distance on the surface are extremely important for the continuation of studies to analyze the consequences of accidents with gas leaks, as they dictate how a new plume can form in the air, for example. This characterization and the understanding of the plume behavior, nonetheless, remains limited.

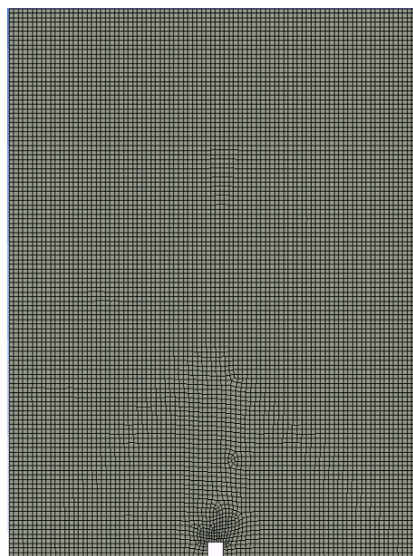
This study aims to present a low computational cost VOF two-phase flow setup capable of predict the behavior of the plume resulting from a gas release, especially on the air-water interface. With the resulting information of the performed simulations, this setup will allow to provide guidance for emergency response formulation in case of accidental subsea gas release scenario.

## 2. DESCRIPTION

### 2.1 Geometry and Mesh

According to the intention of reducing the computational cost of the studies, all simulations in this work were carried out in a 2D domain. The geometry consisted of a 9 m at the base and 12 m high, and the water column considered was 7 m, with the remainder of the domain being composed of air. At a height of 33 cm above the center of the domain's base, air was released from a 34 cm opening, representing the circular 3D area of the work presented by Cloete et al. (2009) [2]. The choice of a 2D domain allowed the simulations to be run with higher speed and without compromising the reliability of the results, when compared to a 3D case with a similar setup.

In addition to the 2D geometry, another alternative to reduce the computational cost of simulation was the use of adaptive mesh. The original mesh contained only 10831 cells. The adaptive criteria were the presence of an air volumetric fraction of 0.06 for refinement, and of 0.05 for coarsening. The criteria were evaluated every 10 iterations. Figure 1 shows the geometry with the initial mesh proposed for the simulations. Once the geometry and mesh were ready and reflecting the experimental arrangement, we started the modelling of the case as discussed in the following section.



**Figure 1** - Initial mesh

## 2.2 Setup proposal

The simulated cases presented in this work are based on the works presented in Literature [2, 10]. However, instead of using a hybrid discrete phase model (DPM) and Volume of Fluid (VOF), only the VOF model was applied. Therefore, the simulations were Eulerian-Eulerian.

The simulations were run in Ansys Fluent, version 20.2. The implicit VOF settings were completed with a constant 0.072 N/m surface tension between air and water, the two immiscible phases chosen for this study. The realizable k-epsilon turbulence model was selected among other tested models. This model is efficient in predicting the dispersion of gases in submerge jets [1].

The boundary conditions consisted in a 100% air velocity inlet through the 34 cm opening, with velocities that allowed equivalent 3D flow rates from 0.0125 to 0.150 m<sup>3</sup>/s; a 0 Pa pressure outlet, with 100% of air backflow at the top of the geometry; and Fluent's default wall settings for the other boundaries.

The simulations were carried out with a transient solver. For pressure-velocity coupling, PISO scheme was selected, and for pressure discretization, PRESTO! scheme was chosen. Second order upwind equations were used for continuity, momentum, and turbulence modeling.

Each 20 s simulation took around 16 hours to run on a 64 GB RAM computer, with 7 cores used.

## 2.3 Setup Validation

The intended setup, described in the previous section, was run in a computational domain like the experimental scheme available in Literature [10]. The same boundary conditions as those of the authors were also applied. Therefore, the proposed setup was run with the release rates of 0.050, 0.100, and 0.450 m<sup>3</sup>/s, at the state at the local of release. Considering the transient analysis performed, the parameters compared with the experimental study were the observed fountain heights, both initially and at 15 seconds of flow. The rise times of the plumes were compared with the data presented by Cloete et al. (2009) [2]. The 2D data of the 0.100 m<sup>3</sup>/s was also compared to a 3D setup equal to the proposed one. After the validation of the setup, a new set of cases were run, according to the next section.

## 2.4 Case Study: Varying the Air Flow Rates

After the setup was validated to the conditions proposed in this work, more simulations, with flow rates varying between 0.0125 and 0.150 m<sup>3</sup>/s, were carried out. The values of the rise times, initial fountain height, 15 s fountain height and fountain horizontal dispersion at initial height were measured. The observed results were discussed.

# 3. RESULTS

## 3.1 Setup Validation

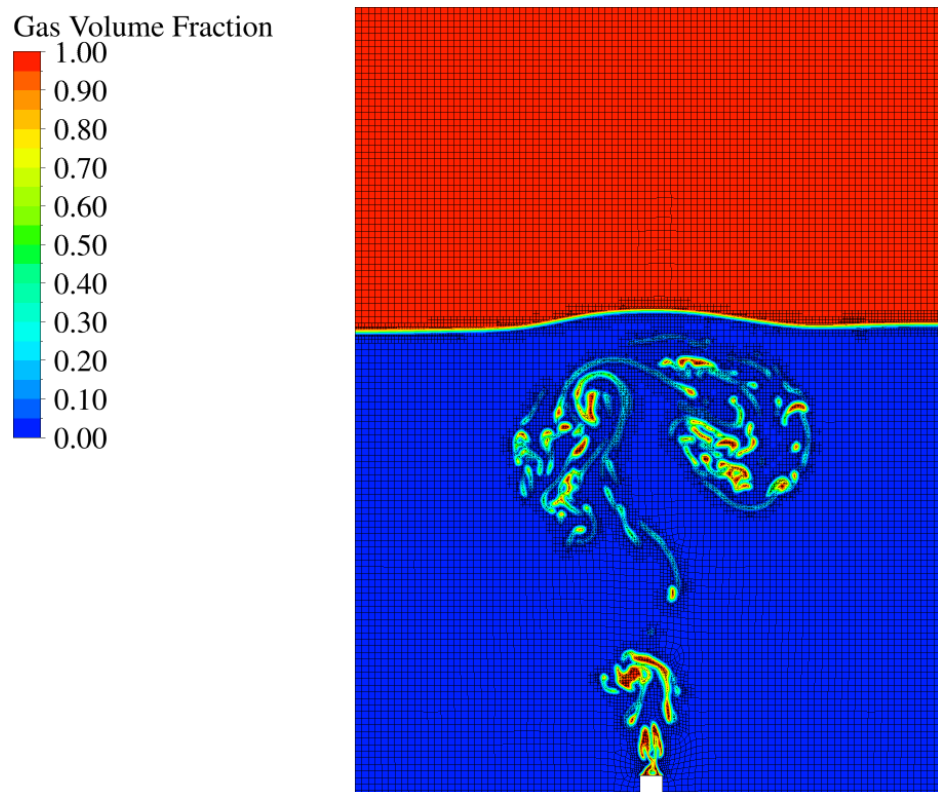
The rise times of the bubbles for the flow rates of 0.050, 0.100 and 0.450 m<sup>3</sup>/s were the first obtained parameters compared to the experimental results of Engebresten et al. (1997) [10], and the simulated data of Cloete et al. (2009) [2]. Table 1 summarizes the acquired data and the data available from the other two works.

The analysis of Table 1 shows that the setup of this work resulted in very precise rise times, if compared to the Experimental data. The results were also very similar to the ones available in Literature [2]. With the exactness of the presented results, the initial fountain heights, and the fountain heights at 15 s were compared with the same reference data.

Figure 2 brings the volume fraction of gas profile at 4.8 s for the flow rate of  $0.100 \text{ m}^3/\text{s}$ , which is the time when the air plume reaches the surface for such flow. It is possible to verify the elevation of the fountain height due to the momentum carried by the ascending air. We also note the gas recirculation due to drag forces between air and water, and a flow pattern close to the liberation point similar to a Kármán vortex street.

**Table 1** - Bubble rise times at flow rates of  $0.050$ ,  $0.100$  and  $0.450 \text{ m}^3/\text{s}$

Rise time (s)	Flow rate ( $\text{m}^3/\text{s}$ )		
	0.050	0.100	0.450
Simulation (This work)	6.0	4.8	3.0
Experimental data [10]	6.0	4.8	3.1
VOF + DPM simulation [2]	5.9	5.08	3.16



**Figure 2** - Volume fraction of gas for gas flow of  $0.100 \text{ m}^3/\text{s}$  at 4.8 s

Table 2 brings the comparison between the initial fountain heights obtained with the proposed 2D setup, the experimental results [10], and the DPM + VOF simulated results [2]. The flow rates analyzed were  $0.100$  and  $0.450 \text{ m}^3/\text{s}$  of air.

**Table 2** - Initial fountain heights obtained for 0.100 and 0.450 m<sup>3</sup>/s

	Flow rate (m <sup>3</sup> /s)	
Fountain height (m)	0.100	0.450
Simulation (This work)	0.33	0.69
Experimental data [10]	0.30	0.45
VOF + DPM simulation [2]	0.28	0.81

This parameter was used only for setup validation, since the initial fountain height a simple water splash, without significative representation of the flow consequences such as the fountain height after some time of flow [10]. Even so, it is important to observe that the proposed setup gave a valid result at 0.100 m<sup>3</sup>/s and, for 0.450 m<sup>3</sup>/s, a simulated result less discrepant from the experimental data, if compared to the simulated result available in Literature [2]. This better description of the fountain height was obtained because the valid flow rate range over which the Eulerian-Eulerian proposed setup is valid is wider than the Eulerian-Lagrangian schemes. The Eulerian-Lagrangian simulations assume that the simulated bubbles occupy no volume in computational domain, what also affects how the interactions between the bubbles occur.

Finally, the validation of the setup was concluded with the comparison between the simulated and the experimental fountain heights at 15 s for the flow rates of 0.100 and 0.450 m<sup>3</sup>/s. Table 3 summarizes the data.

**Table 3** - Fountain heights at 15 s obtained at flow rates of 0.100 and 0.450 m<sup>3</sup>/s

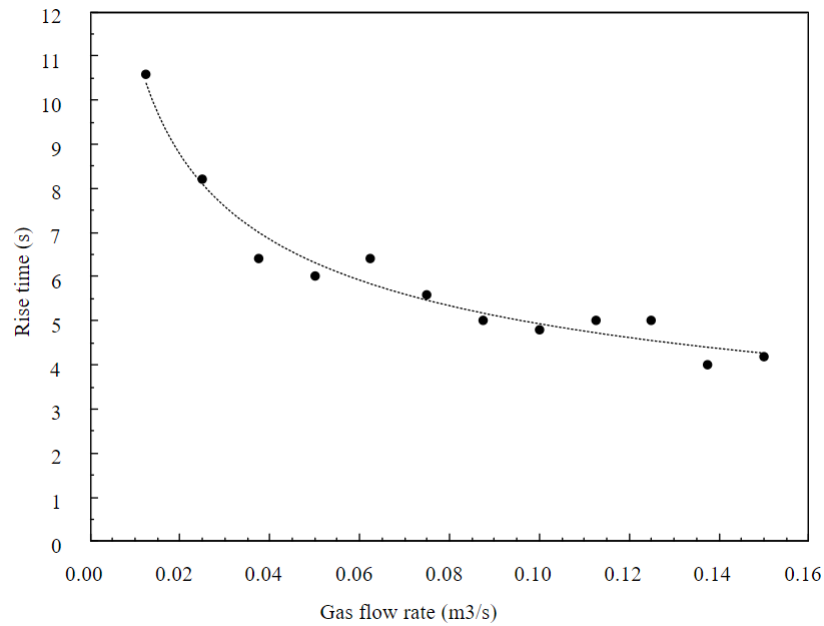
	Flow rate (m <sup>3</sup> /s)	
Fountain height (m)	0.100	0.450
Simulation (This work)	0.69	1.25
Experimental data [10]	0.65	1.25

Once again, the simulated data were quite in agreement with the experimental results. The correct prediction of parameters such as center velocities and fountain heights, the last being the case of this work, indicates that the vertical momentum carried by the plume is well estimated. In addition, the correct distribution of the plume's momentum to the vertical implies that the horizontal distribution also tends to be correct, what is related to the prediction of the horizontal spread of the plume.

Since the three compared parameters showed coherent results for the proposed 2D setup, it was considered valid for transient simulations. Then, it was repeated for other 10 flow rates, varying from 0.0125 m<sup>3</sup>/s to 0.150 m<sup>3</sup>/s. The results are displayed below.

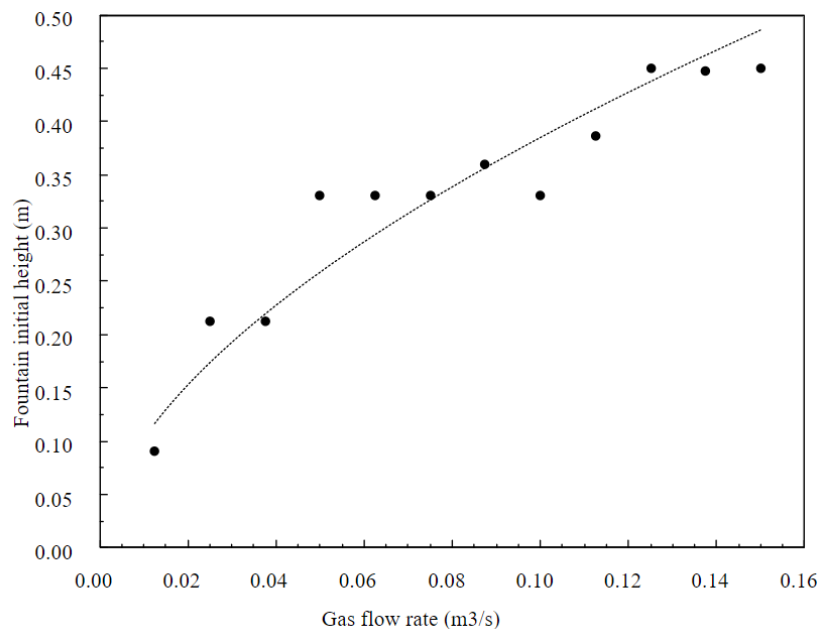
### 3.2 Studies Varying Flow Rate

Figure 3 presents the relation between gas flow rate and rise times. Although we expected a linear relationship, due to the relation between speed, distance, and time, Figure 3 shows a power trendline describing the data. At lower flow rates, the rise time varies more significantly, and the more the flow increases, the less the rise time changes. One possible explanation is the interactions between gas and liquid, such as the surface tension and drag force, which act over the plume and alter the movement of the fluids. At lower velocities, the interactions occur more intensely, what is evidenced by the larger quantity of wake regions.



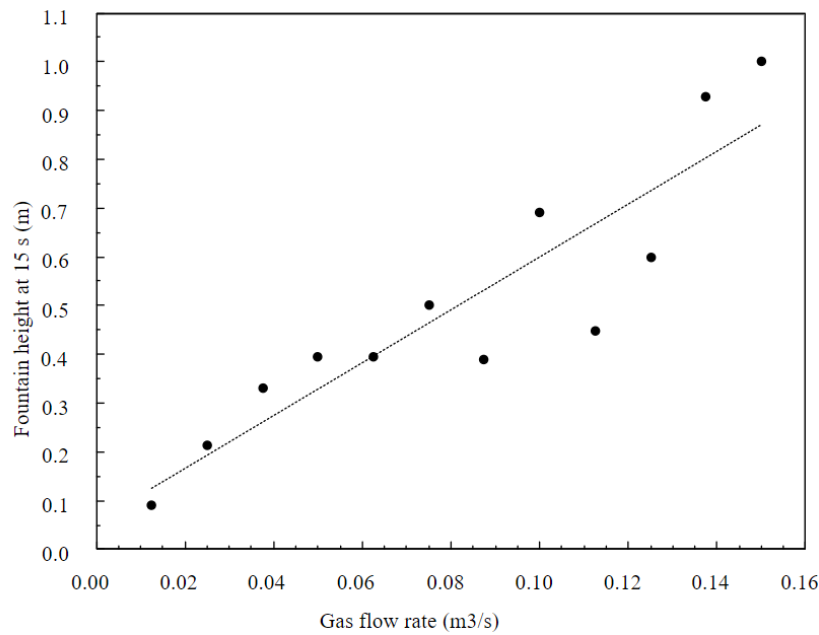
**Figure 3** - Plume rise times as function of different gas flow rates

The same power trendline character was observed for the initial fountain heights as function of the flow rate, as shown in Figure 4. Once again, it is evidenced the interaction between the water and the air plume, which is longer for lower air velocities.



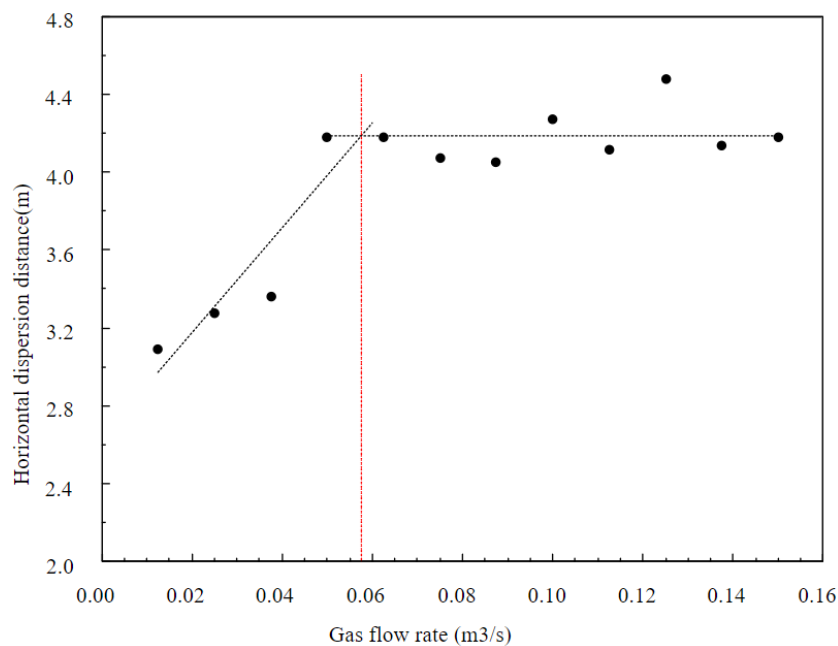
**Figure 4** - Fountain initial heights at different gas flow rates

In the range analyzed, the height of the plume at 15 s and the considered gas flow rates presented a linear relationship. It is possible that, as the flow rate continue to increase, a maximum plume height value will be reached, due to the action of forces such as gravity acceleration and surface tension between the fluids. The data obtained, shown in Figure 5, would, therefore, lose this linear character in this new range of points.



**Figure 5** - Fountain heights at 15 s of flow for different gas flow rates

The plumes' horizontal dispersion distances were measured at the rise times, since for longer times, it was observed the interaction between the plume and the domain walls. Figure 6 summarizes this data. We observed two different groups in the picture. Firstly, there is a region with three points where the distance slightly increases linearly with the increase of the flow rate. Secondly, there is a region where the distance varies around 4.19 m but can be considered practically constant. There is a critical flow rate around 0.058 m³/s which separates the regions with linear and constant behavior. The possible explanation for the graphic aspect after the critical flow rate is the rapid ascending of the plume controlled by forces such as gravity and surface tension, which limits the growth of the water fountain and, consequently, its width. Other possibility is also the interaction between the plumes with higher volume, consequence of higher flow rates, with the domain walls.



**Figure 6** - Fountain's width at different gas flow rates



## CONCLUSION

We have presented a legitimate and precise 2D setup capable of predict satisfactorily properties of gas plumes in water such as plume rise time, fountain heights at different flow times and plume horizontal dispersion. These precise predictions indicates that other parameters, such as plume's horizontal spread, can be also correctly calculated, since they are all related to the distribution of the momentum of the plume to the vertical and horizontal directions. Using the developed setup, we have evaluated the relation between these parameters and the gas flow rates varying from 0.0125 m<sup>3</sup>/s to 0.150 m<sup>3</sup>/s. It was observed that, for lower gas flow rates, the interaction between gas and liquid is extended, what reflects in higher rise times and initial fountain heights. These interactions are related to drag forces and surface tension and result in the recirculation of fluid verified in the flows. The relation between the 15 s plume heights and the flow rates was linear in the considered flow rates interval. Finally, the horizontal dispersion analysis showed that for flow rates equal or greater than 0.050 m<sup>3</sup>/s, the plume width was kept practically constant, probably due to interactions between the plume, the water, and the surface air, and with the domain walls. Future work will focus on study how the size of the leak affects the characteristics of the plumes considering gas leaks with the same Reynolds numbers as the ones studied in this work, as well as verify if the observed plume's behaviors can be observed at different flow conditions. The findings of such studies can be of paramount importance on the prediction of accident consequences with gas leaks.

## 5. REFERENCES:

- [1] Y. SUN, X. CAO, F. LIANG, and J. BIAN, "Investigation on Underwater Gas Leakage and Dispersion Behaviors Based on Coupled Eulerian-Lagrangian CFD Model," *Process Safety and Environmental Protection*, vol. 136, p. 268, (2020).
- [2] S. CLOETE, J. E. OLSEN, and P. SKJETNE, "CFD Modeling of Plume and Free Surface Behavior Resulting from a Sub-Sea Gas Release," *Applied Ocean Research*, vol. 31, no. 3, p. 220, (2009).
- [3] Q. Q. PAN, J. E. OLSEN, S. T. JOHANSEN, M. REED, and L. R. SÆTRAN, "CFD Study of Surface Flow and Gas Dispersion from a Subsea Gas Release," in *International Conference on Offshore Mechanics and Arctic Engineering – OMAE*, vol. 7, p. 1, (2014).
- [4] J. E. OLSEN and P. SKJETNE, "Modelling of Underwater Bubble Plumes and Gas Dissolution with an Eulerian-Lagrangian CFD Model," *Applied Ocean Research*, vol. 59, p. 193, (2016).
- [5] J. E. OLSEN, P. SKJETNE, and S. T. JOHANSEN, "VLES Turbulence Model for an Eulerian–Lagrangian Modeling Concept for Bubble Plumes," *Applied Mathematical Modelling*, vol. 44, p. 61, (2017).
- [6] X. LI, G. CHEN, R. ZHANG, H. ZHU, and C. XU, "Simulation and Assessment of Gas Dispersion above Sea from a Subsea Release: A CFD-Based Approach," *International Journal of Naval Architecture and Ocean Engineering*, vol. 11, no. 1, p. 353, (2019).
- [7] L. XINHONG, C. GUOMING, Z. RENREN, Z. HONGWEI, and F. JIANMIN, "Simulation and Assessment of Underwater Gas Release and Dispersion from Subsea Gas Pipelines Leak," *Process Safety and Environmental Protection*, vol. 119, p. 46, (2018).
- [8] J. E. OLSEN and P. SKJENE, "Summarizing an Eulerian-Lagrangian Model for Subsea Gas Release and Comparing Release Of CO<sub>2</sub> with CH<sub>4</sub>," *Applied Mathematical Modelling*, vol. 79, p. 672, (2020).
- [9] K. WU, S. CUNNINGHAM, S. SIVANDRAN, and J. GREEN, "Modelling Subsea Gas Releases and Resulting Gas Plumes Using Computational Fluid Dynamics," *Journal of Loss Prevention in the Process Industries*, vol. 49, p. 411, (2017).



- [10] T. ENGBRETSSEN, T. NORTHUG, K. SJOEN, and T. K. FANNELOP, “Surface Flow and Gas Dispersion from a Subsea Release of Natural Gas,” *Proceedings of the International Offshore and Polar Engineering Conference*, vol. 1, p. 566, (1997).