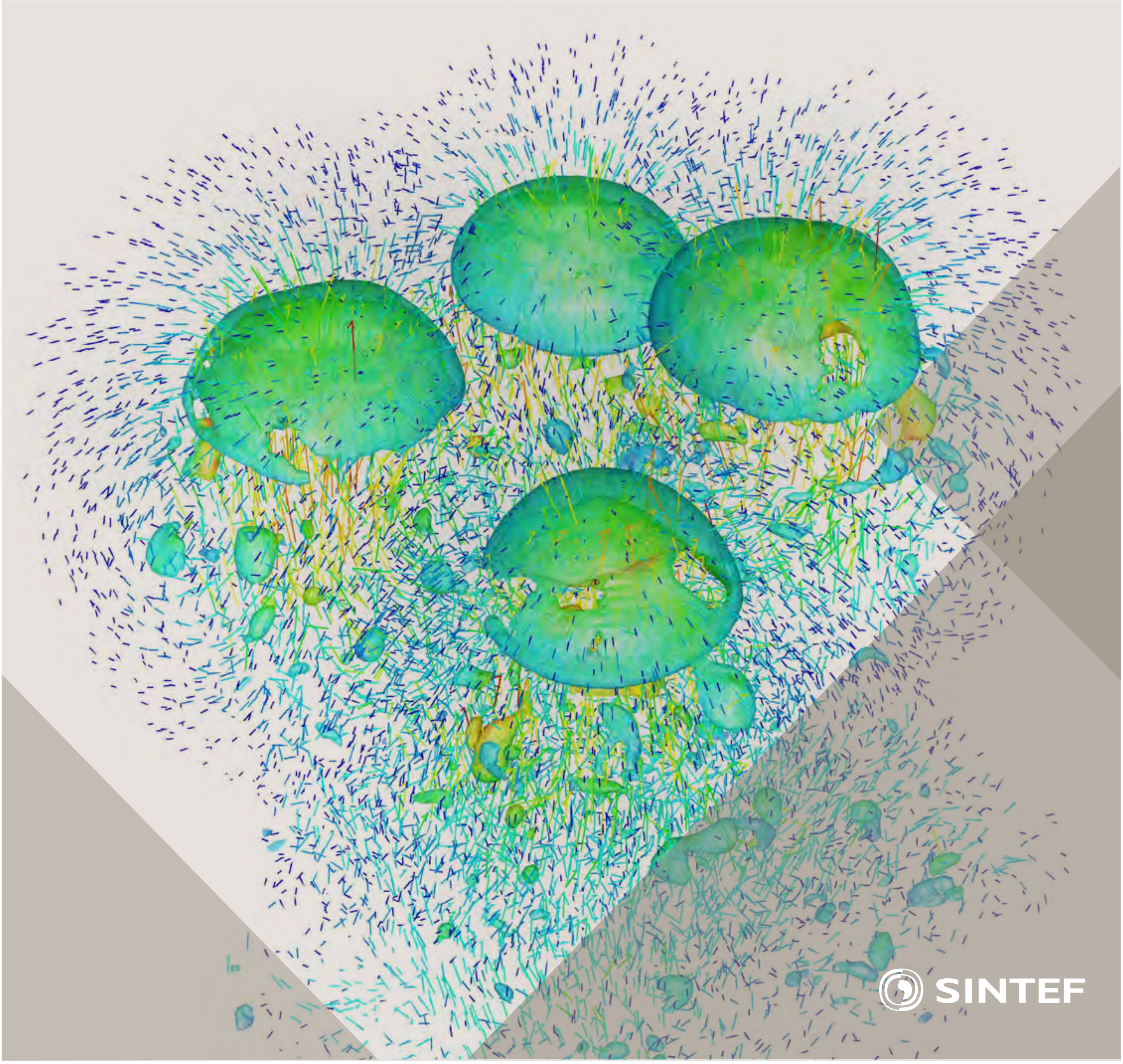
SINTEF
PROCEEDINGS
1

Selected papers from 10th International Conference on Computational Fluid Dynamics in the Oil & Gas, Metallurgical and Process Industries

Progress in Applied CFD



SINTEF

SINTEF Proceedings

Editors:
Jan Erik Olsen and Stein Tore Johansen

Progress in Applied CFD

Selected papers from 10th International Conference on Computational Fluid Dynamics in the Oil & Gas, Metallurgical and Process Industries

SINTEF Academic Press

SINTEF Proceedings no 1
Editors: Jan Erik Olsen and Stein Tore Johansen
Progress in Applied CFD
Selected papers from 10th International Conference on Computational Fluid
Dynamics in the Oil & Gas, Metallurgical and Process Industries

Key words:
CFD, Flow, Modelling

Cover, illustration: Rising bubbles by Schalk Cloete

ISSN 2387-4287 (printed)
ISSN 2387-4295 (online)
ISBN 978-82-536-1432-8 (printed)
ISBN 978-82-536-1433-5 (pdf)

60 copies printed by AIT AS e-dit
Content: 100 g munken polar
Cover: 240 g trucard

SINTEF Academic Press
Address: Forskningsveien 3 B
PO Box 124 Blindern
N-0314 OSLO
Tel: +47 22 96 55 55
Fax: +47 22 96 55 08

www.sintef.no/byggforsk
www.sintefbok.no

SINTEF Proceedings

SINTEF Proceedings is a serial publication for peer-reviewed conference proceedings on a variety of scientific topics.

The processes of peer-reviewing of papers published in SINTEF Proceedings are administered by the conference organizers and proceedings editors. Detailed procedures will vary according to custom and practice in each scientific community.

PREFACE

This book contains selected papers from the 10th International Conference on Computational Fluid Dynamics in the Oil & Gas, Metallurgical and Process Industries. The conference was hosted by SINTEF in Trondheim in June 2014 and is also known as CFD2014 for short. The conference series was initiated by CSIRO and Phil Schwarz in 1997. So far the conference has been alternating between CSIRO in Melbourne and SINTEF in Trondheim. The conferences focus on the application of CFD in the oil and gas industries, metal production, mineral processing, power generation, chemicals and other process industries. The papers in the conference proceedings and this book demonstrate the current progress in applied CFD.

The conference papers undergo a review process involving two experts. Only papers accepted by the reviewers are presented in the conference proceedings. More than 100 papers were presented at the conference. Of these papers, 27 were chosen for this book and reviewed once more before being approved. These are well received papers fitting the scope of the book which has a slightly more focused scope than the conference. As many other good papers were presented at the conference, the interested reader is also encouraged to study the proceedings of the conference.

The organizing committee would like to thank everyone who has helped with paper review, those who promoted the conference and all authors who have submitted scientific contributions. We are also grateful for the support from the conference sponsors: FACE (the multiphase flow assurance centre), Total, ANSYS, CD-Adapco, Ascomp, Statoil and Elkem.

Stein Tore Johansen & Jan Erik Olsen

Organizing committee:

Conference chairman: Prof. Stein Tore Johansen
Conference coordinator: Dr. Jan Erik Olsen
Dr. Kristian Etienne Einarsrud
Dr. Shahriar Amini
Dr. Ernst Meese
Dr. Paal Skjetne
Dr. Martin Larsson
Dr. Peter Witt, CSIRO

Scientific committee:

J.A.M.Kuipers, TU Eindhoven
Olivier Simonin, IMFT/INP Toulouse
Akio Tomiyama, Kobe University
Sanjoy Banerjee, City College of New York
Phil Schwarz, CSIRO
Harald Laux, Osram
Josip Zoric, SINTEF
Jos Derksen, University of Aberdeen
Dieter Bothe, TU Darmstadt
Dmitry Eskin, Schlumberger
Djamel Lakehal, ASCOMP
Pär Jonsson, KTH
Ruben Shulkes, Statoil
Chris Thompson, Cranfield University
Jinghai Li, Chinese Academy of Science
Stefan Pirker, Johannes Kepler Univ.
Bernhard Müller, NTNU
Stein Tore Johansen, SINTEF
Markus Braun, ANSYS

CONTENTS

10th International Conference on CFD in Oil & Gas, Metallurgical and Process Industries
SINTEF, Trondheim, Norway *CFD 2014*
17-19 June 2014

GOVERNING PHYSICS OF SHALLOW AND DEEP SUBSEA GAS RELEASE

Jan Erik OLSEN & Paal SKJETNE
SINTEF Materials and Chemistry, 7465 Trondheim, NORWAY
* E-mail: jan.e.olsen@sintef.no

ABSTRACT

A modelling concept for studying the resulting bubble plume from a subsea gas release is presented. Simulation results show good consistency with available experimental data. The modelling concept is applied to assess the importance of different physics and mechanisms assumed to influence the behaviour of the bubble plume. It is shown that buoyancy, drag, turbulent dispersion and gas dissolution are the governing mechanisms, and that gas dissolution is important for deep releases.

Keywords: subsea, gas release, Lagrangian, parcel, HSE, bubble plume, CFD

NOMENCLATURE

Greek Symbols

α	Volume fraction [ ]
ρ	Mass density [kg/m^3].
μ	Dynamic viscosity [kg/m s]

Latin Symbols

C_D	Drag coefficient[ ]
c	Concentration [kg/m^3]
d	Diameter [m]
$\boldsymbol{F}$	Force [N]
G	Drag correction [ ]
k	Mass transfer coefficient [m/s]
$\dot{m}$	Mass transfer rate [kg/s]
P	Pressure, [Pa]
$\boldsymbol{u}$	Velocity, [m/s]

Sub/superscripts

B	Buoyancy
b	Bubbles
D	Drag
i	Species index
L	Lift
l	Liquid
PG	Pressure gradient
sol	Solubility
TD	Turbuelent dispersion
VM	Virtual mass

INTRODUCTION

Subsea gas release is caused by well blowouts, pipeline failures and other, and poses a threat to the safety of people and assets operating offshore. In order to perform risk assessments it is important to understand the quantitative impact of the gas release. Since realistic experiments are prohibitively expensive and potentially dangerous, quantitative models have been identified as interesting research tools.

Traditional integral methods (Fanneløp and Sjøen 1980) provide a good representation of the rising bubble plume if the model coefficients are tuned properly. However, the method yields limited results for the surface characteristics, which is a limitation since this is where the plume will interact with offshore structures, floating installations and ships. Multiphase computational fluid dynamics (CFD) provides greater generality since it is more fundamental and can, in principle, provide information on both the bubble plume and the surface behaviour. The computational cost of such models is significantly higher than the traditional integral models. However, it has been demonstrated that a 3D transient multiphase CFD model can be applied to the study of the ocean plume and the free surface behaviour (Cloete, Olsen et al. 2009).

A variety of forces and mechanisms influences the rising bubble plume. The significance of these mechanisms varies with gas rate and release depth. Some can be neglected and some must be accounted for. The study presented in this paper assesses the importance of the different mechanisms, and clarifies the governing physics of subsea gas release. Instinctively there is a suspicion that the governing physics might be different in a shallow and deep release. It is difficult to clearly define a shallow and deep release. Here it is linked to the length scales typical for offshore operations. We have chosen 30 meters to represent a *shallow* release and 300 meters to represent a *deep* release.

MODEL DESCRIPTION

An Eulerian-Lagrangian modelling concept has been developed to study subsea gas release. It is based on an Eulerian mixture model with interface tracking of the ocean surface separating the ocean and atmosphere which constitutes two Eulerian phases, and Lagrangian tracking of the dispersed bubbles in the ocean. The Eulerian method with interface tracking is a VOF (volume of fluid) method, and the Lagrangian tracking method is a discrete phase model, i.e DPM. This is also known as a coupled DPM-VOF model (Cloete, Olsen et al. 2009).

The discrete phase model tracks the bubbles as parcels. Each parcel may consist of several bubbles. All bubbles within the same parcel share the same properties, i.e. equal density, diameter and more. This reduces the computational cost considerably since billions of bubbles can be represented by a reasonable amount of parcels. Without this feature, it would not have been feasible to study subsea gas release with Lagrangian bubble tracking. The bubble motion is governed by Newton's second law of motion stating that bubble acceleration equals the sum of all forces acting on the bubbles:

$$\frac{d\boldsymbol{u}_b}{dt} = \boldsymbol{F}_B + \boldsymbol{F}_D + \boldsymbol{F}_L + \boldsymbol{F}_{VM} + \boldsymbol{F}_{PG} + \boldsymbol{F}_{TD} \qquad (1)$$

Here we have listed contributions from buoyancy ($\boldsymbol{F}_B$), drag ($\boldsymbol{F}_D$), lift ($\boldsymbol{F}_L$), virtual mass ($\boldsymbol{F}_{VM}$), pressure gradient ($\boldsymbol{F}_{PG}$) and turbulent dispersion ($\boldsymbol{F}_{TD}$). These are the forces known to influence bubbles in a bubble plume. Note that these forces are normalized with bubble mass. In addition mass transfer due to dissolution of gas into the ocean is believed to have an important effect on the fate of bubbles resulting from the gas release. Gas dissolution is accounted for by the following expression for mass transfer rate

$$\dot{m}_i = -\pi\, d_b^2\, k_i \left(c_i^{sol} - c_i^l\right) \qquad (2)$$

where d_b is bubble diameter, k_i is mass transfer coefficient, c_i^{sol} is solubility concentration and c_i^l is bulk concentration of species i. We will limit this study to release of methane. The solubility data for methane of Lekvam & Bishnoi (1997) and the expression for mass transfer coefficient of Zhang & Zu (2003) are applied. The bubble size is modelled by a transport equation which is governed by turbulence (break-up and coalescence) (Cloete, Olsen et al. 2009).

The continuous phases , i.e. water and atmosphere, are mathematically described by the VOF model as mentioned above. Their motion is coupled to the bubble motion through the drag force which is implemented as an exchange term in the momentum equations. A standard k-ε model was initially assigned to the modelling concept (Cloete, Olsen et al. 2009). As in many implementations of the k-ε model the interphase between water and atmosphere is not recognized as boundary. In reality the ocean surface dampens turbulence since eddies can not be sustained over this interface. Due to this an enhanced implementation of the k-ε model has been developed. It includes a source term in the ε-equation which dampens turbulence at the surface and an additional source term in both equations due to the added buoyancy of density variations of a gas over large pressure variations (Pan, Johansen et al. 2013).

The modelling concept is implemented in the commercial software ANSYS/Fluent 14.5 via several user defined functions. The model has been compared against data from a series of controlled experiments where gas was released in a rectangular basin with a depth of 7 meters and a surface area of 6 x 9 meters (Engebretsen, Northug et al. 1997). Air was released at the bottom at 3 different gas rates, of which the middle gas rate of 170 Nl/sec had the most complete set of results presented. Thus we have compared results from model simulations and experiments at this gas rate.

In Figure 1 we see the water velocity close to the water surface as a function of height above basin floor. Results from simulation performed with the standard and enhanced k-ε model are seen. Close to the surface the enhanced model has significantly better consistency with the experimental results than the standard model. Further down into the water, there is less discrepancy between the models. This is as expected since the enhancement primarily applies to the surface region. Figure 2 shows the velocity profiles at different elevations above the basin floor for both modelling and experimental results. Only data for the enhanced k-ε model are shown since the standard model produced almost equal results. We see a good agreement between model and experiments for the velocity profiles at the lower elevations (1.75 and 3.80 m). There is some discrepancy at the highest elevation (5.88 m). This is believed to be caused by an error in the flow measurements. Höntzsch turbine flow meters were applied. They are optimized for mono-directional flow, but for the bending flow close to the surface they will overpredict the flow velocity.

In these release scenarios gas dissolution has no effect due to the shallow depth and short residence time of the bubbles. The implemented model for mass transfer and gas dissolution have however been validated against experiments with good consistency in a separate study (Skjetne and Olsen 2012).

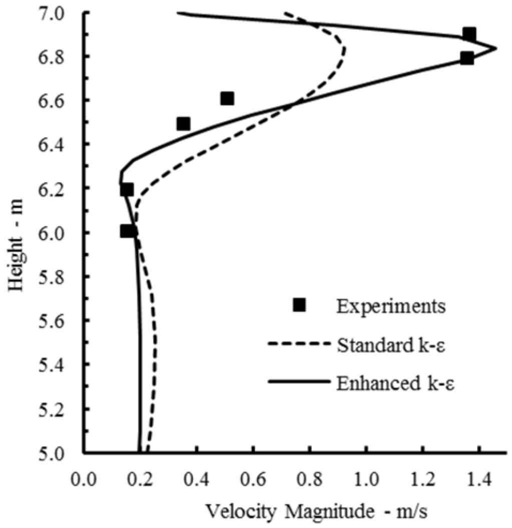


Figure 1: Velocity magnitudes near water surface for a gas rate of 170 Nl/s as a function of height above basin floor at a location 1.75 meters from plume centre. Models are compared with experiments.

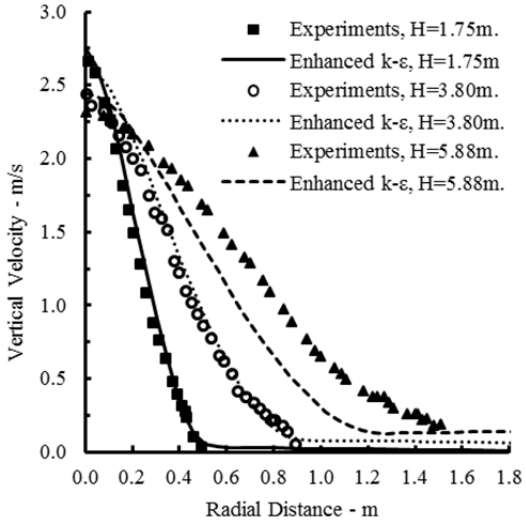


Figure 2: Plume velocity at 3 different elevations for a gas rate of 170 Nl/s. Model and experiments are compared.

SENSITIVITY ASSESSMENT

In the following we present results on sensitivity analyses on different physics assumed to affect the behaviour of a subsea gas release. Since buoyancy is the driving force of the plume, it always has to be accounted for. Drag forces are responsible for the coupling between the dispersed bubble phase and the continuous water phase and should thus not be neglected. The lift force has previously been shown to have a negligible effect on the bubble plumes associated with a subsea gas release (Olsen and Cloete 2009). Thus sensitivity to buoyancy, drag and lift will not be considered here. The effect of other forces and mechanisms are assessed in the following. In the assessment we will primarily focus on the vertical velocity along the centreline from ocean floor to ocean surface (i.e. plume axis).

Gas Expansion

Gas density is a function of pressure and temperature. The density increases with increasing pressure (i.e. depth). Thus the gas expands as the bubbles rise towards the surface. For low pressure this can be expressed by the ideal gas law, but the true density variation is more complex. Here we have applied the ideal gas law due its simplicity. In reality the true density deviates from the ideal gas laws at greater depths, but for a sensitivity analysis we assume that it is a valid assumption.

The effect of gas expansion can be assessed by comparing simulation results between expanding gas and gas with constant density. However, it is difficult to know which constant density to compare against. Here we have compared against bottom density (maximum), top density (minimum) and average density. In Figure 3 we see the effect of different density specifications on the vertical liquid velocity along the plume axis for a release from 30 meter at 10 kg/s. The results are taken after quasi steady state is reached. The height above the ocean floor (y-axis) is normalized with respect to ocean depth. This might exceed the value of 1 if the release is strong enough to sustain a fountain. We see that the velocity based on minimum and maximum densities deviate significantly from the velocity based on density from the ideal gas law. The average density is on average equivalent to the ideal gas law. However, it will give bad estimates of velocities close to the release zone. Note how the constant density approximations seem to decelerate the flow compared to an expanding gas. This tells us that gas expansion has an accelerating effect on the flow which is explained by the increasing buoyancy as gas density decreases during the ascent of the bubbles. In Figure 4 we see the same comparison for a release from 300 meter at 100 kg/s. The trend is the same as for the release from 30 meter, but not as pronounced. There are two reasons for a less pronounced effect of gas expansion. First there is significantly more gas dissolution at higher gas rates leaving less gas to expand. Secondly most gas expansion occurs close to the surface, and the surface region is less dominating in a deep release compared to a shallow release. Still the rise velocity is significantly affected by gas expansion. In addition other indicators such as total gas dissolution, plume spreading and more is affected when neglecting gas expansion (not shown here). Thus gas expansion needs to be accounted for.

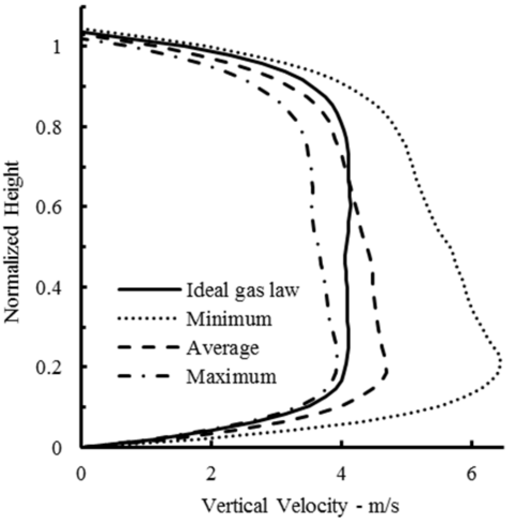


Figure 3: Effect of gas density on vertical velocity for a release from 30m at 10 kg/s.

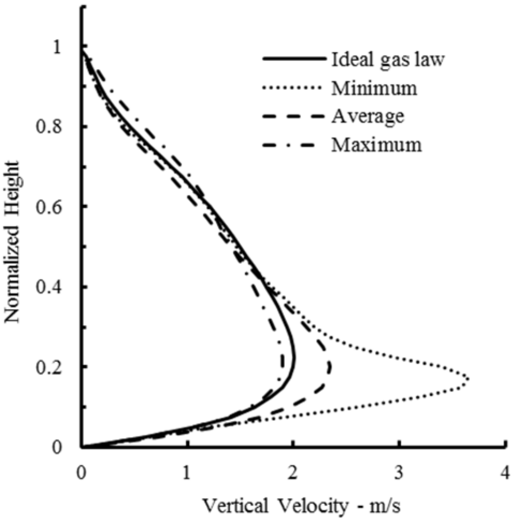


Figure 4: Effect of gas density on vertical velocity for a release from 300m at 100 kg/s.

Turbulent Dispersion

Turbulent dispersion is dispersion of bubbles, droplets and particles due to turbulence. In principle it is a drag force based on the fluctuating contribution to the instantaneous velocity. The standard drag force only accounts for the mean contribution, whereas the particle in reality is exposed to the instantaneous velocity. There are several models describing turbulent dispersion. We apply the random walk model (Gosman and Ioannides 1983) which is frequently used in Lagrangian tracking.

In Figure 5 we see how the vertical velocity varies from ocean floor to surface along the vertical centreline from the release point. Figure 5 shows results from simulations were turbulent dispersion has been neglected and accounted for. The results clearly demonstrate that there is a significant effect of the turbulent dispersion. Turbulent dispersion yields a lower vertical velocity due to lateral dispersion of bubbles. This is also illustrated in Figure 6 where we see that turbulent dispersion is responsible for the widening of the plume and hence the so-called *plume angle*. It is quite clear that turbulent dispersion can not be neglected.

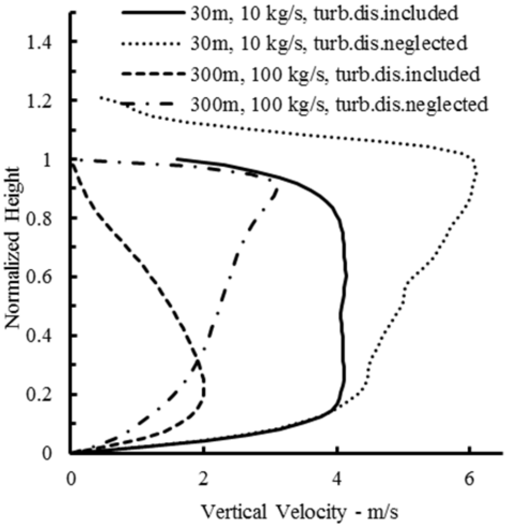


Figure 5: Effect of turbulent dispersion on vertical velocity.

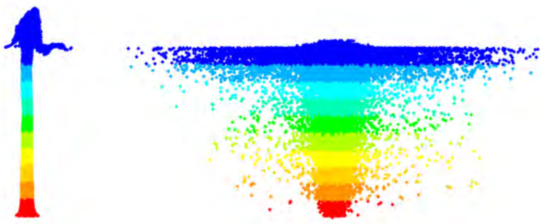


Figure 6: Plume shape coloured by gas density for simulation with neglected (left) and included (right) turbulent dispersion.

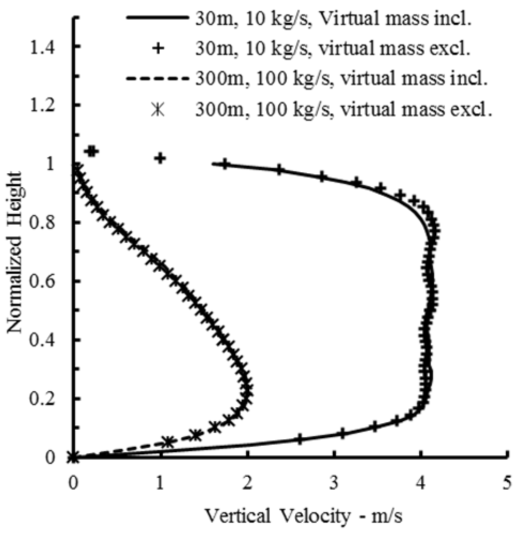


Figure 7: Effect of virtual mass on vertical velocity.

Virtual Mass

The virtual mass force is the force required to accelerate the fluid surrounding the particle. It is expressed by

$$\boldsymbol{F}_{VM} = \frac{1}{2}\frac{\rho_l}{\rho_b}\left(\frac{D\boldsymbol{u}_l}{Dt} - \frac{d\boldsymbol{u}_b}{dt}\right) \qquad (3)$$

Simulations including and neglecting the virtual mass force have been performed. The resulting vertical velocity along the plume axis is shown in Figure 7 for both shallow and deep release. We see that the virtual mass force has very little influence on the results, and it could be neglected.

Pressure Gradient Force

The pressure gradient force is the hydrodynamic force acting on the bubbles due to the pressure gradient in the surrounding liquid. Mathematically it is expressed by

$$\boldsymbol{F}_{PG} = \frac{\rho_l}{\rho_b}\boldsymbol{u}_b \nabla \boldsymbol{u}_l \qquad (4)$$

Results from simulations including and excluding the pressure gradient force are seen in Figure 8 and Figure 9. They show that the effect of the pressure gradient force can be neglected with respect to the vertical velocity along the plume centre axis.

The pressure gradient force could in principle have a more pronounced effect closer to surface as its nature typically affects a bending flow which is present at the surface. The horizontal velocity profile along the ocean surface was thus also assessed. The effect of the pressure gradient force was not detectable.

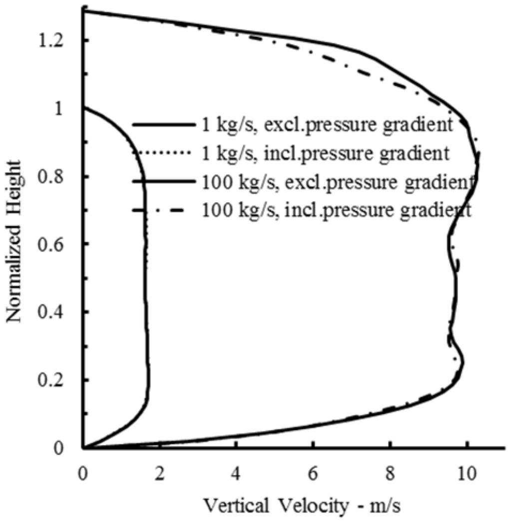


Figure 8: Effect of pressure gradient force on vertical velocity for release from 30 meter.

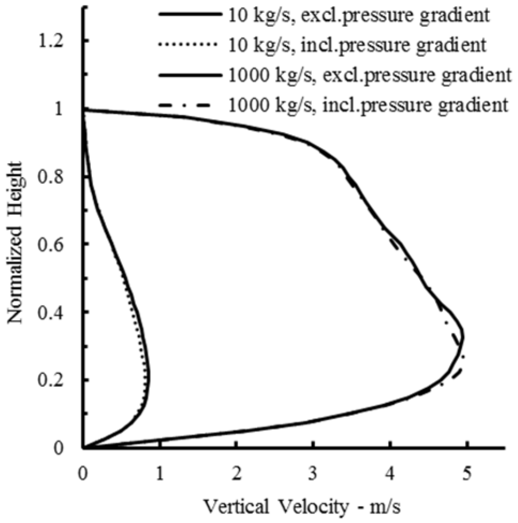


Figure 9: Effect of pressure gradient force on vertical velocity for release from 300 meter.

Gas Dissolution

Gas dissolution transfers mass from bubbles to the ocean and reduces the buoyant potential of the bubble plume. Thus gas dissolution will decrease the amount of gas reaching the surface and reduce the vertical velocity. Gas dissolution increases with residence time and is thus expected to have a greater impact on a deep release than on a shallow release. According to Eq.(2) will also bubble size have a significant impact on gas dissolution. This is illustrated by Figure 10. The figure shows the lifetime of a methane bubble exposed to gas dissolution at a depth of 30 or 300 meters as a function of bubble size. The data is based on the assumptions of single bubbles with a slip velocity of 0.3 m/s and thus represents a minimum lifetime. There is some variation with depth due to variations in solubility and methane density. We see that the lifetime varies significantly with bubble size. A bubble with a diameter of 1mm has a typical lifetime of 1 minute, whereas a bubble with a diameter of 5 mm can survive for 7-8 minutes. Thus estimating the bubble size accurately is vital for estimating the gas dissolution accurately.

Simulations with gas releases from 30 and 300 meter show that gas dissolution is significant when gas is released from 300 meters and almost neglectable when gas is released from 30 meters. This is indicated by the simulation results shown in Table 1.The table show that the rise time (i.e. time for first gas to reach the surface) correlates strongly with the release depth and to some extent with the release rate which was also shown by Bettelini and Fanneløp (1993). The rise time is affected by the inertia of the water column which needs to be accelerated by the first gas. The bubbles following the first gas will thus travel faster to the surface and obtain a lower residence time than the first bubbles. The mean residence time when a quasi-steady state is reached is shown in Table 1. It confirms that the quasi-steady state residence time is shorter than the rise time, and that the residence time and rise time has a similar dependence on release depth and release rate.

Since gas dissolution is a transient process, the amount of dissolution obviously depends on the residence time. This is supported by the data in Table 1 which is illustrated by Figure 11. We see that gas dissolution increases with residence time, but naturally levels out when there is no more gas to dissolve. Due to this correlation, there is a big difference in gas dissolution from a shallow and deep release.

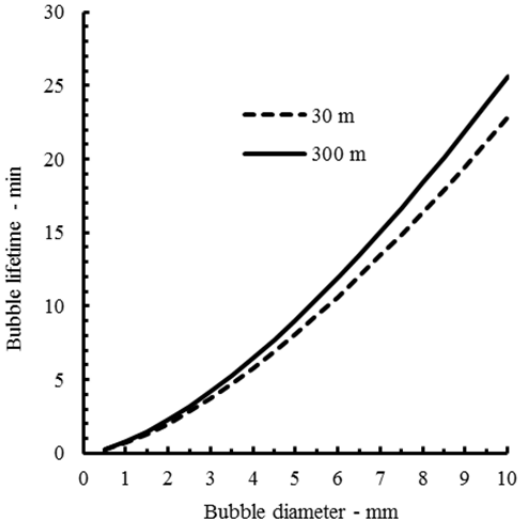


Figure 10: Effect of gas dissolution on vertical velocity for release from 30 meter.

Table 1: Global parameters for shallow and deep gas release.

	30 m.		300 m.	
	1 kg/s	10 kg/s	10 kg/s	100 kg/s
Rise time - s	18.7	8.1	502.3	169.3
Mean residence time - s	18.2	9.6	369.8	170.1
Surface flux - kg/s	0.93	9.97	0.12	19.2
Relative gas dissolution - %	6.7	0.3	98.8	80.8
Surface radius - m	16.9	18.2	69.8	128.5

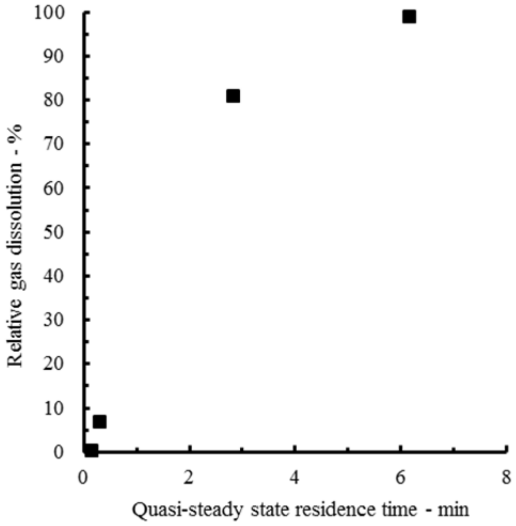


Figure 11: Correlation between residence time and gas dissolution.

CONCLUSION

An assessment of relevant physics and mechanism governing the fate of shallow and deep subsea gas releases have been conducted by mathematical simulations with an Eulerian-Lagrangian transient 3D modelling concept as described above. Buoyancy is the driving force of the gas release, while drag is the interaction force between gas bubbles and water which sets the water in motion. Since these are governing mechanisms, they have to be accounted, and thus their importance has not been assessed in this assessment. The assessment was carried out on releases from 30 meters and 300 meters and it shows that gas expansion and turbulent dispersion are also of great importance. These effects can not be neglected. Exemptions can only be made for very shallow releases (depth < 2m) were gas expansion can be neglected and for releases with very low gas rates where turbulence and turbulent dispersion has no effect. The assessment shows that the pressure gradient force and virtual mass force are insignificant. Previous studies also show that lift forces have little effect.

Gas dissolution is important for bubbles residing sufficiently long in the water column. Residence time increases with release depth and decreases with gas rate. The effect of gas dissolution also strongly depends on bubble size. Thus the effect of gas dissolution cannot be assessed by residence time alone. The simulation examples shown in this paper support that gas dissolution is significant for a deep release (300 meter) whereas the effect is small in a shallow release (30 meter).

ACKNOWLEDGEMENT

This study was performed in the SURE joint industry project funded by BP, Gassco, Shell, Statoil, Total, DNV LG, Safetec, Wild Well Control and PSA (Petroleum Safety Authorities).

REFERENCES

Bettelini, M. S. G. and T. K. Fanneløp (1993). "Underwater plume from an instantaneously started source." Applied Ocean Research **15**: 195-206.

Cloete, S., J. E. Olsen and P. Skjetne (2009). "CFD modeling of plume and free surface behavior resulting from a sub-sea gas release." Applied Ocean Research **31**(3): 220-225.

Engebretsen, T., T. Northug, K. Sjøen and T. K. Fanneløp (1997). Surface Flow and Gas Dispersion from a Subsea Release of Natural Gas. 7th int. offshore and polar engineering conference, Honolulu, International Society of Offshore and Polar Engineers.

Fanneløp, T. K. and K. Sjøen (1980). "Hydrodynamics of Underwater Blowouts." AIAA(Paper No. 80-0219).

Gosman, A. D. and E. Ioannides (1983). "Aspects of computer simulation of liquid-fuelled combustors." J.Energy **7**: 482-490.

Lekvam, K. and P. R. Bishnoi (1997). "Dissolution of methane in water at low temperatures and intermediate pressures." Fluid Phase Equilibria **131**(1-2): 297-309.

Olsen, J. E. and S. Cloete (2009). Coupled DPM and VOF model for analyses of gas stirred ladles at higher gas rates. Seventh International Conference on CFD in the Minerals and Process Industries, Melbourne, CSIRO.

Pan, Q. Q., S. T. Johansen, L. Sætran and M. Reed (2013). An enhanced k-epsilon model for bubble plumes. 8th International Conference on Multiphase Flow (ICMF), Jeju, Korea.

Skjetne, P. and J. E. Olsen (2012). "A parcel based modelling concept for studying subsea gas release and the effect of gas dissolution." Progress in Computational Fluid Dynamics **12**(2-3): 187-195.

Zhang, Y. and Z. Xu (2003). "Kinetics of convective crystal dissolution and melting, with applications to methane hydrate dissolution and dissociation in seawater." Earth and Planetary Science Letters **213**: 133-148.