MARY KAY O'CONNOR PROCESS SAFETY CENTER
TEXAS A&M ENGINEERING EXPERIMENT STATION

19th Annual International Symposium
October 25-27, 2016 • College Station, Texas

The Impact of Subsea Gas Releases and Resulting Gas Plumes Using Computational Fluid Dynamics

K. Wu†, S. Cunningham, S. Sivandran, and J. Green
BMT Fluid Mechanics
67 Stanton Avenue, Teddington, UK. TW11 0JY

† Presenter E-mail: kwu@bmtfm.com

Abstract

A Computational Fluid Dynamics (CFD) model was developed to describe the behavior of a subsea gas release and the subsequent rising gas plume. Four numerical approaches were assessed for their suitability to capture the characteristic behaviors in a rising gas plume by comparing the CFD results with experimental data obtained from an underwater gas release experiment carried out in a 10 m depth towing tank basin.

The k-ε turbulence model was found to be unsatisfactory in capturing random wandering behavior of the subsea gas plume due to the inherent Reynolds-Averaged Navier-Stokes (RANS) nature of the approach. The result is an over-prediction of the plume central line velocity and an under-prediction of the plume width as there was no mechanism to distribute and dissipate the high momentum gained during the initial gas release phase. The results obtained using the Large Eddy Simulation (LES) approach show the inherently random wandering behavior of the plume is successfully captured and both the centerline velocity and the velocity profile are in much better agreement with the experimental data.

Introduction

Vessels and platforms that operate in offshore fields are at risk of subsea gas releases resulting from accidental loss of containment caused by well blowouts or ruptured pipelines.

A subsea gas plume can be divided into three main zones based on the flow characteristics. The first is the zone of establishment located near the release source where the plume is dominated by high speed flow caused by the momentum gained through the high gas well pressure. The second zone is located above the zone of establishment and is known as the zone of established flow. Here the gas plume is dominated by buoyancy and the turbulent dispersion force. The last zone is where the significance of the water surface should be considered and is known as the surface zone.

Previously, most subsea gas releases were studied using an integral modelling method. This method can only predict gas plume behavior once it has reached the water surface and attained a "steady-state" condition. The method is not suitable to predict the transient features of the rising gas plume and additionally does not yield any information on the plume-water surface interaction in the surface zone. This is a major limitation since the surface zone is where the plume will interact with the offshore platforms and vessels.

Computational Fluid Dynamics (CFD) is a more detailed methodology for studying the gas plume since the method explicitly solves for the equations describing the fluid flow and therefore is able to provide information on both the bubble plume and the surface behavior.

Cloete [1] has developed a CFD model to describe the rising gas plume and validation was carried out against experimental data [2]. Although the results are in acceptable agreement with the experiment, there are a few drawbacks and limitations to the model. The main limitations are the application of the model to releases with low gas rates and an under-prediction of surface velocities due to the incomplete turbulence modelling near the free surface.

Pan [3] further developed Cloete's [1] model by addressing the issues mentioned above. However, neither Pan's [3] nor Cloete's [1] model were able to capture the transient random wandering behavior of the rising gas plume. Furthermore, the accuracy of both models is highly dependent on the correct calibration of the modelling constants against experimental data.

The objective of this paper is to create a CFD model which can take into account the various physical phenomena present in a rising gas plume within an aqueous environment.

Methodology

Governing Equations

The CFD model presented in this paper uses the Multiple Size Group Model (MUSIG). The model is based on an Eulerian–Eulerian population balance method where the bubble phase is modelled in an Eulerian approach rather than the particle based Lagrangian method used by Cloete [1] and Pan [7]. Since the water and the bubbles are both modelled in an Eulerian frame, implementation of advanced physics models and turbulence effects are significantly simplified.

The MUSIG model introduces a scalar field, α_g, which signifies the volume fraction of the bubble phase. In order to account for differences in behavior of bubbles of different sizes, several size groups are defined. The i^{th} group is assigned a fraction f_i of the total dispersed phase volume (i.e., $\sum_i f_i = 1$).

The current study utilizes the homogeneous MUSIG model, where the continuity equations for each bubble size group are defined, but only one set of momentum conservation equations is defined for all groups, as it is assumed bubbles of all sizes move with the same velocity at any given location in the domain. This assumption is supported by experimental evidence [4] which suggests the average bubble size in the center of the plume is of the order of 1 cm, which corresponds to the elliptic flow regime, so bubble slip velocity is not changing significantly with bubble volume.

Mass balances in the liquid and gas phases are represented by equations (1) and (2) respectively.

$$\frac{\partial}{\partial t}(\rho_l \alpha_l) + \nabla \cdot (\rho_l \alpha_l \boldsymbol{u}_l) = 0 \quad (1)$$

$$\frac{\partial}{\partial t}(\rho_g \alpha_g f_i) + \nabla \cdot (\rho_g \alpha_g \boldsymbol{u}_g f_i) = S_i \quad (2)$$

Where S_i is a source term due to coalescence and the breakup of the gaseous phase.

Momentum balances in the liquid and gas phases are represented by equations (3) and (4) respectively.

$$\frac{\partial}{\partial t}(\rho_l \alpha_l \boldsymbol{u}_l) + \nabla \cdot (\rho_l \alpha_l \boldsymbol{u}_l \boldsymbol{u}_l) = -\alpha_l \nabla \mathrm{p} + \alpha_l \rho_l \mathbf{g} - \boldsymbol{F}_{gl} + \alpha_l (\mu_l + \mu_t) \nabla^2 \mathbf{u} \quad (3)$$

$$\frac{\partial}{\partial t}(\rho_g \alpha_g \boldsymbol{u}_g) + \nabla \cdot (\rho_g \alpha_g \boldsymbol{u}_g \boldsymbol{u}_g) = -\alpha_g \nabla \mathrm{p} + \alpha_g \rho_g \mathbf{g} + \boldsymbol{F}_{gl} + \alpha_g (\mu_g + \mu_t) \nabla^2 \mathbf{u} \quad (4)$$

Where the eddy viscosity μ_t is taken from the turbulence model and $\boldsymbol{F}_{gl}$ is the force exerted by the liquid phase on the gas phase. The model also accounts for compressibility of the gas phase.

Air bubbles within the gas plume were modelled as ideal gases. Viscosity and surface tension values were therefore functions of temperature.

The following sub-models were used in conjunction with the MUSIG formulation.

As the gas plume rises through the water column the flow behavior is dominated by the drag force acting on the individual gas bubbles within the zone of established flow. A modified Grace drag model [5] with a physical volume fraction correction parameter was used to account for the drag force as defined below:

$$C_D = r_c^p C_{D\infty} \quad (5)$$

Where $C_{D\infty}$ is the drag coefficient for flow passing a single bubble given by the standard Grace drag model:

$$C_{D\infty} = \frac{4}{3} \frac{g d_p}{{U_T}^2} \frac{\Delta\rho}{\rho_c} \quad (6)$$

In equation (6) U_T is the terminal velocity and is a function of both the Etövös number and the Morton number. The volume fraction correction coefficient, p, in equation (5) is a function of air bubble size. The correction coefficient takes a value of -1 for air bubbles with diameters less than 2 mm and a value of 4 for air bubbles larger than 20 mm [5]. A linear relationship was assumed between the two values for air bubbles with diameters between 2 mm and 20 mm.

The lift forces experienced by the rising gas bubbles are described by the Tomiyama model and the lift coefficient, C_L, is given by equation (7):

$$C_L = \begin{cases} \min\left[0.288\tanh\left(0.121Re_p, f(Eo')\right)\right] & Eo' \leq 4 \\ f(Eo') & 4 < Eo' \leq 10 \\ -0.27 & 10 < Eo' \end{cases} \quad (7)$$

Where $f(Eo') = 0.00105Eo'^3 - 0.0159Eo'^2 - 0.0204Eo' + 0474$ and Eo' is the modified Etövös number given by the expression:

$$Eo' = \frac{g\Delta\rho d_H{}^2}{\sigma} \quad (8)$$

The term d_H is the long axis of the deformable bubble and is given by:

$$d_H = d_p(1 + 0.163Eo^{0.757})^{\frac{1}{3}} \quad (9)$$

Within a subsea gas release, plume coalescence and breakup of the individual bubbles is another non-trivial mechanism affecting the plume development. Bubble breakup phenomenon is based on the model by Luo and Svendsen [6] whereas bubble coalescence is based on the model developed by Prince and Blanch [7].

The current CFD model also takes into account the phenomena of virtual mass and the turbulent dispersion force by utilizing the corresponding models available in the ANSYS CFX software.

Computational Grid

The CFD simulations were carried out on a grid of approximately one million cells as illustrated in Figure 1. The computational domain included both the water and the air phases in order to capture the interaction between the plume and the water surface. The cells were refined in the areas near the water surface and the region where the gas plume is expected to occur. Inflation layers were used at the water surface to capture a sharp interface between the water and the air phases as shown in Figure 2.

A mesh sensitivity study was carried out before the analysis to ensure the results obtained using the chosen grid achieved grid independence. The sensitivity study also revealed that the refined grid region had a significantly impact on the plume behavior. Therefore the refined grid region was made to be large enough to cover the entire volume where the gas plume is expected to occur.

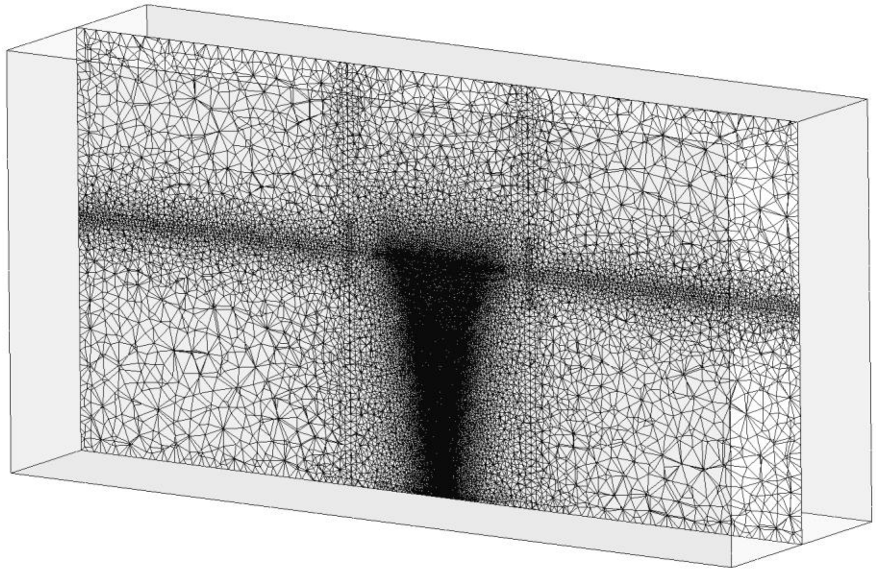

Figure 1 – Cross section of the grid used in the CFD analysis

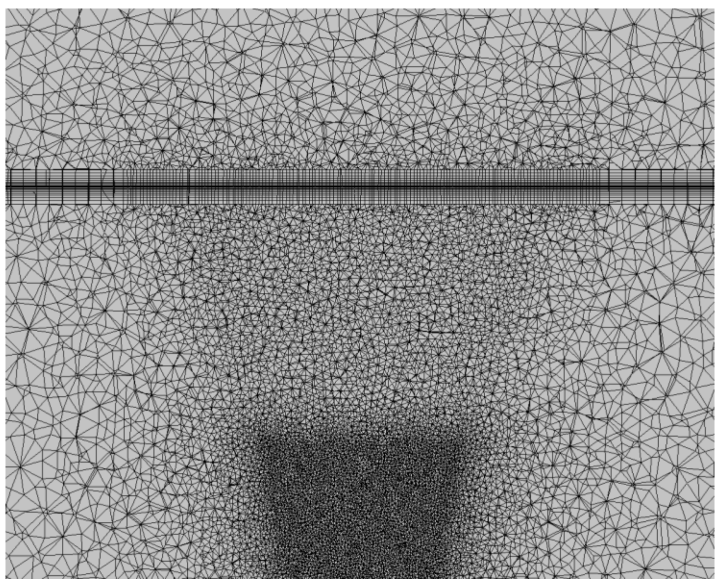

Figure 2 – Inflation layer in the computational grid at the water and air interface

Numerical Approaches

Four different numerical approaches were assessed for their suitability to be used in gas plume modelling. They can be classified into three main categories, the Reynolds-Averaged Navier–Stokes (RANS) approach, the Unsteady Reynolds Averaged Navier-Stokes (URANS) approach, and the Large Eddy Simulation (LES) approach.

In the RANS approach the equations of motion for fluid flow are time averaged by Reynold’s decomposition method. The fluctuation components in the RANS equation are expressed in a term known as the Reynolds stress. The Reynolds stress term has to be modelled as a function of the mean flow using the method known as the turbulence model in order to remove the equation’s dependency on the fluctuating components. For transient flow simulations, the equations are ensemble-averaged. The resulting equations are sometimes called URANS.

Under the RANS approach, two types of turbulence model were assessed for the subsea gas plume model, the standard k-ε, and the modified k-ε [3]. In the modified k-ε turbulence model, three additional terms were added to the standard k-ε turbulence model to take into account the free surface turbulence damping, the additional turbulence generated by the buoyancy of the bubbles in the plume, and additional turbulence induced by bubble wake.

In contrast, the approach of using LES is to solve directly for large-scale fluctuating motions and uses "sub-grid" scale turbulence models for the small-scale motion. Turbulent flows contain a wide range of length and time scales, with large scale motions being generally more energetic than the small scale ones. The LES approach is a suitable option for cases where the flow is likely to be unstable, with large scale fluctuation of a shear layer [5] as present in a gas plume. The Wall-Adapting Local Eddy-viscosity (WALE) “"sub-grid" scale turbulence model was used to model the “sub-grid” scale motions.

In an alternative method, a new class of URANS approach has been developed that can provide LES-like behavior in detached flow regions [8] [9]. The Scale-Adaptive Simulation (SAS) concept is based on the introduction of the von Karman length-scale into the turbulence scale equation. The information provided by the von Karman length-scale allows SAS models to dynamically adjust to the resolved structures in an URANS simulation, which results in LES-like behavior in unsteady regions of the flow field. At the same time, the model provides standard RANS capabilities in stable flow regions.

<u>Validation Case</u>

CFD models using the four numerical approaches were validated against the experimental data presented in [10] and the results were compared. In Fannelop’s experiment [10], air was released at a depth of 10 m below the water surface with a release rate of 0.022 Nm^3/s. The experiment was carried out in a towing basin that is 260 m in length, 10.5 m in width and 10 m in depth. The plume velocity profiles were recorded at various depths throughout the water column.

Results and Discussions

Figure 3 illustrates the velocity profile on a plane through the center of the plume at 40 seconds after the initial release when the plume has firmly established in the water column.

Figure 4 presents a comparison of the vertical velocity profiles at various elevations through the water column between the CFD results for the four numerical approaches and the experimental data.

For the standard k-ε model, the central line velocity and the plume width were consistently over predicted and under predicted respectively. The maximum discrepancy between the CFD results and the experimental data is approximately 45%. This occurs at a water depth of 7.75 m.

One of the causes of the discrepancy between the CFD results and the experimental data is the inherently RANS nature of the standard k-ε turbulence model. RANS approach is a time-averaged method and therefore tends to average out all the fluctuations in the flow field. The fluctuating flow field leads to plume wandering which is a major mechanism in dissipating the momentum gained by the plume from the initial jet flow in the zone of establishment. A reduced flow field fluctuation will result in the retention of the momentum to the core of the gas plume instead of distributing to the surrounding water domain and therefore maintaining a high plume central line velocity throughout the water depth. Momentum retention also leads to the narrowing of the plume width in the CFD results compared to the experimental data.

The modified k-ε turbulence model tries to address the above issue by introducing additional modification terms to the turbulence equations in order to take into account of the turbulence generated by both the plume wandering behavior and the buoyancy force.

The results show that the modified k-ε turbulence model was better at capturing the plume width than the standard k-ε model. However, the plume central line velocities are still over-predicted by this model, albeit improved when compared to the standard k-ε model.

Although the modifications to the k-ε turbulence model were able to mitigate some of the issues caused by the RANS approach, they were unable to resolve the fundamental issue of capturing the fluctuating velocity components. More advanced numerical approaches would be needed to overcome the above issue.

Both the LES and SAS approaches are able to capture the dynamic plume behavior that the RANS approach failed to achieve. They also showed reasonable agreements in terms of both the central line velocities and plume width for all elevations. Although the SAS model has a noticeable improvement over the RANS approach, the over-prediction of the central line velocity (maximum discrepancy is 20%) and the under-prediction of the plume width when compared to the experimental data still exists. The LES model correlates more closely to the experiment.

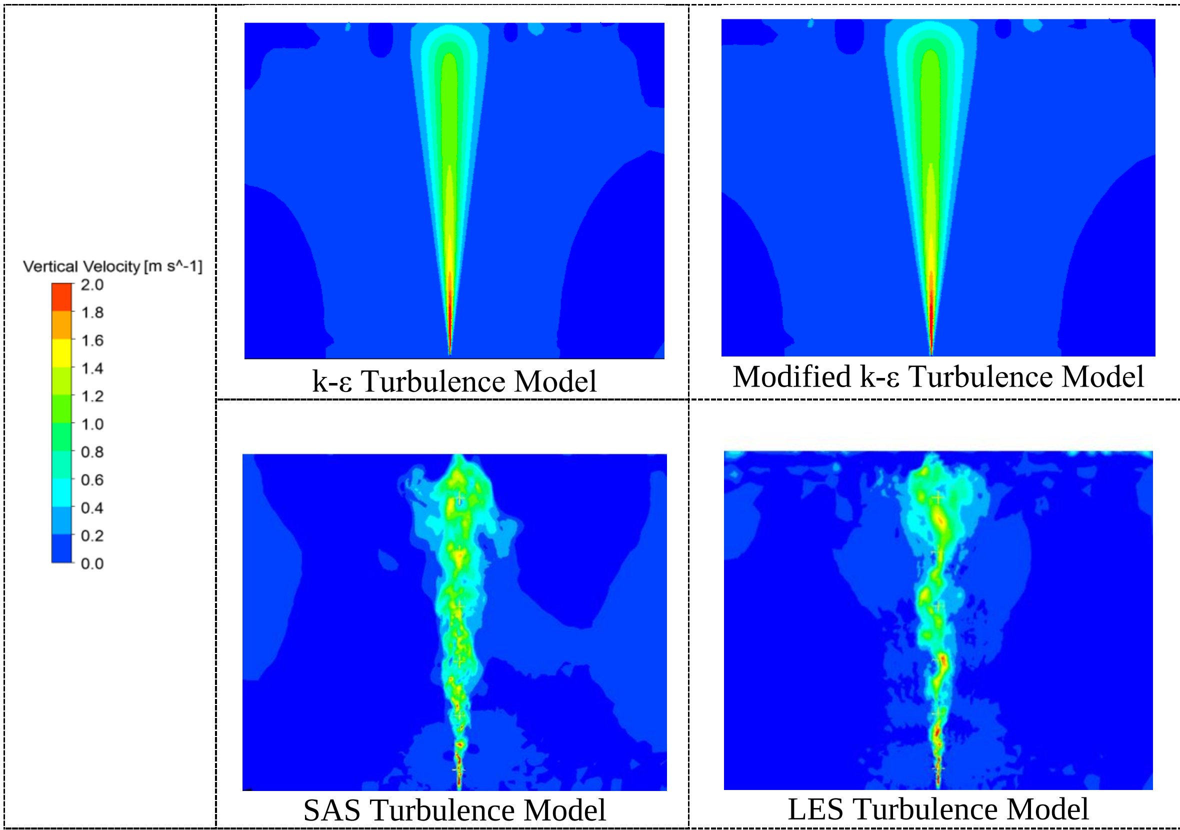

Figure 3 – Contours of vertical velocity in the plume for the k-ε, Modified k-ε, SAS and LES turbulence models

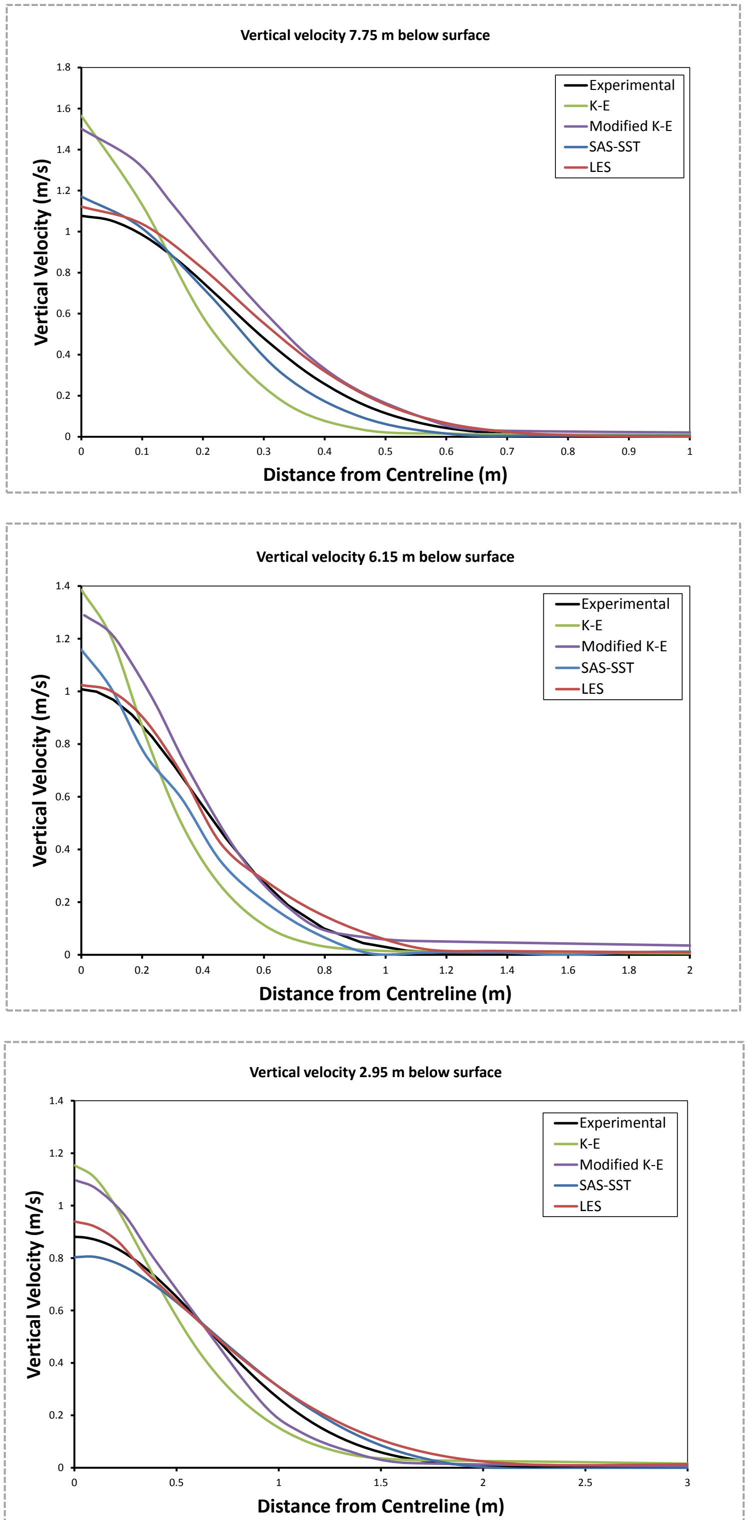

Figure 4 – velocity profiles at the elevation 7.75 m, 6.15 m, and 2.95 m below water surface

Conclusions

The k-ε turbulence models from the RANS approach were unable to capture the fluctuating velocity components in a rising subsea gas plume due to the inherent RANS nature of the model. This results in the over-prediction of the plume central line velocities and under-prediction of the plume width as there was no mechanism to distribute and dissipate the high momentum gained during the initial gas release phase.

In order to improve the capture of small scale structures in bubble column flows, LES or SAS approach is desirable.

The SAS approach has a noticeable improvement over the RANS approach. However, over-prediction of the central line velocity and under-prediction of the plume width still exists.

The LES approach produces a more accurate representation of the rising bubble plume. The results obtained using this turbulence model show that both the centerline velocity and the velocity profile are in much closer agreement with the experimental data. The model is also successful in predicting the inherently random wandering behavior of the subsea gas plume.

However, the disadvantage of using the LES model is the long simulation time associated with this methodology. The LES simulation requires the usage of very small timesteps in order to avoid numerical stability issues and to keep the CFL number to recommended values.

The SAS approach would be a good candidate to achieve a balance between accuracy in CFD results and the computational time required in carrying out simulations.

Abbreviations

CFD	Computational Fluid Dynamics
LES	Large Eddy Simulation
MUSIG	Multiple Size Group Model
RANS	Reynolds-Averaged Navier–Stokes
SAS	Scale-Adaptive Simulation
URANS	Unsteady Reynolds-Averaged Navier–Stokes
WALE	Wall-Adapting Local Eddy-viscosity

References

[1] S. Cloete, J. E. Olsen and P. Skjetne, “CFD modelling of plume and free surface behaviour resulting from a subsea gas release,” Applied Ocean Research, vol.31, pp. 220-225 , 2009.

[2] Engebretsen T., et al, Surface flow and gas dispersion from a subsea release of natural gas, Honolulu, USA: Proceedings of the seventh international offshore and polar engineering conference, May 25-30, 1997.

[3] Q. Pan and S. T. Johansen, “An enhanced k-epsilon model for bubble plumes,” in *8th International Conference on Multiphase Flow*, Korea, May 2013.

[4] Milgram, J. H., Mean flow in round bubble plumes, Journal of Fluid Mechanics vol. 133, pp. 345-376, 1983.

[5] Ansys CFX 14.5 Reference manual, 2012.

[6] Luo H., Svendsen H.F., Theoretical model for drop and bubble breakup in turbulent dispersions, AIChE Journal, 1996.

[7] Prince M. J., Blanch H. W., Bubble coalescence and break-up in air-sparged bubble columns, AIChE Journal, 1990.

[8] Y. M. F. Egorov, “Development and Application of SST-SAS Turbulence Model in the DESIDER Project,” in *Second Symposium on Hybrid RANS-LES Methods*, Corfu, Greece, 2007.

[9] E. Y. Menter F. R., “A Scale-Adaptive Simulation Model using Two-Equation Models,” *AIAA paper 2005-1095,* November 2005.

[10] Sojen, T. K. Fannelop & K., Hydrodynamics of underwater blowouts, Pasadena, CA: Proc. AIAA 18th Aerospace Sci. Meeting, 1980.

[11] Tomiyama A., Struggle with computational bubble dynamics, Lyon, France: ICMF'98, 3rd Int. Conf. Multiphase Flow, pp. 1-18, June 8-12, 1998.