Eleventh International Conference on CFD in the Minerals and Process Industries
CSIRO, Melbourne, Australia
7-9 December 2015

VLES TURBULENCE MODEL FOR AN EULERIAN-LAGRANGIAN MODELLING CONCEPT FOR BUBBLE PLUMES

Jan Erik OLSEN*, **Paal SKJETNE, Stein Tore JOHANSEN**

SINTEF Materials & Chemistry, 7465 Trondheim, NORWAY
*Corresponding author, E-mail address: Jan.E.Olsen@sintef.no

ABSTRACT

Traditional Reynolds-averaged Navier-Stokes (RANS) approaches to turbulence modelling, such as the k-ϵ model, has some well-known shortcomings when modelling transient flow phenomena. To mitigate this, a filtered URANS model has been derived where turbulent structures larger than a given filter size (typically grid size) is captured by the flow equations and smaller structures are modelled according to a modified k-ϵ model. This modelling approach is also known as a VLES model (Very Large Eddy Scale model), and provides more details of the transient turbulence than the k-ϵ model at little extra computational cost.

In this study a two-phase extension to the VLES model is described. A modelling concept for bubble plumes has been developed in which the bubbles are tracked as particles and the flow of liquid is solved by the Navier-Stokes equations in a traditional mesh based approach. The flow of bubbles and liquid is coupled in an Eulerian-Lagrangian model. Turbulent dispersion of the bubbles is treated by a random walk model. The random walk model depends on an estimation of the eddy life time. The eddy life time for the VLES model differs from a k-ϵ model, and its mathematical expression is derived.

The model is applied to ocean plumes emanating from discharge of gas at the ocean floor. Validation with experiments and comparison with k-ϵ model are shown.

NOMENCLATURE

b plume radius [m]
d diameter [m]
C coefficient [ ]
F force per mass [N/kg]
g coefficient of gravity [m/s^2]
k turbulent kinetic energy [m^2/s^2]
l length scale [m]
M molecular weight [kg/mole]
m mass of [kg]
p pressure [Pa]
U average velocity [m/s]
u velocity [m/s]
u' velocity fluctuations [m/s]
R gas constant [J/K mol]
r radius [m]
Re Reynolds number [ ]
S momentum source term [N/m^3]
T temperature [K]
t time [s]
V volume [m^3]
z height above release source [m]

α volume fraction [ ]
Δ filter size [m]
ϵ turbulent dissipation rate [m^2/s^3]
ξ random number
μ dynamic viscosity [Pa s]
ν kinematic viscosity [m^2/s]
ρ density [kg/m^3]
σ surface tension [N/m]
τ time scale [s]

Indexes
b bubble
c computational cell, centre
cb coalescence and break up
e eddy
eff effective
D drag
t turbulence
VM virtual mass
Δ sub-filter quantities

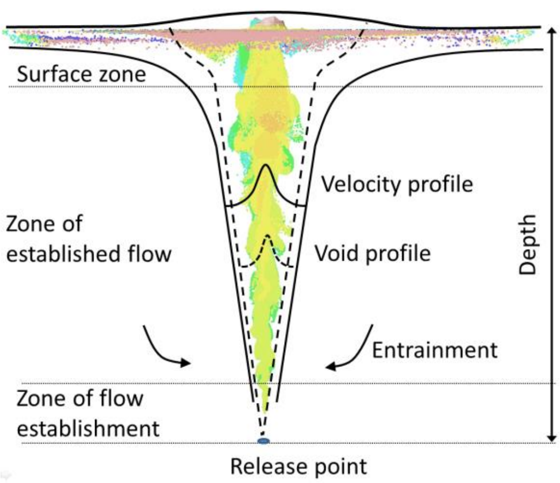

Figure 1: Bubble plume in ocean.

INTRODUCTION

Bubble plumes are found in various industrial processes, in coastal and harbour facilities and in natural and accidental subsea discharges. Accurate mathematical predictions of their behaviour enable cost effective process optimization and reliable risk assessments.

Mathematical modelling of bubble plumes dates back to WWII (Morton *et al.*, 1956)[1] when classical models for buoyant plumes were developed. These models have later been enhanced to include more relevant physics such as gas expansion, two phase effects and gas dissolution. The models assume a profile (typically Gaussian) for the vertical velocity and the volume fraction of bubbles. An integration of the conservation equations for mass and momentum is integrated in the radial direction creating a 1D set of equations. The models are thus also known as integral models.

CFD became a viable tool for two-phase flows during the 1980's enabling modelling of bubble plumes. Schwarz and Turner (1988) developed an Eulerian-Eulerian model for bubble plumes in metal reactors. Euler-Lagrangian approaches were also developed in the 1980's. Johansen and Boysan (1988) published an axisymmetric model for bubble plumes in reactors. It has been argued that in full 3D and for relevant gas release rates, Lagrangian tracking of the resulting number of bubbles is very demanding on computer resources, and thus an Eulerian-Eulerian approach is preferable for a bubble plumes with a huge quantity of bubbles. Swan and Moros (1993) solved this issue by tracking groups of bubbles instead of individual bubbles. They adopted the technique in an axisymmetric model for subsea blowouts similar to Johansen and Boysan (1988).

Fully 3D modelling based on the Eulerian-Lagrangian approach was demonstrated by Cloete *et al.* (2009) to accurately reproduce experimental results from pool experiments. The Eulerian-Lagrangian modelling concept is based on a VOF (volume of fluid) model for capturing the flow in the continuous phases and the interphase between the continuous phases, and a discrete phase model, DPM, for tracking the bubble motion. The bubbles are tracked in parcels representing many bubbles (or particles) where all bubbles share the same properties similar to Swan and Moros (1993). Thus the VOF-DPM approach is computationally affordable.

Most of these two-phase models deploy a k-ϵ model for quantifying the turbulence in the flow. The model is robust and computationally affordable, but is known to fail on predictions of certain aspects of transient behaviour. The LES model is better suited for transient flow. However, the computational cost can become substantial. Thus a more affordable modelling approach for turbulence in transient flows has been proposed which inherently captures the larger turbulent structures. It is known as a VLES model (very large eddy scale). In the following chapter we describe the Eulerian-Lagrangian modelling concept for bubble plumes and how a VLES model is coupled to the modelling concept.

[1] The work was initiated prior to WWII, but published much later.

MODEL DESCRIPTION

A bubble plume model calculates the flow of bubbles, liquids and if necessary gas above liquids. In a Lagrangian framework the bubbles move according to Newton's second law. The bubble acceleration is given by a force balance:

$$\frac{d\boldsymbol{u}_b}{dt} = \frac{\boldsymbol{g}(\rho_b - \rho)}{\rho_b} + \boldsymbol{F}_D + \boldsymbol{F}_{VM} \tag{1}$$

The first term on the right hand side is the specific buoyancy force (force divided by bubble mass). The other forces are drag and virtual mass force. The specific drag force is

$$\boldsymbol{F}_D = \frac{18\mu}{\rho_b d_b^2}\frac{C_D \mathrm{Re}}{24}(\boldsymbol{u}_b - \boldsymbol{u}) \tag{2}$$

where C_D is the drag coefficient, Re is the Reynolds number, ρ_b is the density of the bubble gas and d_b is the bubble diameter. The driving mechanism of the drag force is the velocity difference between the bubbles and the liquid $\boldsymbol{u}_b - \boldsymbol{u}$. Note that $\boldsymbol{u}$ is the instantaneous velocity of the background fluid

$$\boldsymbol{u} = \boldsymbol{U} + \boldsymbol{u}' \tag{3}$$

accounting for both the average velocity $\boldsymbol{U}$ and the turbulent fluctuations $\boldsymbol{u}'$. The turbulent fluctuations in the drag force cause turbulent dispersion. As in all models not resolving the turbulence, the turbulent dispersion is calculated by a sub-model. For Lagrangian tracking of bubbles (or particles) we apply a *random walk model* (Gosman & Ioannides, 1983) in which the turbulent velocity fluctuations is calculated by

$$\boldsymbol{u}' = \boldsymbol{\xi}\sqrt{k} \tag{4}$$

if a k-ϵ turbulence model is deployed. Here $\boldsymbol{\xi}$ is a random number and k is the turbulent kinetic energy. The time of which this velocity fluctuation is applied in the integration of the bubble trajectory is limited by the eddy lifetime (or the time it takes for a bubble to traverse through a turbulent eddy). The eddy lifetime is

$$\tau_e = 0.15\frac{k}{\epsilon} \tag{5}$$

for a k-ϵ model. The drag coefficient is provided by the expression of Tomiyama *et al.* (1998) for contaminated conditions with a correction for bubble interactions at higher volume fractions based on Tsuji *et al.* (1982).

Virtual mass force also known as added mass force is the force added to a bubble because an accelerating body is deflecting some volume of the surrounding fluid as it moves through it. The specific force is given as

$$\boldsymbol{F}_{VM} = C_{VM}\frac{\rho}{\rho_b}\left(\frac{D\boldsymbol{u}}{Dt} - \frac{d\boldsymbol{u}_b}{dt}\right) \tag{6}$$

where $C_{VM} = 0.5$ is the virtual mass coefficient. Lift force is normally included in reactor modelling, but sensitivity studies show no effect of the lift force in typical bubble plumes in open waters. This is due to the absence of walls close to the bubbles. In such scenarios the shear rate is relatively small and the lift force can be discarded (Olsen & Popescu, 2014).

The bubble size in dense plumes is assumed to be governed by turbulence break up and coalescence. In more dilute plumes mass transfer and gas expansion due to pressure gradients will dominate. A bubble size model for an Eulerian framework accounting for break up and coalescence was developed by Laux and Johansen (1999). When recasting this model into a Lagrangian concept and including the effect of mass transfer and gas expansion the bubble size is given by the following differential equation

$$\dot{d}_b = \frac{d_b^{eq} - d_b}{\tau_{cb}} + \frac{d_b}{3}\left(\frac{\dot{m}_b}{m_b} - \frac{\dot{\rho}_b}{\rho_b}\right) \tag{7}$$

Here m_b is the mass of a bubble, $\dot{m}_b$ is the mass transfer rate from a bubble, $\dot{\rho}_b$ is the Lagrangian time derivative of the bubble density, τ_{cb} is the time scale for coalescence or break up and d_b^{eq} is the bubble diameter obtained by a bubble if it is exposed to given flow conditions (turbulent dissipation and plume density) for a sufficient time (i.e equilibrium is reached). This equilibrium bubble diameter is

$$d_b^{eq} = C_1\sqrt{\alpha}\,\frac{\left(\sigma/\rho\right)^{0.6}}{\epsilon^{0.4}}\left(\frac{\mu_b}{\mu}\right)^{0.25} + C_2 \tag{8}$$

where α is the volume fraction of bubbles, σ is the surface tension and ϵ is the turbulent energy dissipation. For the model coefficients we assume $C_1 = 4.0$ (typical for bubbles in liquids) and $C_2 = 200\,\mu$m (smallest expected bubble size). For further details, including time scale for coalescence or break up, refer to Laux and Johansen (1999).

The bubble (i.e. gas) density and its derivatives is governed by the compressibility of gas (i.e. pressure dependence). Since bubbles normally rise in water towards a lower hydrostatic pressure, we frequently describes this as gas expansion. The gas density is a function of pressure and for moderate depths we apply the ideal gas law

$$\rho_b = \frac{M_b p}{RT} \tag{9}$$

Here p is pressure, M_b is molecular weight of gas in bubble, R is the gas constant and T is temperature. For deeper plumes (typically below 200 meters) higher order correlations are required.

The motion of the bubbles is coupled to the flow of the background fluid. The background fluid is a liquid with a gas on top as illustrated in Figure 1. The bubbles are removed upon entering the gas phase. An Eulerian VOF method conserving mass and momentum through the Navier-Stokes equations is deployed to calculate the flow of the continuous background phases (Hirt & Nichols, 1981). The interface between the continuous liquid and gas phases are tracked by the GEO reconstruct scheme (Youngs, 1982). The coupling with the Lagrangian bubbles is achieved through a source term in the momentum equation accounting for bubble drag

$$\rho\frac{D\mathbf{U}}{Dt} = \rho\boldsymbol{g} - \nabla p + \nabla\cdot\left[\mu_{\text{eff}}(\nabla\boldsymbol{U} + \nabla\boldsymbol{U}^T)\right] + \boldsymbol{S}_b \tag{10}$$

where μ_{eff} is the effective viscosity (molecular + turbulent) and S_b is the source term due to drag of bubbles

$$\boldsymbol{S}_b = \sum\frac{18\mu C_D \text{Re}}{24\rho_b d_b^2}(\boldsymbol{u}_b - \boldsymbol{u})\dot{m}_b\frac{\Delta t}{\Delta V_c} \tag{11}$$

Here Δt is the time step and ΔV_c is the volume of the computational cell. Turbulence and turbulent viscosity is accounted for by the standard k-ϵ model (Launder & Spalding, 1974). Turbulence is damped at the interface between the continuous liquid phase and the gas phase above because turbulent structures are not carried through the interface. This is not inherently accounted for by VOF models since the interfaces are not treated as boundaries. Thus a source term in the dissipation equation for turbulence is added to increase dissipation and dampen turbulence at the interface (Pan *et al.*, 2014).

VLES model

Modelling turbulence by a RANS approach (e.g. k-ϵ model) is quite common in engineering computations of turbulent flows. The models are robust and computationally affordable, but have some well-known deficiencies for transient flows. LES modelling is an alternative, but is computationally quite expensive. By introducing a filter in an unsteady RANS approach Johansen *et al.* (2004) developed an affordable transient turbulence model. The model captures the turbulent structures above a given filter size by the momentum equations and leaves the remaining turbulent spectrum to a sub-filter model similar to a RANS model. Thus less of the turbulence spectrum is left to the model assumptions of the RANS approach. The turbulence model is known as a VLES model (very large eddy scale) and has later been adopted by others, e.g. Labois and Lakehal (2011).

Johansen *et al.* (2004) showed by filtering the turbulence spectrum that the kinetic viscosity is

$$\nu_t = C_\mu\,\frac{k_\Delta^2}{\epsilon_\Delta}\cdot\text{MIN}\left[1\,;\ \frac{\Delta\,\epsilon_\Delta}{k_\Delta^{3/2}}\right] \tag{12}$$

if a filter size Δ for the turbulence is applied. The subscript Δ on k and ϵ indicate that the turbulence model only captures the turbulence spectrum below the filter size. The large scale turbulence is captured by the momentum equation. We see that for large filter sizes the turbulent viscosity is given by the sub-filter model. With a standard k-ϵ model as the sub-filter model, the VLES model is defined by Eq.(12) and the partial differential equations for kinetic energy and energy dissipation of the k-ϵ model. We apply the grid size as the filter size Δ. Thus for coarse grids most of the turbulence is maintained by the sub-filter model (u'), and for finer grids more of the turbulence is maintained within the velocity field governed by the momentum equations (U).

In order to apply the VLES model in an Eulerian-Lagrangian modelling framework an expression for the eddy time scale is required for the random walk model, i.e. Eq.(5) needs to be modified. Eddy lifetime for non-resolved turbulence is

$$\tau_{e_\Delta} = \frac{3}{2}\frac{\nu_t}{k_\Delta} = \frac{3}{2}C_\mu\,\frac{k_\Delta}{\epsilon_\Delta}\cdot\text{MIN}\left[1\,;\ \frac{\Delta\,\epsilon_\Delta}{k_\Delta^{3/2}}\right] \tag{13}$$

The subindex Δ in the above equation indicates that the kinetic energy and dissipation is based on the turbulent energy residing within the length scales of the filter.

The modelling concept is implemented in ANSYS/Fluent 14.0. The PISO scheme is applied for pressure-velocity coupling, spatial discretization are second order or higher and the time discretization is implicit first order. The PISO scheme is normally robust with fast convergence.

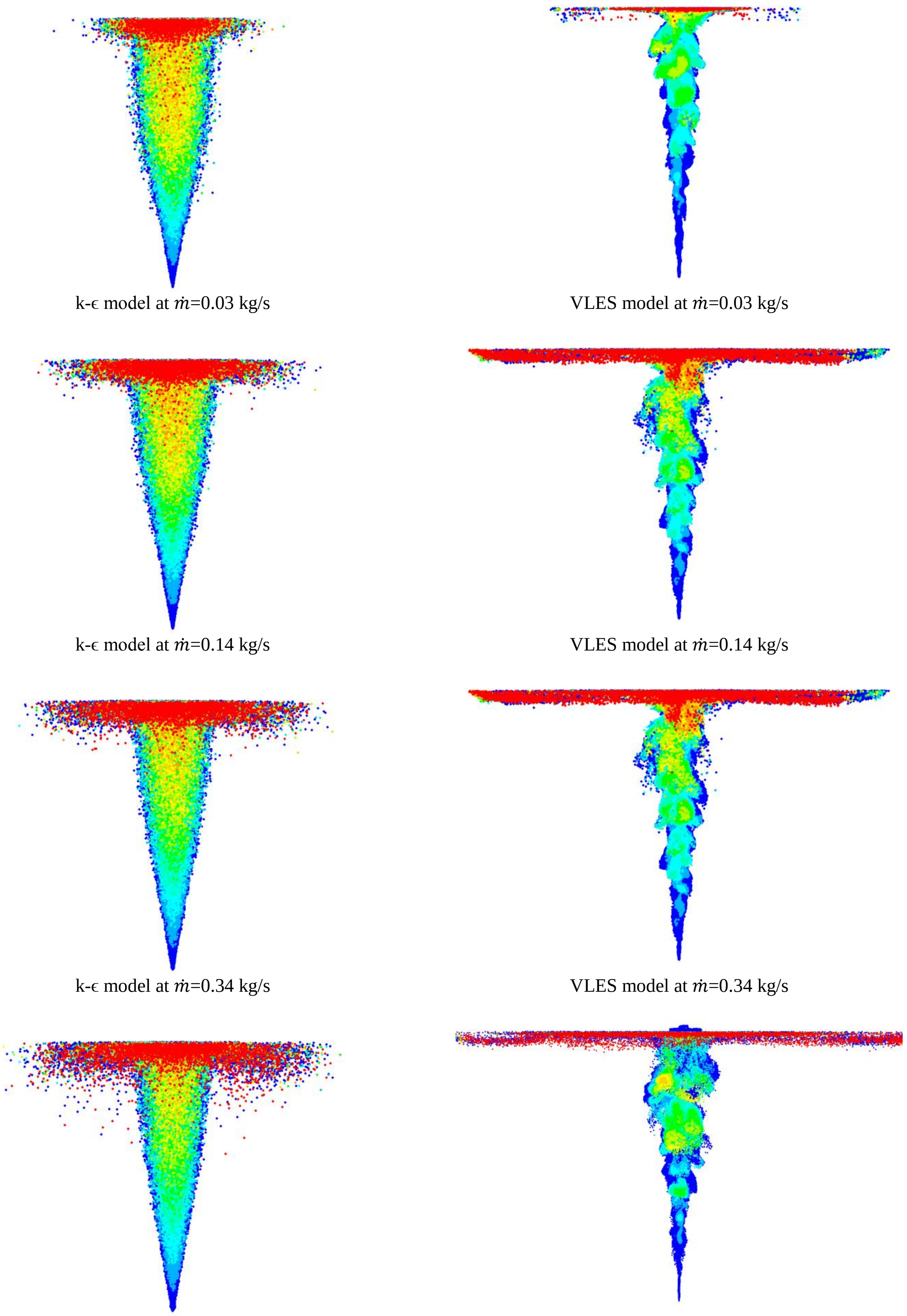

Figure 2: Plume shapes coloured by bubble distance out of the image plane for different gas rates and turbulence models. Blue equals 0 meter and red equals 8 meters.

RESULTS

The above modelling concept has been applied to a series of gas releases from 50 meters depth. This is equivalent to the cases presented in the experimental study of Milgram (1983) who released air at gas rates of 0.03, 0.14, 0.34 and 0.71 kg/s in a sink hole in Florida. The bubble plumes achieved after a quasi-steady state is reached are shown in Figure 2. Results for both VLES and k-ϵ turbulence models are compared. We see how the results with the k-ϵ model reflect the averaged nature of the turbulence model with clear cone shaped plumes. The plume shapes of the VLES model include turbulent structures typically observed in experiments. These observations do not necessarily support that the VLES model is superior, as a time average of the VLES model might result in the same plume shapes as for the k-ϵ model.

Qualitatively we see that the VLES model captures more of the turbulent structures. This is also confirmed by Figure 3 where the vertical velocity midway between release source and water surface is plotted. The k-ϵ model produces a typical averaged velocity plot with very small fluctuations. The VLES model reproduces the larger velocity fluctuations as expected from this kind of turbulence model.

The model results can also be compared quantitatively with the experimental results of Milgram (1983). Milgram measured velocities at different heights above the release source and fitted the measurement to Gaussian velocity profiles

$$U(z,r) = U_c(z) \cdot \exp(-r^2/b(z)^2) \tag{14}$$

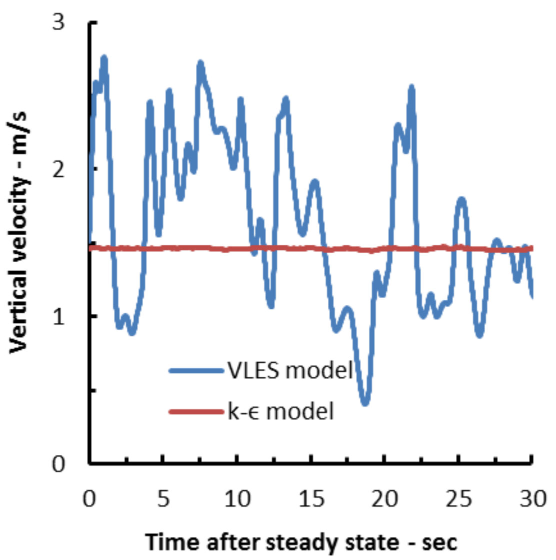

Figure 3: Vertical velocity 25 meters above release point at an arbitrary time period after steady state is established.

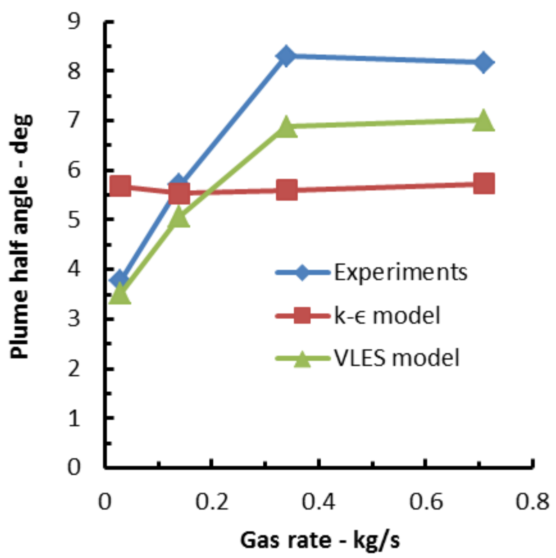

Figure 4: Plume angle as function of gas rate.

where the plume radius, b, and the axis velocity, U_c, varies with distance, z, above the release source. By defining a plume angle based on the plume radius 43.9 meters above the release source, quantitative comparisons are made between experiments and the two turbulence models. The results are seen in Figure 4. We see that the k-ϵ model do not capture the trend of the experimental values. The VLES model captures the trend, but underpredicts the plume angle by roughly 10-15%. The deviation between the VLES model and the experiments are smaller at the lower gas rates.

SUMMARY AND DISCUSSION

An Eulerian-Lagrangian modelling concept for bubble plumes has been presented. A VLES turbulence model has been introduced into the modelling concept. When compared to a k-ϵ model, the VLES model captures more of the turbulence spectrum inherently and leaves less of the spectrum to the model assumptions of the sub-filter turbulence model. When comparing the model with experimental results of a series of gas discharges in a sink hole from a depth of 50 meters, we find that the VLES model is more consistent with the experimental observations than the k-ϵ model.

The VLES model underpredicts the plume angle somewhat. Two possible reasons have been identified. Firstly it should be emphasized that the model does not properly resolve the flow close to the release source. Underprediction of the plume spreading in this region will be carried along with the plume all the way to the surface. Secondly bubble induced turbulence is not accounted for. Bubble induced turbulence will create more spreading of the plume. Thus future efforts will focus on bubble induced turbulence and the ability to capture more of the physics close to the release source.

ACKNOWLEDGEMENT

This study was performed within the SURE-project (phase II) supported by Gassco, Statoil, Total, Safetec, Wild Well Control and Petroleum Safety Authority Norway.

REFERENCES

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

GOSMAN, A.D., & IOANNIDES, E. (1983). "Aspects of computer simulation of liquid-fuelled combustors". *J.Energy*, **7**, 482-490.

HIRT, C.W., & NICHOLS, B.D. (1981). "Volume of fluid (VOF) method for the dynamics of free boundaries". *Journal of Computational Physics,* **39**(1), 201-225.

JOHANSEN, S.T., & BOYSAN, F. (1988). "Fluid Dynamics in Bubble Stirred Ladles: Part II. Mathematical Modelling". *Metallurgical Transcations B,* **19B**, 756-764.

JOHANSEN, S.T., WU, J., & SHYY, W. (2004). "Filter-based unsteady RANS computations". *Heat and Fluid Flow,* **25**, 10-21.

LABOIS, M., & LAKEHAL, D. (2011). "Very-Large Eddy Simulation (V-LES) of the flow across a tube bundle". *Nuclear Engineering and Design,* **241**(6), 2075-2085.

LAUNDER, B.E., & SPALDING, D. (1974). "The numerical computation of turbulent flows". *Computer methods in applied mechanics and engineering,* **3**, 269-289.

LAUX, H., & JOHANSEN, S.T. (1999). A CFD analysis of the air entrainment rate due to a plunging steel jet combining mathematical models for dispersed and separated multiphase flows *Fluid Flow Phenomena in Metals Processing* (pp. 21-30).

MILGRAM, J.H. (1983). "Mean flow in round bubble plumes". *Journal of Fluid Mechanics,* **133**, 345-376.

MORTON, B.R., TAYLOR, G.I., & TURNER, J.S. (1956). "Turbulent gravitational convection from maintained and instantaneous sources". *Proc.Roy.Soc.A,* **234**, 171-178.

OLSEN, J.E., & POPESCU, M. (2014). "On the effect of lift forces in bubble plumes". *Progress in Computational Fluid Dynamics,* **5**.

PAN, Q.Q., OLSEN, J.E., JOHANSEN, S.T., REED, M., & SÆTRAN, L. (2014). "CFD Study of Surface Flow and Gas Dispersion From a Subsea Gas Release ". *Proc. of the ASME 2014 33rd International Conference on Ocean, Offshore and Arctic Engineering,* San Francisco.

SCHWARZ, M.P., & TURNER, W.J. (1988). "Applicability of the standard k-ϵ turbulence model to gas-stirred baths". *Applied Mathematical Modelling,* **12**(3), 273-279.

SWAN, C., & MOROS, A. (1993). "The hydrodynamics of a subsea blowout". *Applied Ocean Research,* **15**(5), 269-280.

TOMIYAMA, A., KATAOKA, I., ZUN, I., & SAKAGUCHI, T. (1998). "Drag coeffcients of single bubbles under normal and micro gravity conditions". *JSME International Journal, Series B,* **41**, 472-479.

TSUJI, Y., MORIKAWA, Y., & TERASHIMA, K. (1982). "Fluid-dynamic interaction between two spheres". *Int.J.Multiphase Flow,* **8**(1), 71-82.

YOUNGS, D.L. (1982). Time-Dependent Multi-Material Flow with Large Fluid Distortion. In K. W. Morton & M. J. Banes (Eds.), *Numerical Methods for Fluid Dynamics*: Academic Press.

