V European Conference on Computational Fluid Dynamics
ECCOMAS CFD 2010
J. C. F. Pereira and A. Sequeira (Eds)
Lisbon, Portugal,14-17 June 2010

STABILISED FINITE ELEMENTS FOR HIGH REYNOLDS NUMBER, LES AND FREE SURFACE FLOW PROBLEMS

Guillaume François*, **Elie Hachem*** **and Thierry Coupez***

*Center for Material Forming, MINES ParisTech
1, rue Claude DAUNESSE
06 904 Sophia Antipolis, FRANCE
e-mail: thierry.coupez@mines-paristech.fr, web page: http://www.cemef.mines-paristech.fr/

Key words: Advanced numerical methods, Level Set, Large Eddy Simulations, Stabilized finite elements method

Abstract. *We propose a stabilized finite element method to address complex high Reynolds flows with free surface. An implicit stabilized finite element method (SFEM) is used for solving the incompressible two-phase Navier-Stokes equations in three-dimensions. A novel approach to deal with turbulent two-phase flows is highlighted by coupling a local convected levelset method to a Large Eddy simulation turbulent modeling. A comparison between the static and dynamic eddy-viscosity models is analyzed. We assess the behaviour and accuracy of the proposed stabilized finite element method coupled to the two-phase turbulent approximation in the simulation of complex 3D flows, such as the flow in a partially filled two communicating tanks. Results are compared with the experimental data and show that the present implementation is able to exhibit good stability and accuracy properties for high Reynolds number flows using unstructured meshes.*

1 INTRODUCTION

The analysis of high-Reynolds number turbulent multiphase flows has attracted considerable attention during the past few decades particularly in view of industrial and environmental applications such as sea waves, mold filling, casting and many others.

Various numerical techniques have been proposed, with and without turbulence modeling to overcome the challenges related to updating the interface intrinsically coupled with turbulent fluid dynamics equations. The great majority of these methods have adopted the Reynolds Averged Navier-Stokes (RANS) modeling in which only averaged quantities are computed [15] while others have used the Large Eddy Simulation (LES) [7], known to be more accurate for simulating two-phase character of the flow.

The main objective of this paper is then to present the numerical simulation of free surface turbulent flow using the finite element method with unstructered meshes. The interface between the liquid and air is captured by solving a convected LevelSet method [4, 22]. The proposed approach enables first to restrict convection resolution to the neighborhood of the interface and second to replace the reinitialisation steps by an advective reinitialisation. Such modifications provides an efficient method with a restraint calculation cost.

The turbulent eddy viscosity modelled via the LES turbulence model is computed directly from velocity field calculated at latest time step. Two approaches are used; the Smagorinsky approximation [20] which can be seen as a convenient and simple way to model subgrid scale behavior. A dynamic procedure [7] was used next to avoid the use of arbitrary and highly dissipative term yielding more accurate results.

Several applications can be found in the literature; the most common is the dam breaking experiment [1, 6, 13], for which experimental data are available [19]. Other authors were interested in wave behavior in 2D [10, 13, 16] and in 3D [1, 18]. 3D mold-filling simulations can also be found [12]. However, simple three dimensional turbulent filling cases are rarely treated. In this paper, we present and report new experimental results of a simple water filling process and compare them with three dimensional simulations. Finally, conclusions and perspective are outlined.

2 Governing equations

2.1 Flow equations

Navier-Stokes equations of unsteady incompressible flow are written as follows:

$$\rho\frac{\partial \boldsymbol{u}}{\partial t} + \rho\boldsymbol{u}\nabla\cdot\boldsymbol{u} - \nabla\cdot(2\eta\varepsilon) + \nabla p = \boldsymbol{f} \tag{1}$$

$$\nabla\cdot\boldsymbol{u} = 0 \tag{2}$$

Where ρ, $\boldsymbol{u}$, p, η, ε and $\boldsymbol{f}$ are the density, velocity, pressure, dynamic viscosity, strain rate tensor, and specific body force. In the present monolithic approach, physical data as density and viscosity are taken non constant and depend on a phase function α. For a two-phase flow, we have:

$$\begin{aligned}\rho &= H(\alpha)\rho_1 + (1-H(\alpha))\rho_2\\ \eta_a &= H(\alpha)\eta_1 + (1-H(\alpha))\eta_2\end{aligned} \quad ; \quad H(\alpha) = \begin{cases} 1 & \text{in fluid 1} \\ 0 & \text{in fluid 2}\end{cases} \tag{3}$$

2.2 Interface tracking

A recently developed Level-Set method presented in [4, 22] is used to track the interface based on localized convected level-set function. This function enables to reduce computational cost and eases the use of boundary conditions. The convection equation is combined with reinitialisation by the so called *"convective reinitialisation"* method.

The modified level-set function is represented as follows:

$$\alpha = \begin{cases} \dfrac{2E}{\pi}\sin\left(\dfrac{\pi}{2E}\phi\right) & \text{for } |\phi| < E \\ 2E/\pi & \text{for } \phi > E \\ -2E/\pi & \text{for } \phi < -E \end{cases} \tag{4}$$

where ϕ stands for the standard distance function, and E the truncation thickness.

The basic idea of the method is to use the physical time and convective time derivative in the classical Hamilton-Jacobi reinitialisation equation. Once combined to the convection equation, it can be solved using:

$$\begin{cases} \dfrac{\partial \alpha}{\partial t} + \boldsymbol{u}.\nabla\alpha + \lambda s\left(|\nabla\alpha| - \sqrt{1-\left(\frac{\pi}{2E}\alpha\right)^2}\right) = 0 \\ \alpha(t=0,x) = \alpha_0(x) \end{cases} \tag{5}$$

where $\lambda = h/\Delta t$.

3 Computational methodology

3.1 Stable multiscale variational approach for Navier-Stokes equations

In this section the general equations of time-dependent Navier-Stokes equation are solved. The stabilizing schemes from a variational multiscale point of view are described and presented. The velocity and the pressure spaces are enriched by a space of bubbles that cures the spurious oscillations in the convection-dominated regime as well as the pressure instability.

Following the lines in [11, 9, 8], we consider an overlapping sum decomposition of the velocity and the pressure fields into resolvable coarse-scale and unresolved fine-scale $\vec{u} = \vec{u}_h + \vec{u}'$ and $p = p_h + p'$. Likewise, we regard the same decomposition for the weighting functions $\vec{w} = \vec{w}_h + \vec{w}'$ and $q = q_h + q'$. The unresolved fine-scales are usually modelled using residual based terms that are derived consistently. The static condensation consists in substituting the fine-scale solution into the large-scale problem providing additional terms, tuned by a local time-dependent stabilizing parameter, that enhance the stability and accuracy of the standard Galerkin formulation for the transient non-linear Navier-Stokes equations. Thus, the mixed-finite element approximation of problem (1-2) can read:

$$\text{Find a pair } (\vec{u}, p) \in \mathcal{V} \times \mathcal{Q}, \text{ such that: } \forall\ (\vec{w}, q) \in V_0 \times Q_0$$

$$\begin{cases} \rho\left(\partial_t(\vec{u}_h + \vec{u}'), (\vec{w}_h + \vec{w}\,')\right)_\Omega \\ \qquad + \rho\left((\vec{u}_h + \vec{u}')\cdot\boldsymbol{\nabla}(\vec{u}_h + \vec{u}'), (\vec{w}_h + \vec{w}\,')\right)_\Omega \\ \qquad + \left(2\eta\ \underline{\underline{\dot{\epsilon}}}(\vec{u}_h + \vec{u}\,') : \underline{\underline{\dot{\epsilon}}}(\vec{w}_h + \vec{w}\,')\right)_\Omega \\ \qquad - \left((p_h + p'), \boldsymbol{\nabla}\cdot(\vec{w}_h + \vec{w}\,')\right)_\Omega = \left(\vec{f}, (\vec{w}_h + \vec{w}\,')\right)_\Omega \\ \left(\boldsymbol{\nabla}\cdot(\vec{u}_h + \vec{u}\,'), (q_h + q')\right)_\Omega = 0 \end{cases} \tag{6}$$

To derive the stabilized formulation, we first solve the fine scale problem, defined on the sum of element interiors and written in terms of the time-dependant large-scale variables. Then we substitute the fine-scale solution back into the coarse problem, thereby *eliminating the explicit appearance of the fine-scale while still modelling their effects*. At this stage, three important remarks have to be made:

i) when using linear interpolation functions, the second derivatives vanish as well as all terms involving integrals over the element interior boundaries;

ii) the subscales will not be tracked in time, therefore, quasi-static subscales are considered here;

iii) the convective velocity of the nonlinear term may be approximated using only the large-scale part;

Consequently, applying integration by parts and then substituting the expressions of both the fine-scale pressure and the fine-scale velocity into the large-scale equations, we get

$$\left\{\begin{array}{l} \rho\left(\partial_t \vec{u}_h, \vec{w}_h\right)_\Omega + \left(\rho \vec{u}_h \cdot \boldsymbol{\nabla} \vec{u}_h, \vec{w}_h\right)_\Omega \\ \quad - \displaystyle\sum_{K \in \Omega_h} \left(\tau_K \boldsymbol{\mathcal{R}}_{\mathrm{M}}, \rho \vec{u}_h \boldsymbol{\nabla} \vec{w}_h\right)_K + \left(2\eta\ \underline{\underline{\dot{\epsilon}}}(\vec{u}_h) : \underline{\underline{\dot{\epsilon}}}(\vec{w}_h)\right)_\Omega \\ \quad - \left(p_h, \boldsymbol{\nabla} \cdot \vec{w}_h\right)_\Omega + \displaystyle\sum_{K \in \Omega_h} \left(\tau_{\mathrm{C}} \mathcal{R}_{\mathrm{C}}, \boldsymbol{\nabla} \cdot \vec{w}_h\right)_K \\ \qquad\qquad = \left(\vec{f}, \vec{w}_h\right)_\Omega \quad \forall \vec{w}_h \in V_{h,0} \\ \left(\boldsymbol{\nabla} \cdot \vec{u}_h, q_h\right)_\Omega - \displaystyle\sum_{K \in \Omega_h} \left(\tau_K \boldsymbol{\mathcal{R}}_{\mathrm{M}}, \boldsymbol{\nabla} q_h\right)_K = 0 \quad \forall q_h \in Q_{h,0} \end{array}\right. \tag{7}$$

When compared with the Galerkin method (6), the proposed stable formulation involves additional integrals that are evaluated element wise. These additional terms, obtained by replacing the approximated $\vec{u}\,'$ and p' into the large-scale equation, represent the effects of the sub-grid scales and they are introduced in a consistent way to the Galerkin formulation. All of these terms enable to overcome the instability of the classical formulation arising in convection dominated flows and to satisfy the inf-sup condition for the velocity and pressure interpolations.

For sake of simplicity in the notation and for a better representation of all the additional terms in equation (7), it is worth to mention that the condensation procedure of the small-scale into the large scale is masked under some stabilizing parameters. However, from the implementation point of view, the structure of the stabilizing parameters can be computed naturally via the element-level matrices [8].

In the bibliography a lot of estimations of stabilizing parameters can be found. For illustration, the most common used definition for the transient Navier-Stokes problems and linear elements comes from reference [21, 3, 2, 9]

$$\tau_K = \frac{1}{\sqrt{\left(\frac{2}{\Delta t}\right)^2 + \left(\frac{4\eta}{h^2}\right)^2 + \left(\frac{4|u_k|}{h}\right)^2}} \tag{8}$$

$$\tau_C = \left(\left(\frac{\mu}{\rho}\right)^2 + \left(\frac{c_2}{c_1}\frac{\|\vec{u}\|_K}{h}\right)^2\right)^{1/2} \tag{9}$$

3.2 Coupled LES-LevelSet method

3.2.1 Filtered Equations

The LES method is applied to compute the eddy viscosity. The basic idea is to decompose a variable Φ into two scales: the resolved scale term $\bar{\Phi}$ that is obtained by spatial averaging on the domain, and the subgrid scale term Φ' that has to be modelled to approach the problem and obtain $\bar{\Phi}$. For an heterogeneous flow, a new filtering operator $\widetilde{\Phi} = \frac{\overline{\rho\Phi}}{\bar{\rho}}$ can also be introduced. As shown in [14], the following formulation is obtained by filtering the Navier Stokes equations:

$$\begin{cases} \nabla\cdot\tilde{\boldsymbol{u}} = \sigma_0 \\ \bar{\rho}\dfrac{\partial\tilde{\boldsymbol{u}}}{\partial t} + \nabla\cdot(\bar{\rho}\tilde{\boldsymbol{u}}\otimes\tilde{\boldsymbol{u}}) + \nabla\cdot(-2\bar{\mu}\bar{\boldsymbol{\varepsilon}} + \bar{p}\mathbb{1}) - \bar{\rho}\boldsymbol{g} - \bar{\sigma}\bar{\kappa}\bar{\boldsymbol{n}}_{\boldsymbol{\Gamma}}\delta = \sum_{i=0}^{3}\boldsymbol{\tau_i} \end{cases} \tag{10}$$

New subgrid terms τ_i appear in the equation and are defined as follows:

$$\begin{aligned} &\boldsymbol{\tau}_0 = -\rho\overline{\frac{\partial\boldsymbol{u}}{\partial t}} + \bar{\rho}\frac{\partial\tilde{\boldsymbol{u}}}{\partial t} \quad ; \quad \boldsymbol{\tau}_1 = -\nabla\cdot(\overline{\rho\boldsymbol{u}\otimes\boldsymbol{u}} - \bar{\rho}\tilde{\boldsymbol{u}}\otimes\tilde{\boldsymbol{u}}) \\ &\boldsymbol{\tau}_2 = -\nabla\cdot(\overline{\mu\boldsymbol{\varepsilon}(\boldsymbol{u})} - \bar{\mu}\boldsymbol{\varepsilon}(\bar{\boldsymbol{u}})) \quad ; \quad \boldsymbol{\tau}_3 = \sigma\overline{\kappa\boldsymbol{n}_\Gamma}\delta - \sigma\bar{\kappa}\bar{\boldsymbol{n}}_\Gamma\delta \end{aligned} \tag{11}$$

These latter represent the coupling between multi-fluid flow and turbulence. τ_0, τ_1, τ_2 and τ_3 are respectively linked to subgrid acceleration, inertia, viscosity and interfacial term. These terms cannot be resolved by the system, they have to be modelled.
As shown in [14, 23], τ_1 is the main eddy term that has to be modelled to introduce an eddy viscosity [20].

3.2.2 Smagorinsky model

The Smagorinsky model [20] consists in modelling τ_1 by the contribution of a viscosity stress tensor, such as $\boldsymbol{\tau}_1 = 2\nabla\cdot(\bar{\rho}\nu_t\bar{\boldsymbol{\varepsilon}})$. For a monofluid computation, eddy viscosity ν_t can be written as follows:

$$\nu_t = (C_S\Delta)^2\,|\bar{\boldsymbol{\varepsilon}}| \tag{12}$$

where $|\boldsymbol{\varepsilon}| = \sqrt{2\varepsilon_{ij}\varepsilon_{ij}}$, Δ is the space filter length, C_S is the Smagorinsky constant.
It is usually assumed that the spatial filter application domain is the element volume, therefore the filter length can be set as $\Delta = V_{element}^{1/d}$. This assumption can then be made as well for an

isotropic as for an anisotropic mesh. The value of C_S is not fully determined by physics and may vary from one case to another, its value is generally set between 0.1 and 0.2.

3.2.3 Germano dynamic procedure

This procedure is based on the use of a second filter, whose length is larger than the previous filter. The correlation between computed data and their filtered value enables to determine C_S for each element. Consequently, the eddy viscosity computation can be more accurate and less dissipative.
Let $\bar{\cdot}$ and $\widehat{\cdot}$ be the first and second filter operators ($\widehat{\Delta} > \Delta$). Application of the second order filter applied to first order filtered Navier Stokes equations (10) yields:

$$\begin{aligned} &\mathbb{L} = C_S^2 \mathbb{M} && ; \quad \mathbb{L} = \widehat{\bar{\rho}\tilde{\boldsymbol{u}} \otimes \tilde{\boldsymbol{u}}} - \frac{\widehat{\bar{\rho}\tilde{\boldsymbol{u}}} \otimes \widehat{\bar{\rho}\tilde{\boldsymbol{u}}}}{\widehat{\bar{\rho}}} \\ &\xi = \frac{\Delta^2(\bar{\varepsilon} : \bar{\varepsilon})^{3/2}}{\hat{\Delta}^2(\widehat{\bar{\varepsilon}} : \widehat{\bar{\varepsilon}})^{3/2}} && ; \quad \mathbb{M} = \Delta^2(\widehat{\bar{\rho}\,|\bar{\boldsymbol{\varepsilon}}|\,\bar{\boldsymbol{\varepsilon}}}) - \xi\widehat{\bar{\rho}}\widehat{\Delta}^2\,|\widehat{\bar{\boldsymbol{\varepsilon}}}|\,\widehat{\bar{\boldsymbol{\varepsilon}}} \end{aligned} \tag{13}$$

A simple way to compute C_S from $\mathbb{L}$ and $\mathbb{M}$ is to use a least squares method [17]:

$$C_S^2 = \frac{\mathbb{L} : \mathbb{M}}{2\mathbb{M} : \mathbb{M}} \tag{14}$$

4 3D filling

As a validation of a turbulent sharp interface method in three-dimensions, we consider a simple water filling between two communicated tanks as shown in figure 1. The first tank (A), full of water is elevated from tank (B) which is initially filled by air. The experiment starts when the flood gate between the tanks is opened. The stabilized finite element method is applied to solve both the incompressible Navier-Stokes and the transport equations.

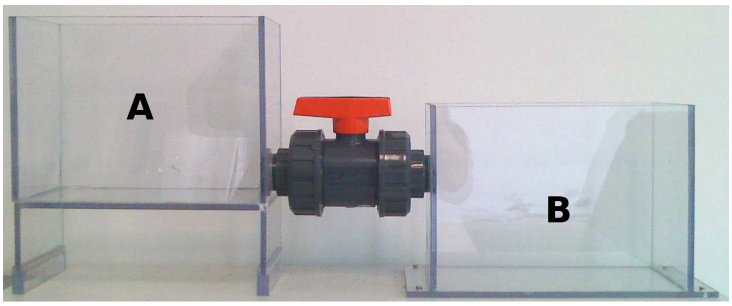
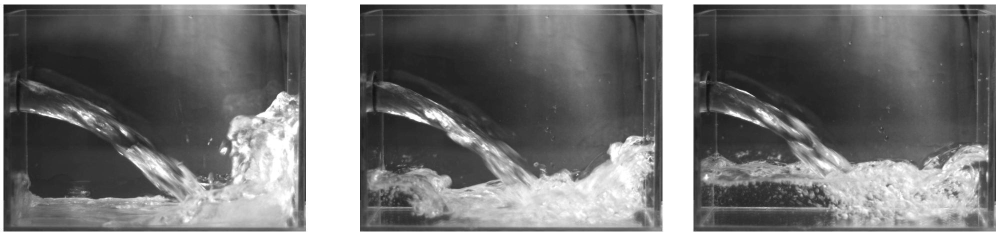


Figure 1: Experimental setup and illustrative picture

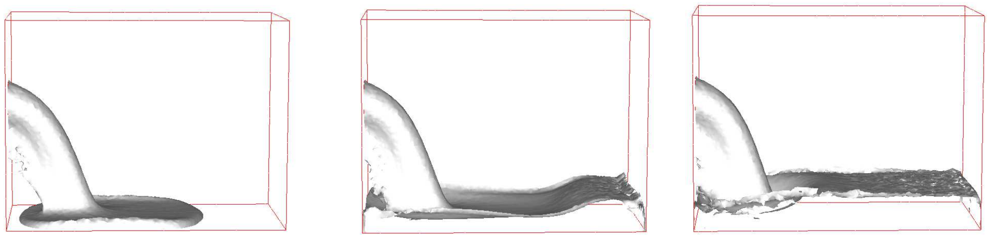
To simulate the flow in both tanks A and B, we use a 1 000 000 tetrahedrical elements in a three dimensional unstructured mesh. The time step is set to half CFL condition [5] in order to improve computational cost and limit instabilities.

Both static and dynamic Smagorinsky models have been used to simulate the flow for actual air/water densities and viscosities. The slippery conditions appeared to be most realistic conditions, but a wall friction model should be used to better represent boundary layer behavior.

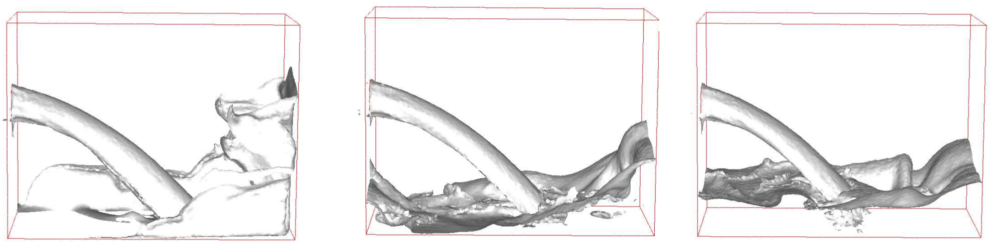
The comparison between experimental and numerical results are presented in figure 2. Computational results are presented for static and dynamic cases. As expected, the Smagorinsky static method is not adapted to give a realistic flow description, because subgrid dissipation is overestimated while the dynamic model provides an appropriate descrition during the whole simulation.



(a) $t = 0.5s$ (b) $t = 1s$ (c) $t = 1.5s$



(d) $t = 0.5s$ (e) $t = 1s$ (f) $t = 1.5s$



(g) $t = 0.5s$ (h) $t = 1s$ (i) $t = 1.5s$

Figure 2: Time evolution of water-air interface: experimental (top), and numerical results computed with static model (middle) and with dynamic model (bottom)

One important characteristic of the experimental flow is the apparition of several bubbles at the base of the jet and in the water filling the tank. The Level-Set method is not able to describe such small geometries as it can be seen in figure 2. Nonetheless, this does not seem to affect the general flow behavior.

REFERENCES

[1] H.C. Chen and K. Yu. Cfd simulations of wave-current-body interactions including greenwater and wet deck slamming. *Computers & fluids*, 2008.

[2] R. Codina, , and J. Blasco. Analysis of a stabilized finite element approximation of the transient convection-diffusion-reaction equation using orthogonal subscales. *Comput. Visual. Sci.*, 4(3):167–174, 2002.

[3] R. Codina. Stabilized finite element approximation of transient incompressible flows using orthogonal subscales. *Comput. Methods Appl. Mech. Engrg.*, 191(39-40):4295–4321, 2002.

[4] T. Coupez. Rinitialisation convective et locale des fonctions level set pour le mouvement de surfaces et d'interfaces. *Journées Activités Universitaires de Mécanique - La Rochelle, 31 août et 1er septembre 2006*, 2006.

[5] R. Courant, K. Friedrichs, and H. Lewy. Uber die partiellen diefferenzengleichungen der mathematischen physik. *Mathematische Annalen*, 100(1):32–74, 1928.

[6] M.A. Cruchaga, D. J. Celentano, and T.E. Tezduyar. Collapse of a liquid counn : numerical simulation and experimental validation. *Comput. Mech.*, 39:453–476, 2007.

[7] M. Germano, U. Pionelli, P. Moin, and W.H. Cabot. a dynamic subgrid-scale eddy viscosity model. *Physics of Fluids*, A3:1760–1765, 1991.

[8] E. Hachem. *Stabilized Finite Element Method for Heat Transfer and Turbulent Flows inside Industrial Furnaces.* PhD thesis, Ecole Nationale Supérieure des Mines de Paris, 2009.

[9] E. Hachem, B. Rivaux, T. Kloczko, H. Digonnet, and T. Coupez. Stable mixed-finite element method for incompressible flows with high reynolds number. *submitted to Journal of Computational Physics, December 2009.*

[10] P.D. Hieu, T. Katsutoshi, and V.T. Ca. Numerical simulation of breaking waves using a two-phase flow model. *Applied Mathematical Modelling*, 28:983–1005, 2004.

[11] T. J. R. Hughes, G. R. Feijoo, L. Mazzei, and J.-N. Quincy. The variational multiscale method - a paradigm for computational mechanics. *Comput. Methods Appl. Mech. Engrg.*, 166:3–24, 1998.

[12] F. Ilinca and J.F. Hetu. Finite element solution of three dimensional turbulent flows appied to mold-filling problems. *International Journal for Numerical methods in Fluids*, 34:729–750, 2000.

[13] M.J. Ketabdari, M.R.H. Nobari, and M. Moradi Larmaei. Simulation of waves group propagation and breaking in coastal zone using a navier-stokes solver with an improved vof free surface treatment. *Applied Ocean Research*, 2008.

[14] E. Labourasse, D. Lacanette, A. Toutant, P. Lubin, S. Vincent, O. Lebaigue, J.P. Caltagirone, and P. Sagaut. Towards large eddy simultaion of isothermal two-phase flows : Governing equations and a priori tests. *International journal of multiphase flow*, 33:1–39, 2007.

[15] B.E. Launder and D.B. Spalding. Lectures in mathematical models of turbulence. *Academic Press Inc. (London) Ltd*, 1972.

[16] T. Li, P. Troch, and J. De Rouck. Interactions of breaking waves with a current over cut cells. *Journal of Computational Physics*, 223:865–897, 2007.

[17] D.K. Lilly. A proposed modification of the germano subgrid-scale closure method. *Phys. Fluids*, pages 633–635, 1992.

[18] P. Lubin, S. Vincent, S. Abadie, and J.P. Caltagirone. Three-dimensional large eddy simulation of air entertainment under plunging breaking waves. *Coastal Engineering*, 53:631–655, 2006.

[19] J.C. Martin and M.J. Moyce. Some gravity wave problems in the motion of perfect liquids. *Philos. Trans. Roy. Soc. London Ser.*, 244:231–334, 1952.

[20] J. Smagorinsky. General circulation experiments with primitive equations. *Mon. Weather Rev.*, 91:99–164, 1963.

[21] Tayfun E. Tezduyar and Yasuo Osawa. Finite element stabilization parameters computed from element matrices and vectors. 190(3-4):411–430, 2000.

[22] Laurence Ville, Luisa Silva, and Thierry Coupez. Convected level set method for the numerical simulation of fluid buckling. *International Journal for Numerical Methods in Fluids*, accepted.

[23] S. Vincent, J. Larocque, D. Lacanette, A. Toutant, P. Lubin, and P. Sagaut. Numerical simulation of phase separation and a priori two-phase les filtering. *Computers & fluids*, 37:898–906, 2008.