A compressible Lagrangian framework for the simulation of the underwater implosion of large air bubbles

K. Kamran, R. Rossi, E. Oñate, S.R. Idelsohn

1. Introduction

The underwater implosion of air-filled bubbles has been studied by many authors in the last century due to its main role in a number of phenomena in science, like sonoluminescence, sonochemistry and sonofusion and a series of applications in engineering like cavitation damages, seabed detection and structure safety in the vicinity of the imploded volumes.

Historically, the first work on the cavitation and bubble dynamics was done by Rayleigh [1], who considered the collapse of both empty and gas filled cavities in an inviscid incompressible liquid. Plesset extended his work by adding surface tension and viscous effects that resulted in the famous Rayleigh–Plesset equation [2]. The presence of high pressures in liquid near the interface in addition to the damping oscillations of the bubble lead many authors to take into account for the compressibility of the surrounding liquid in their analysis [3–5]. Depending on the initial radius of the bubble and external driving pressure two types of behavior are observed in the bubble motion, namely weakly oscillating and strongly collapsing.

In physics, violent collapse of μm size bubbles excited by the sound waves may lead to UV-light emission of picosecond duration that is known as sonoluminescence, SL [6]. Super compression of the internal air and high velocities obtained during the final stage of the violent collapse put in doubt the stability of the bubble. Later studies on the shape stability of the bubble [7,8] revealed that the spherical symmetry assumption cannot be rigorously correct especially in the final stage of the violent collapse. Bogoyavlenskii [8] demonstrated that time derivatives of shape perturbations grow significantly as the bubble radius vanishes. In general two different types of instabilities are prone to be excited during this stage, (i) interfacial instabilities, i.e. Rayleigh–Taylor (RT) instabilities, which occur when a gas is strongly accelerated into the liquid, and (ii) shape instabilities due the excitation of non-spherical modes causing the bubble to take on a non spherical shape. The shape instability is well shown in the DNS results provided by Nagrat et al. [9] for a micron-size bubble implosion. Although they predict the RT instabilities during the very short time interval that the bubble radius is near its minimum, no numerical evidence is presented in their results.

Keywords: Lagrangian shock hydrodynamics
Variational multiscale stabilization
Two phase flow
Pfem
Bubble implosion

Article history:
Received 26 March 2012
Received in revised form 22 November 2012
Accepted 24 November 2012
Available online 14 December 2012

Abstract

A fully Lagrangian compressible numerical framework for the simulation of underwater implosion of a large air bubble is presented. Both air and water are considered compressible and the equations for the Rayleigh shock hydrodynamics are stabilized via a variationally consistent multiscale method. A nodally perfect matched definition of the interface is used and then the kinetic variables, pressure and density, are duplicated at the interface level. An adaptive mesh generation procedure, which respects the interface connectivities, is applied to provide enough refinement at the interface level. This framework is verified by several benchmarks which evaluate the behavior of the numerical scheme for severe compression and expansion cases. This model is then used to simulate the underwater implosion of a large cylindrical bubble, with a size in the order of cm. We observe that the conditions within the bubble are nearly uniform until the converging pressure wave is strong enough to create very large pressures near the center of the bubble. These bubble dynamics occur on very small spatial (0.3 mm) and time (0.1 ms) scales. During the final stage of the collapse Rayleigh–Taylor instabilities appear at the interface and then disappear when the rebounce starts. At the end of the rebounce phase the bubble radius reaches 50% of its initial value and the bubble recover its circular shape. It is when the second collapse starts, with higher mode shape instabilities excited at the bubble interface, that leads to the rupture of the bubble. Several graphs are presented and the pressure pulse detected in the water is compared by experiment.

1. Introduction

The underwater implosion of air-filled bubbles has been studied by many authors in the last century due to its main role in a number of phenomena in science, like sonoluminescence, sonochemistry and sonofusion and a series of applications in engineering like cavitation damages, seabed detection and structure safety in the vicinity of the imploded volumes.

Historically, the first work on the cavitation and bubble dynamics was done by Rayleigh [1], who considered the collapse of both empty and gas filled cavities in an inviscid incompressible liquid. Plesset extended his work by adding surface tension and viscous effects that resulted in the famous Rayleigh–Plesset equation [2]. The presence of high pressures in liquid near the interface in addition to the damping oscillations of the bubble lead many authors to take into account for the compressibility of the surrounding liquid in their analysis [3–5]. Depending on the initial radius of the bubble and external driving pressure two types of behavior are observed in the bubble motion, namely weakly oscillating and strongly collapsing.

In physics, violent collapse of μm size bubbles excited by the sound waves may lead to UV-light emission of picosecond duration that is known as sonoluminescence, SL [6]. Super compression of the internal air and high velocities obtained during the final stage of the violent collapse put in doubt the stability of the bubble. Later studies on the shape stability of the bubble [7,8] revealed that the spherical symmetry assumption cannot be rigorously correct especially in the final stage of the violent collapse. Bogoyavlenskii [8] demonstrated that time derivatives of shape perturbations grow significantly as the bubble radius vanishes. In general two different types of instabilities are prone to be excited during this stage, (i) interfacial instabilities, i.e. Rayleigh–Taylor (RT) instabilities, which occur when a gas is strongly accelerated into the liquid, and (ii) shape instabilities due the excitation of non-spherical modes causing the bubble to take on a non spherical shape. The shape instability is well shown in the DNS results provided by Nagrat et al. [9] for a micron-size bubble implosion. Although they predict the RT instabilities during the very short time interval that the bubble radius is near its minimum, no numerical evidence is presented in their results.

1. Introduction

The underwater implosion of air-filled bubbles has been studied by many authors in the last century due to its main role in a number of phenomena in science, like sonoluminescence, sonochemistry and sonofusion and a series of applications in engineering like cavitation damages, seabed detection and structure safety in the vicinity of the imploded volumes.

Historically, the first work on the cavitation and bubble dynamics was done by Rayleigh [1], who considered the collapse of both empty and gas filled cavities in an inviscid incompressible liquid. Plesset extended his work by adding surface tension and viscous effects that resulted in the famous Rayleigh–Plesset equation [2]. The presence of high pressures in liquid near the interface in addition to the damping oscillations of the bubble lead many authors to take into account for the compressibility of the surrounding liquid in their analysis [3–5]. Depending on the initial radius of the bubble and external driving pressure two types of behavior are observed in the bubble motion, namely weakly oscillating and strongly collapsing.

In physics, violent collapse of μm size bubbles excited by the sound waves may lead to UV-light emission of picosecond duration that is known as sonoluminescence, SL [6]. Super compression of the internal air and high velocities obtained during the final stage of the violent collapse put in doubt the stability of the bubble. Later studies on the shape stability of the bubble [7,8] revealed that the spherical symmetry assumption cannot be rigorously correct especially in the final stage of the violent collapse. Bogoyavlenskii [8] demonstrated that time derivatives of shape perturbations grow significantly as the bubble radius vanishes. In general two different types of instabilities are prone to be excited during this stage, (i) interfacial instabilities, i.e. Rayleigh–Taylor (RT) instabilities, which occur when a gas is strongly accelerated into the liquid, and (ii) shape instabilities due the excitation of non-spherical modes causing the bubble to take on a non spherical shape. The shape instability is well shown in the DNS results provided by Nagrat et al. [9] for a micron-size bubble implosion. Although they predict the RT instabilities during the very short time interval that the bubble radius is near its minimum, no numerical evidence is presented in their results.
In engineering applications, on the other hand, larger size bubbles, in the order of cm, are of interest. Acoustic waves emanated from broken glass spheres are used to indicate the contact of the equipment with the seabed. Orr et al. [10] have reported the pressure signatures and energy–density spectra for a series of preweak-end hollow glass spheres implored at ocean depths of approximately 3 km. The signature of all implosions have many features in common. Basically each consists of a low flat negative-pulse followed by a sharp positive-pressure spike of roughly 0.2 ms duration. Different pressure signature is reported for shallow depth implosion, less than 300 m. Here linear oscillation, resembling a strongly damped sinusoid with damping factor of e per oscillation cycle occurs. McDonald et al. [11] propose turbulent instabilities excited by the shape oscillation of the bubble as the decay mechanism in shallow depth. Recently, Turner [12] studied the influence of the structure failure on the pressure pulse. Four glass spheres of diameter 7.62 cm were implored in a pressure vessel at a hydrostatic pressure of 6.996;MPa and the pressure–time histories were compared with numerical results obtained from different failure rates. He reported an error of 44% for models that do not account for structure failure.

The development of efficient algorithms to understand rapid bubble dynamics presents a number of challenges. The foremost challenge is to efficiently represent the coupled compressible fluid dynamics of internal air and surrounding water. Secondly, the method must allow one to accurately detect or follow the interface between the phases. Finally, it must be capable of resolving any shock waves which may be created in air or water during the final stage of the collapse. Regarding the Eulerian approach and for small bubbles, μm-size, Nagrat et al. [9] proposed a DNS solution of the full hydrodynamics set of equations stabilized by the SUPG method. The interface is tracked by a modified level-set method and a DC operator is added to provide smooth transition in shock zones. Surface tension is also included to see its influence on the final shape of the bubble. Concerning large bubbles, in the order of cm, Farhat et al. [13] solved the Euler equation for the multi-fluid problem using a ghost fluid method for the poor (GFMP) generalized for an arbitrary equation of state (EOS). Viscous effect and surface tension are neglected due to the size of the bubbles and an exact Riemann solver is used to resolve the shock at the interface.

Lagrangian frameworks to solve the Euler equations in the presence of the shock waves and with large mesh movement have been developed by different authors. Efforts have been dedicated to improve the robustness of the simulation with respect to mesh distortion, while maintaining second order accuracy in smooth regions of the flow [14–16]. Scovazzi et al. [17,18,14,19,20] developed a robust second-order FEM method with continuous linear approximation of kinematic and kinetic variables that is stabilized by operators driven from the variational multiscale paradigm. This work is recently equipped with a conservative synchronized ALE remap approach to reposition the mesh in distorted zones without changing connectivities [20,21]. In moving Lagrangian curvilinear coordinates, traditional staggered grid hydro (SGH) methods, that use continuous linear representation for kinematic variables and discontinuous constant field for thermodynamic variables, have been extended for higher order elements [22].

All of the above mentioned Lagrangian methods are able to treat air or water phase as well as to represent shock waves in them. None of them, however, is designed to capture possible large distortions of the interface that appear in multi-fluid flow. The recent developments in the Particle Finite Element Method (PFEM) [23–26] to deal with multi-fluid flows provide a good basis to capture interface instabilities in the final stage of the collapse. In particular, Idelsohn et al. [25,26] successfully track the interface in an incompressible heterogeneous flows in the presence of large density and viscosity jumps. Interface is forced to match the nodes and due to the jump in pressure gradient across the interface, a discontinuous pressure gradient projection is necessary to stabilize the flow near the interface.

In this work we present a fully Lagrangian shock hydrodynamics framework to solve two-phase flow problems with large distortions at the interface and big pressure and density jumps. To solve the hydrodynamic set of equations in each phase the stabilized variational multiscale method presented by Scovazzi et al. [17,18,14] is adopted. Later we improve the interface detecting technique proposed by Idelsohn et al. [25,26], by conserving the interface connectivities. This method is then extended to compressible multi-fluid flows by considering a discontinuous representation of the kinetic variables, i.e. pressure and density, at the interface level. The simulation of the large-bubble implosion using the proposed framework allows to identify, to our best knowledge for the first time, the appearance of the RT instabilities in these bubbles, at the final stage of the collapse. The possibility of the appearance of such instabilities has been reported by many authors [2,6,8,9,27]. We continue the simulation during the rebound phase till instabilities disappear and the second collapse occurs. The second collapse ends up with the rupture of the air bubble.

The outline of the paper is as follows: In the next section a review of the stabilized variational multiscale method developed by Scovazzi et al. [17,18,14] to solve the Euler hydrodynamic set of equations is presented. Then, the interface-following technique proposed by Idelsohn et al. [25,26] is presented and extended for compressible flow, emphasizing on the use of a discontinuous pressure along the interface. In the final section we present some numerical examples that verify the method and show its potential for simulating the implosion of a large size cylindrical bubble.

2. Governing equations

2.1. Lagrangian hydrodynamic equations

As we are interested in the simulation of large bubbles, the effect of surface tension and viscosity can be neglected and therefore the Lagrangian compressible hydrodynamic set of equations governing the conservation of mass, momentum and energy is written as,

\[ \rho \frac{\partial \mathbf{u}}{\partial t} + \rho \mathbf{u} \cdot \nabla \mathbf{u} = -\nabla p + \mathbf{f}, \quad \text{in } \Omega, \ t \in [0,T]. \]  
\[ \rho \mathbf{u} \cdot \mathbf{u} = -\nabla \cdot \mathbf{q} + \rho r, \quad \text{in } \Omega, \ t \in [0,T]. \]  
\[ \rho \frac{\partial e}{\partial t} + \rho \mathbf{u} \cdot \nabla e = \frac{1}{2} \rho \nabla \cdot \mathbf{u} \rho, \quad \text{in } \Omega, \ t \in [0,T]. \]  

The gradient derivatives, \( \nabla \), are calculated in the current configuration and ( ) refers to the material time derivative. \( J \) is the deformation Jacobian determinant, \( \rho_0 \) is the reference density, \( \rho \) is the current density, \( \mathbf{u} \) is the velocity vector, \( \mathbf{b} \) is the body force, \( p \) is the thermodynamic pressure, \( r \) is the energy source term, \( \mathbf{q} \) is the heat flux and \( \varepsilon \) is the internal energy per unit mass.

Although the energy equation is not written in the conservative form it can still be used to develop a globally conservative variational formulation [14]. For the sake of simplicity and without loss of generality a homogeneous Dirichlet boundary condition is considered. These equations and the boundary conditions, in addition to an equation of state for the pressure \( p \) and a constitutive law for the heat flux, \( q \), together with the appropriate initial conditions define the evolution of the system. An equation of state of the form \( p = p(\rho, e) \) is assumed. In this case the energy equation (1c) can be reformulated, using the Gibbs identity and some algebra [17] to,

\[ \rho \frac{\partial c_v}{\partial p} \left( \frac{\partial p}{\partial p} + \rho c_v \nabla \cdot \mathbf{u} \right) = 0. \]
Here \( c_s \) is the isentropic speed of sound in the medium. In general for a compressible flow we have, \( \rho \frac{\partial v}{\partial t} \neq 0 \) and hence this term can be simplified in (2). Note that in this case the momentum and energy equations can be combined into the mixed first order wave equation in \( \textbf{u} \) and \( p \), as

\[
p\textbf{u} + \nabla p = 0, \quad p + \rho c_s^2 \nabla \cdot \textbf{u} = 0. \tag{3}
\]

As the hydrodynamic equation (1) are quite nonlinear, we first develop the stabilized variational multiscale form of the linearized version of these equations. Such linearized problem is presented in the next section.

2.2. Stabilization of the linearized form

The linearization is done considering a small-strain approximation in which the derivatives in the reference and current configurations coincide and the solution of the thermodynamic variables, density, pressure and internal energy, is considered as a small perturbation to the constant reference state. As the result, the conservation of mass decouples from the momentum and energy equations and we can focus on these Eqs. (3) with constant coefficients to develop the variational multiscale form.

In the absence of source terms and with homogeneous Dirichlet boundary conditions the equivalent variational form of the linearized Eqs. (3) is read as, find \( \textbf{u} \in \mathcal{V}_u \) and \( p \in \mathcal{V}_p \) such that for all test functions \( \textbf{v} \in \mathcal{V}_u \) and \( q \in \mathcal{V}_p \),

\[
(\rho \textbf{u}, \textbf{v}) - (p, \nabla \cdot \textbf{v}) = 0, \quad (p + \rho c_s^2 \nabla \cdot \textbf{u}, q) = 0. \tag{4}
\]

Here the appropriate space for velocity is \( \mathcal{V}_u = C^0([0, T], \mathcal{H}_0(div, \Omega)) \cap C^1([0, T], L^2(\Omega)) \) and for pressure is \( \mathcal{V}_p = C^0([0, T], L^2(\Omega)) \). By \( C^1([0, T], \mathbb{R}) \) we mean all functions that are \( \mathbb{C}^0 \) continuous in time interval \([0, T]\) and \( \mathcal{H}_0(div, \Omega) \) is the space of functions in \( L^2(\Omega) \) with their derivatives also in \( L^2(\Omega) \) and with zero support in the boundary.

We consider a decomposition of the unknowns to the coarse and fine scales where the coarse scale solution is captured by the finite element and the fine scale is approximated by a piecewise linear polynomial with zero support in the fine scales where the coarse scale solution is captured by the finite element and the fine scale is approximated by the piecewise linear polynomial with zero support in the fine scales. The decomposition has the form,

\[
\textbf{u} = \textbf{u}^h + \textbf{u}' \quad \text{and} \quad p = p^h + p',
\]

and the fine scales \( \textbf{u}' \), \( p' \) are approximated by means of the residuals of the momentum and energy equations [14,30,31] as,

\[
\textbf{u}' = -\tau p^h \frac{\partial \textbf{u}^h}{\partial t}, \quad p' = -\tau p^h. \tag{5}
\]

Here \( \tau \) has dimension of time and the coarse scale residuals at each element are defined as,

\[
\begin{align*}
\mathcal{R}_u^h & := \rho \frac{\partial \textbf{u}^h}{\partial t} + \nabla p^h, \\
\mathcal{R}_p^h & := \frac{\partial \textbf{u}^h}{\partial t} + \rho c_s^2 \nabla \cdot \textbf{u}^h.
\end{align*} \tag{6}
\]

The final stabilized form for the linearized case is written as,

\[
(\rho \textbf{u}^h, \textbf{v}^h) - (p^h, \nabla \cdot \textbf{v}^h) + (p^h + \rho c_s^2 \nabla \cdot \textbf{u}^h, \nabla \cdot \textbf{v}^h) + (\rho c_s^2 \nabla \cdot \textbf{u}^h, \textbf{v}^h) + (p^h + \rho c_s^2 \nabla \cdot \textbf{u}^h, \textbf{v}^h), \tag{7}
\]

The main stabilization effect comes from an incomplete Laplacian of the velocity, \( \nabla_s \cdot \textbf{v}^h \), in the momentum equation and a full Laplacian of the pressure, \( \nabla_s p^h \), in the energy equation. In addition, the presence of the full residual avoids reduction in accuracy order, which is quite common in residually inconsistent stabilizations.

A similar set of stabilized equations can be obtained using the Finite Calculus (FIC) approach [32,33]. In the following section we present the full stabilization for the nonlinear case, inspired in the linearized one previously mentioned.

2.3. Stabilization of the nonlinear form

The variational multiscale stabilization of the linearized case reveals the overall structure of the stabilization method. To elaborate the variational multiscale stabilization of the nonlinear form, first the variational equivalence of the hydrodynamic set of Eq. (1) is presented. We consider homogeneous Dirichlet boundary conditions and \( \textbf{r}, \textbf{q} \) and \( b \) are set to zero. We are interested in finding \( \textbf{u} \in \mathcal{V}_u \) and \( p, \rho, \epsilon \in \mathcal{V}_p \) such that, \( \forall \textbf{v} \in \mathcal{V}_u \) and \( \forall q \in \mathcal{V}_p \)

\[
\int_{\Omega} q (\rho_0 - \rho f) d\Omega = 0, \tag{8a}
\]

\[
\int_{\Omega} \textbf{v} \cdot (\rho \frac{\partial \textbf{u}}{\partial t}) d\Omega + \int_{\Omega} \sigma \cdot \nabla \textbf{v} d\Omega = 0, \tag{8b}
\]

\[
\int_{\Omega} q (\rho \epsilon - \sigma : \nabla \textbf{u}) d\Omega = 0. \tag{8c}
\]

Here \( \nabla \cdot \textbf{u} \) is the symmetric part of the gradient operator and \( \sigma = -p \mathbb{I} \). Piece-wise linear continuous approximation of both kinematic, \( \textbf{u} \), and thermodynamic, \( \rho \), \( \epsilon \), variables are considered. We consider the same decomposition as performed for the velocity and pressure in Section 2.2, but this time the decomposition of the \( \epsilon \) is also added. This gives,

\[
\textbf{u} = \textbf{u}^h + \textbf{u}', \quad p = p^h + p', \quad \epsilon = \epsilon^h + \epsilon'. \tag{9}
\]

Scovazzi [14] consider also the decomposition of \( \rho \) to develop the multiscale method. However, as it has no apparent contribution to the final stabilized form, the density decomposition is omitted here. Furthermore, due to the intrinsic ability of the Lagrangian description to exactly track contact discontinuities without adding numerical dissipation, and as the mass conservation is associated with one of these discontinuities in the form of standing entropy wave, we only focus on the momentum and energy equations to construct the stabilized form. Inserting the decomposition (9) into the variational form \( 8b,8c \), the coarse scale equations is formulated as,

\[
\int_{\Omega} \textbf{v} \cdot (\rho_0 (\textbf{u}^h + \textbf{u}')) d\Omega - \int_{\Omega} (p^h + p') (\nabla \cdot \textbf{v}) d\Omega = 0 \tag{10}
\]

\[
\int_{\Omega} \textbf{v} \cdot (\rho \frac{\partial \textbf{u}^h}{\partial t} + \rho \frac{\partial \textbf{u}'}{\partial t}) d\Omega + \int_{\Omega} q (p^h + p') \nabla \cdot (\textbf{u}^h + \textbf{u}') d\Omega = 0. \tag{11}
\]

Using integration by parts, the energy equation (11) can be reformulated as,

\[
\begin{align*}
\int_{\Omega} q (\rho (\epsilon^h + \epsilon')) d\Omega + \int_{\Omega} q (p^h + p') \nabla \cdot \textbf{u} d\Omega & - \int_{\Omega} \nabla_s q^h \cdot (p^h + p') \textbf{u} d\Omega - \int_{\Omega} q^h (\nabla (p^h + p')) \cdot \textbf{u} d\Omega \\
& + \sum_{j=1}^{n_e} \int_{\Gamma_{hj}} q^h (p^h + p') \textbf{u} \cdot \textbf{n} \, d(\Gamma_{hj}) = 0. \tag{12}
\end{align*}
\]
Last term in (12), which is a boundary integral, has two terms; one inter-element boundary term and one domain boundary term. Similar to the linearized case, some simplifications are considered. The fine scale solution, \( u' \), is considered to be zero on the domain boundary. On the other hand, assuming a continuous solution for \( p, u \) and a continuous approximation for \( u' \), the fine scale solution turns out to be continuous in the inter-element boundary too i.e. \( u^s = u - u' \). As the result, last term in (12) is zero. Furthermore and to preserve the global conservation [14], the internal work done by the fine-scale velocity, \( u' \), is assumed to be zero in a weak sense i.e.

\[
\int_\Omega q^s (\nabla (p^s + p') \cdot u') \, d\Omega = 0.
\]

We again consider the quasi-static tracking of the subscales and neglect the product of the fine-scale terms. The simplified form of the exact coarse scale Eqs. (10) and (11) can be written now as,

\[
\begin{align*}
\int_\Omega & \left( \rho \left( \frac{\partial u'}{\partial t} \right) \right) \, d\Omega - \int_\Omega (p^s + p') \left( \nabla \cdot u' \right) \, d\Omega = 0, \\
\int_\Omega & q \left( \rho \left( \frac{\partial u'}{\partial t} \right) + p^s \nabla \cdot u' \right) \, d\Omega + \int_\Omega q (p^s + p') \nabla \cdot u' \, d\Omega \\
& - \int_\Omega \nabla \cdot q \left( \rho \left( \frac{\partial u'}{\partial t} \right) \right) \, d\Omega = 0.
\end{align*}
\]

(13)

Certain similarities are observed comparing the nonlinear coarse-scale Eqs. (13) and their linear counterpart [14]. In particular the momentum equations are practically the same and also recalling (2),

\[
\rho \varepsilon^h + p^h \nabla \cdot u = \rho \frac{\partial e}{\partial p} \cdot \left( \dot{p} + \rho \varepsilon^h \nabla \cdot u \right),
\]

(14)

we observe similarities in the linear and nonlinear energy equations. These similarities give clue to design approximation for the subscales \( u' \) and \( p' \). Note that in comparison to the linearized case (7) a new stabilization term of the form,

\[
\int_\Omega q \left( \frac{\partial u'}{\partial t} \right) \nabla \cdot u' \, d\Omega
\]

appears. This term has a fundamental role in global conservation of total energy [14].

We complete our multiscale stabilization by writing the counterpart of Eqs. (10) and (11), this time tested by the fine scales and rearranged as.

\[
\begin{align*}
\int_\Omega & \left( \frac{\partial u'}{\partial t} \right) \cdot u' \, d\Omega - \int_\Omega p \left( \nabla \cdot u \right) \, d\Omega = -\langle R_{u^s}^h, u' \rangle, \\
\int_\Omega & q \frac{\partial e}{\partial p} \cdot \left( \dot{p} + \rho \varepsilon^h \nabla \cdot u \right) \, d\Omega + \int_\Omega q \left( \frac{\partial u'}{\partial t} \right) \cdot (u' + u) \, d\Omega = -\langle R_{q^s}^h, q' \rangle.
\end{align*}
\]

(15)

where \( < R_{u^s}^h > \) and \( < R_{q^s}^h > \) belong to the dual space of \( v' \) and \( q' \) respectively. They are calculated by doing integration by parts and neglecting the boundary terms, i.e.

\[
\langle R_{u^s}^h, v \rangle := \int_\Omega \left( \frac{\partial u'}{\partial t} \right) \cdot u' \, d\Omega
\]

\[
\langle R_{q^s}^h, q \rangle := \int_\Omega q \left( \frac{\partial u'}{\partial t} \right) \cdot \nabla \cdot u' \, d\Omega.
\]

Similar to the linearized case and by means of the Riesz representation theorem, the coarse scale residuals, \( \rho \varepsilon^h u + \nabla \cdot p^h \) and \( \rho \varepsilon^h + p^h \nabla \cdot u' \), can be identified as the members of \( (L^2(\Omega))^h \) and \( L^2(\Omega) \) respectively. We need to approximate subscales from the fine scale Eq. (15) and plug them into the coarse scale Eq. (13). Note that the only fine scales that appear in (13) are \( u' \) and \( p' \). Recalling Eq. (14), there is a one by one relationship between \( R_{u^s}^h \) and \( R_{p^s}^h \), defined in (6), so we come out with the same expression as the linear case to approximate \( p' \), i.e.

\[
p' = -\tau R_{p^s}^h
\]

(16)

Although \( u' \) can be approximated invoking the same rationalism applied in the linearized case, i.e. \( u' = -\tau \frac{1}{2} R_{u^s}^h \), and it is still dimensionally consistent, a more delicate structure could be obtained considering the construction of the term \( \int_\Omega \nabla q^s \cdot (p'^s u') \) that appears in the coarse scale Eq. (13). The alternative form we propose for the velocity subscales to be substituted into the coarse scale Eq. (13) is,

\[
p'^s u' = -\tau (\varepsilon_{i j}^h) R_{p^s}^h,
\]

(17)

where \( \tau \) has the dimension of time and is taken as \( \Delta t \). An intuitive interpretation of this choice is to note that \( u' \propto R_{u^s}^h \) and \( p \propto \varepsilon_{i j}^h \).

Introducing the fine scales approximation (16), (17) into (13), the stabilized variational form of the nonlinear hydrodynamic equations is summarized as,

\[
\begin{align*}
\int_\Omega & \left( \frac{\partial u'}{\partial t} \right) \cdot u' \, d\Omega - \int_\Omega p \left( \nabla \cdot u \right) \, d\Omega + \int_\Omega \nabla \cdot u' \, d\Omega \\
& + \int_\Omega q \left( \frac{\partial u'}{\partial t} \right) \cdot (u' + u) \, d\Omega = 0,
\end{align*}
\]

(18)

\[
\begin{align*}
\int_\Omega & q \left( \frac{\partial u'}{\partial t} \right) \cdot \nabla \cdot (u' + u) \, d\Omega \\
& + \int_\Omega \varepsilon_{i j}^h R_{p^s}^h \nabla \cdot u' \, d\Omega = 0.
\end{align*}
\]

(19)

Note that the first integral in energy equation (19) can be replaced by means of Eq. (14) to get a \( u, p \) form similar to the linearized case (7). Also by means of (17), all stabilization terms are scaled by the acoustic type kinematic viscosity \( \varepsilon_{i j}^h \) as mentioned in [14]. Finally the presence of a full residual provides a variationally consistent stabilization that does not reduce the order of accuracy.

The formulation presented above does not apply to the regions where strong discontinuities or shocks appear. In this case a discontinuity capturing operator in the form of artificial viscosities is introduced.

2.4. Discontinuity-capturing (DC) operator

In practice we compute an artificial symmetric stress tensor, \( \nabla \cdot u \), and an artificial flux vector, \( \nabla \cdot p \), that will be activated in the compression zones. The discontinuity capturing operator is then given by,

\[
\sigma = \begin{cases} 
\rho \nabla \cdot u, & \nabla \cdot u < 0, \\
0, & \text{else},
\end{cases}
\]

\[
\lambda = \begin{cases} 
\nabla \cdot p, & \nabla \cdot u < 0, \\
0, & \text{else},
\end{cases}
\]

(20)

where \( \sigma \) is similar to the quadratic viscosity of von Neumann– Richtmyer [34] type and \( \lambda \) provides stabilization on the pressure in the energy equation. The artificial viscosities are,

\[
\begin{align*}
\nabla \cdot u &= C_1 |\nabla \cdot u|^2 h_u^{1/2}, \\
\nabla \cdot p &= C_2 \left( \frac{\nabla \cdot p}{\rho} \right) h_u^{1/2},
\end{align*}
\]

(21)

\( \nabla \cdot p \) is zero when the pressure is constant and so it does not affect the solution at the contact discontinuities and \( h_u \) is a measure of the element length along the normal to the shock front. The same definition as [14] is chosen here to define the normal direction, that is a weighted average of the direction of the acceleration vector and the density gradient, as follows,

\[
\mathbf{n} = \frac{0.75 \nabla p + 0.25 \frac{\nabla \cdot u}{|\nabla \cdot u|}}{|0.75 \nabla p + 0.25 \frac{\nabla \cdot u}{|\nabla \cdot u|}|}
\]

(22)

and subsequently \( h_u \) is defined as,
\[ h_x = \frac{2}{\sqrt{n^T (F^F)n}}, \quad F = \frac{\partial X}{\partial \xi}, \]

where \( F \) is the Jacobian of the gradient deformation from the parent domain of the element to the current one and the denominator in (23) is a measure of the stretch along the direction of the normal \( n \). A \( h_x \) as defined in [19].

\[ h_x = C \sqrt{A}. \]  

Here \( A \) is the area of the element and \( C \) is a scaling factor. Recent developments on the definition of artificial viscosity for tetrahedral grids can be found in [19].

In the next section we present an explicit iterative time integrator for the stabilized nonlinear hydrodynamics equations (18) and (19). A study of the conservation properties of this scheme can be found in [17,18,14].

2.5. Explicit predictor-multi-corrector time scheme

Considering the stabilized form (18) and (19), the subscales Eqs. (16) and (17) and the DC operator (20), the final form of the hydrodynamic Eq. (8) can be written as,

\[ \int_{\Omega_i} \left( \nu \cdot \rho \frac{\partial \mathbf{u}}{\partial t} \right) d\Omega = 0, \]

\[ \int_{\Omega} \left( \nabla \cdot \mathbf{u} \right) d\Omega + \int_{\Omega} (\sigma + \sigma_d) : \nabla \mathbf{u} = 0, \]

\[ \int_{\Omega} \left( \nu \cdot \partial \mathbf{u} \right) d\Omega - \int_{\Omega} \left( \nabla \cdot \mathbf{u} \right) d\Omega = 0, \]

where \( \sigma = (p^d + p^e) \mathbf{I}, \lambda = p^e \mathbf{u} \) and \( \lambda_6 \) is the artificial, shock-capturing vector introduced in (20).

The system of Eq. (25) is now discretized in time. An explicit predictor/multi-corrector strategy applied to the mid-point scheme is used. In this way the conservation of mass, momentum and energy are enforced not only at the end of step but also at each nonlinear iteration [14]. Let us introduce the following notation for the mid-point value of a quantity \( f \),

\[ f_{n+1/2} = f \frac{n + f_{n+1}}{2}, \]

where \( f_{n+1} = f(t_n) \). The discretized equivalence of the mass conservation equation is,

\[ \int_{\Omega} \nu \cdot \mathbf{u} d\Omega = \int_{\Omega} \mathbf{u} d\Omega = \int_{\Omega} \mathbf{u} \cdot \frac{\partial \mathbf{u}}{\partial t} d\Omega \]

and therefore the nodal densities at time step \( n + 1 \), \( \rho_{n+1} \), are computed as

\[ V_{n+1} \rho_{n+1} = M_0. \]

\( M_0 \) is the vector of nodal mass and its \( i \)th component, \( m_i \), is defined as

\[ m_i = \int_{\Omega} N_i \rho_0 d\Omega. \]

Here \( N_i \) is the global shape function related to node \( i \). In the same way the lumped nodal volume matrix, \( V_{n+1} = \text{diag}(V_{n+1}) \), is defined as

\[ V_{n+1} = \{V_{n+1}\}, \quad V_{n+1} = \int_{\Omega} N_i d\Omega_{n+1}. \]

Applying the mid-point rule to the momentum balance in (25) we have,

\[ \int_{\Omega} \mathbf{u} \cdot \mathbf{u} + \rho \mathbf{u} d\Omega = \int_{\Omega} \mathbf{u} d\Omega = \int_{\Omega} \mathbf{u} \cdot \frac{\partial \mathbf{u}}{\partial t} d\Omega. \]

The matrix form of the momentum Eq. (29) is then written as,

\[ M_0 \mathbf{u}_{n+1} + \Delta t F_{n+1/2} = 0. \]

Here \( \mathbf{u} \) is the nodal vector of velocity and recalling definition (28), the lumped matrix \( M_0 \) is,

\[ M_0 = \text{diag}(M_0). \]

The definition of \( F_{n+1/2} \) is clear from (30) and (29) and has the following form for node \( i \),

\[ F_{n+1/2} = \{F_{n+1/2}\}, \]

\[ F_{n+1/2, i} = \int_{\Omega} (\sigma : \nabla \mathbf{u}_i) d\Omega. \]

The discretization of the energy equation (25) is done in a similar way. We have,

\[ \frac{1}{2} \int_{\Omega} \mathbf{u} \cdot \mathbf{u} d\Omega = \int_{\Omega} \mathbf{u} \cdot \frac{\partial \mathbf{u}}{\partial t} d\Omega. \]

Note that all integrals calculated in the domain \( \Omega_{n+1/2} \), in Eqs. (36) and (33), are understood in the sense of Eq. (30).

As any Lagrangian formulation the position of the nodes need to be updated. This can be done integrating in time the set of equations for nodal displacement to get,

\[ \mathbf{x}_{n+1} = \mathbf{x}_n + \Delta \mathbf{u}_{n+1/2}. \]

Once the updated value for \( \mathbf{x}_{n+1} \) and \( \rho_{n+1} \) has been obtained the nodal pressures are updated by means of the EOS at each nodes,

\[ p_{n+1} = \frac{\rho}{\rho_{n+1}}(\rho_{n+1}, \mathbf{e}_{n+1}). \]

and a piece-wise continuous pressure field is constructed by updated nodal values.

The update of velocity and internal energy from (31) and (35) needs the knowledge of step \( n + 1 \). A fully explicit predictor/multi-corrector strategy is adopted to avoid the inverse of systems and also to best handle the nonlinearities in the calculation of the force and work terms.

Table 1 shows the predictor/multi-corrector algorithm. Let us introduce \( \mathbf{Y} = \begin{bmatrix} \rho & \mathbf{u} & \mathbf{e} & \mathbf{P} & \mathbf{X} \end{bmatrix}^T \) as the state of the system at time \( t \) where \( \mathbf{P} \) is the vector of nodal pressures. The mid-point values at each iteration are calculated form the previous step and the current iteration data.
3.2. Discontinuous kinetic field

The preceding definition of the interface has been successfully tested for incompressible heterogeneous flows with large density and viscosity jumps [25,26]. For incompressible heterogeneous flows a viscosity jump results in a discontinuous pressure across the interface. The effect of the interface can be introduced. This interface passes through elements having two different nodal flags, i.e., air and water. It is already shown in [25] that this definition of the interface results in a spurious pressure field near the interface due to the inability of linear elements for capturing discontinuous pressure gradients. In the Eulerian context different remedies are presented to resolve this problem that end up with adding new degrees of freedom along the elemental interface [36,37].

Beside the elemental interface, a nodally perfect matched definition of the interface is obtained by considering one of the phases as the dominant one in the two-phase flow system. Regarding the two-phase system of Fig. 1 we choose water as the dominant flag. This implies that those elements that have at least one node as water are colored as water elements and only those elements that have all nodes as air are colored as air elements. The nodally perfect matched interface now appears as the boundary layer that separates air and water elements. As the initial position of the interface is known, the choice of the dominant flag is done in such a way that the resulting nodal interface matches the initial interface at the beginning.

### Table 1

Mid-point explicit predictor/multi-corrector algorithm.

<table>
<thead>
<tr>
<th>Calculate $M_i, M_0$ and $M$.</th>
</tr>
</thead>
<tbody>
<tr>
<td>For $n = 0, \ldots, n_{\text{steps}}$ (Begin time step loop)</td>
</tr>
<tr>
<td>Calculate $\Delta t$ by CFL condition</td>
</tr>
<tr>
<td>Predict: $Y_{n+1}^i - Y_i$</td>
</tr>
<tr>
<td>For $i = 1, \ldots, n_{\text{ele}}$ (Begin multi-corrector loop)</td>
</tr>
<tr>
<td>Assemble: $F_{n+1}^i$</td>
</tr>
<tr>
<td>Velocity update: $u_{n+1}^i = u_n - \Delta t [M_n]^{-1} F_{n+1}^i$</td>
</tr>
<tr>
<td>Position update: $X_{n+1}^i = X_n + \Delta t u_{n+1}^i$</td>
</tr>
<tr>
<td>Volume update: $V_{n+1}^i = V(X_{n+1}^i)$</td>
</tr>
<tr>
<td>Density update: $\rho_{n+1}^i = [M_n]^{-1} M_0$</td>
</tr>
<tr>
<td>Assemble: $W_{n+1}^i$</td>
</tr>
<tr>
<td>Energy update: $\epsilon_{n+1} = \epsilon_n - \Delta t [M_n]^{-1} W_{n+1}^i$</td>
</tr>
<tr>
<td>Pressure update: $P_{n+1}^i = -\rho(\rho_{n+1}^i, \epsilon_{n+1})$</td>
</tr>
<tr>
<td>End (multi-corrector loop)</td>
</tr>
<tr>
<td>Update data base: $Y_{n+1}^i = Y_{n+1}^i$</td>
</tr>
<tr>
<td>End (Time step loop)</td>
</tr>
</tbody>
</table>

In the context of incompressible flow, multi-fluid systems with large density jumps and interface distortion have been successfully resolved via the Particle Finite Element Method (PFEM) [23–26]. As a jump in density results in the continuous pressure field but discontinuous pressure gradient across the interface, the PFEM has a nodally perfect matched definition of the interface to avoid any duplication of pressure and applies a discontinuous pressure gradient projection to stabilize oscillations around the interface. The particle nature of the Lagrangian formulation in which material properties are assigned to each particle, provides simple definitions of the interface. Fig. 1 shows the mesh configuration around the interface for a two-fluid system of air and water. Mesh nodes, that represent particles, are flagged as either air or water. At this point, and without any extra assumption, an elemental definition for the interface can be introduced. This interface passes through elements having two different nodal flags, i.e., air and water. It is already shown in [25] that this definition of the interface results in a spurious pressure field near the interface due to the inability of linear elements for capturing discontinuous pressure gradients. In the Eulerian context different remedies are presented to resolve this problem that end up with adding new degrees of freedom along the elemental interface [36,37].

![Fig. 1. Nodally matched interface.](image)
kinetic field i.e. air-pressure for air elements and water-pressure for water elements. On the other hand, having all kinetic and kinematic variables carried by the particles, the duplication of the kinetic fields, pressure and density, at the interface particles suffices to yield air-field and water-field variables.

This duplication can be also seen as the result of two types of elements, air-element and water-element, being connected together and through the interfacial common nodes. Note that here particles are not being duplicated at the interface but just the kinetic variables are. Hence, comparing to the elemental definition of the interface, a minimum number of degrees of freedom is added to capture the discontinuity.

3.3. Mesh construction

An appropriate mesh construction technique is necessary to precisely follow the evolution of the interface at the presence of the large distortions of the FE mesh.

Two major properties of this mesh construction technique are, first, conserving the connectivities of the interface as much as possible from one mesh configuration to the other one and second, adaptive refinement of the mesh in the vicinity of the interface.

Fig. 2 shows an undesirable change at the position of the interface that may happen from one mesh configuration to the another. As we can see the position of the cloud of the nodes does not change considerably from one configuration to the other one, as we may expect for two successive solution steps, but it is quite common that some connectivities change. If this “trivial” change of connectivity is allowed to occur at the interface, it would imply an abrupt change in its position yielding loss of the mass in one fluid and even provoke local instabilities. A constrained construction of the mesh that respects connectivities at the interface can avoid these difficulties and allows us a more precise tracking of the interface, as is desired to detect RT instabilities.

Concerning adaptive mesh refinement, a distance function that vanishes at the interface is used. Later a length parameter, \( H \), calibrated by the distance function is assigned to each node of the mesh. The idea is simply to provide a fine mesh near the interface and a coarse one in farther zones. Therefore, the calibration function, \( f \), can be chosen as a linear function of the distance, \( d \), a step function or simply any other function that provides desired refinement at the interface zone.

\[
H = f(d).
\]

For 2D case problems we choose the Triangle [38] software to build the FE mesh at each step. This mesh generator permits an adaptive constrained Delaunay triangulation of the domain. This is necessary to conserve the connectivities at the interface, as much as possible, and refine the mesh. Interface connectivities are stored in the segment list provided by Triangle and then the constrained remeshing is performed as described in [38]. It is necessary to mention that new nodes are added automatically to the interface at the moments of expansions of the interface to maintain the same mesh size at the interface level. Adaptive refinement is always done by use of a primary mesh. This mesh could be the previous step mesh or one that is created at the beginning of the remeshing phase. To each element of the base mesh an area, \( A \), is assigned. This area would be the maximum area that is expected after the refinement of the element. The length parameter, \( H \), that has been already assigned to each node is used to compute the maximum area that is the one of the equilateral triangle with the edge size equal to the mean value, \( \overline{H} \), of the nodal length parameters, \( H \), as

\[
A = \frac{\overline{H}^2 \sqrt{3}}{4}.
\]

Table 2 summarizes different steps of the mesh generation process. Note that as we mentioned before, the refinement is done by use of a base mesh that is created in the fifth step. The database for the new nodes that are created during the refinement process are interpolated from the database of the base mesh i.e. each new node falls in one of the elements of the base mesh and therefore its database is filled by a linear interpolation of the databases of the nodes of that element. Having the nodally perfect matched interface, the material flag of a new node is easily inherited from the material flag of the base element.

<table>
<thead>
<tr>
<th>Table 2</th>
<th>Constrained mesh generation process</th>
</tr>
</thead>
<tbody>
<tr>
<td>Remesh:</td>
<td>Calculate distance and assign length parameter, ( H ).</td>
</tr>
<tr>
<td></td>
<td>Mark nodes.</td>
</tr>
<tr>
<td></td>
<td>Detect interface connectivities (fill segment list).</td>
</tr>
<tr>
<td></td>
<td>Delete nodes if are assigned by Mark node process.</td>
</tr>
<tr>
<td></td>
<td>Generate a constrained triangulation of the domain.</td>
</tr>
<tr>
<td></td>
<td>Assign area constraint to each element.</td>
</tr>
<tr>
<td></td>
<td>Generate the final refined mesh respecting.</td>
</tr>
<tr>
<td></td>
<td>Area and segment constraints.</td>
</tr>
<tr>
<td></td>
<td>Interpolate for new added nodes.</td>
</tr>
</tbody>
</table>

4. Numerical examples

In this section we verify the Lagrangian compressible framework proposed in previous sections for a series of benchmark examples and then simulate the underwater implosion of large bubbles, in the order of cm. Three-noded linear isotropic triangular elements are used in all examples.

As the 1D hydrodynamics of the bubble predicts a successive chain of contraction–expansion of the bubble, we choose Noh [39] (Fig. 3) and Sedov [40] (Fig. 4(a)) tests to verify our formulation in severe compression and expansion flow regimes. In the contraction phase, called implosion, converging pressure waves results in a shock created at the center of the bubble. Noh test, designed to study in detail the conversion from kinetic energy into internal energy, is a challenging test for compressible codes and is studied in
the first section. The reaction to the contraction phase is an expansion (explosion) manifested by pressure waves traveling spherically in the domain. Sedov proposed a self-similar solution for the state variables that define the perturbation produced by a detonation at the center of the symmetry. The standard [41] version of his test is studied in the second section to verify the behavior of the code in diverging wave examples. Both tests are done in 1D and 2D configurations and with fine and coarse meshes. A 1D air–water (Fig. 5) tube test with jumps of several orders of magnitudes in density and pressure examines the robustness of the discontinuous multiphase scheme proposed in Section 3. Finally the underwater implosion of a cm size cylindrical bubble is studied in the last section. We observe instabilities at the interface of the bubble in the implosion of a cm size cylindrical bubble is studied in the last section. We observe instabilities at the interface of the bubble in the final stage of the collapse phase that disappear at the beginning of the expansion phase. The bubble finally ruptures at the end of the second expansion–contraction cycle. We verify our simulation by comparing the pressure pulse detected in water for the air bubble implosion (Fig. 6) with the experiment [12] provided by Terner.

Our formulation is implemented in the free source parallel multi-physics software platform of KRATOS [42,43] developed at CIMNE. The pre and post processing of data is provided by GID [44] software.

4.1. Noh test

Noh test [39] models the conversion of kinetic energy to internal energy. The setup of the test (see Fig. 3) consists of a shock tube in which a cold, uniform perfect gas is driven by a piston with constant speed into a rigid wall. It can be easily generalized to 2D and 3D cases if we identify the rigid wall with the axis of a cylinder or the center of a sphere, and the piston with their respective outer walls.

The shock tube has the length of unity and the piston moves with the velocity of \( u_0 = -1 \text{ m/s} \) towards the wall. Density is uniform everywhere and is equal to 1, \( \rho = 1.0 \text{ kg/m}^3 \). The initial pressure and internal energy are zero along the tube. At the instance the piston is pushed inward, a stagnation shock of infinite strength is created at the origin (rigid wall). Zero pressure in the pre shock zone results in zero sound velocity and therefore the Mach number of the shock is infinity, something that makes this problem challenging to numerical methods.

The gas is ideal with \( \gamma = 5/3 \) and the analytical solution suggests that the shock will propagate with the velocity equal to \( \frac{1}{4} u_0 \) (\( n_d = \frac{3}{2} u_0 / 2 \)). The exact values for pressure and density in the past shock zone are \( p = 4^{\alpha} / 3 \) and \( \rho = 4^{\alpha} \), respectively, where \( n_d \) is 1 for the 1D case and 2 for the 2D one. Considering the shock speed of \( 1/3 \), we expect to see the shock wave at radial position of \( 0.2 \text{ m} \) from the center of the shock at the time instant \( t = 0.6 \text{ s} \). The results for the 1D simulation can be seen in Fig. 7. The exact values for pressure, density, velocity and internal energy are shown in solid lines. The pressure value for the pre shock zone is \( p = 4 / 3 \) and a constant density equal to 4 is expected in this zone. Results for the 50 and 100 element meshes are compared with the exact values [39].

The computational domain for the 2D case is a \([0, 1] \times [0, 1] \) square domain divided once into 50 and once into 100 triangular elements along each side. Initial values for the pressure and density are the same as for the 1D case. All nodes except the one of interest compared with the results obtained for the coarse and fine mesh along the radius. Note the exact pressure distribution, \( p = 1 + 0.6 / r \), that is captured in the pre shock zone.
Good agreement between the numerical and exact solution is obtained for both position of the shock and the pre and post shock zone values. Note that the shock capturing terms are just activated in some elements along the shock. The known wall heating phenomenon [45] is also seen here in the excessive values of the internal energy at the center.

4.2. Sedov test

Sedov [40] provided analytical solution for the motion of a gas initially at rest and being disturbed by a detonation at the center of symmetry (Fig. 4(a)). We choose here the constant initial density case among the Sedov setting tests for which the initial density,
$\rho_0 = 1 \text{ g/cm}^3$, is uniform and constant everywhere, the gas is ideal with $\gamma = 7/5$ and the internal energy, $e$, is zero everywhere except in a small zone at the center of symmetry. Planar and cylindrical tests are considered for which $e = 0.0673185$ for the planar case, and $e = 0.311357$ for the cylindrical one, in some elements near the center. These values are chosen in accordance with [41] to be able to reproduce the exact self-similar Sedov solutions. For the 1D case a 1 m long tube as shown in Fig. 4(b) is discretized with triangular elements. Two mesh sizes of 0.01 and 0.02 are considered. The left wall has a fixed velocity and the rest of the domain has zero initial velocity. The shock, instantaneously created at the left wall, reaches the middle of the tube at time $t = 1.0$ s. The mesh configuration at this instant is shown in Fig. 4(c). In Fig. 10 the self-similar exact solutions for the state variables (solid lines) are compared with the numerical ones at $t = 1.0$ s. The extension to the cylindrical case is obtained by considering a square domain of $[0.12] 	imes [0.12]$. Two Cartesian meshes, with the same element sizes as the 1D case, are considered. The initial internal energy is zero everywhere except at the center of symmetry where an initial value of $e = 0.311357$ is assigned to few elements near the center. Zero velocity is considered everywhere and the initial density is equal to 1. After 1 s the circular detonation shock wave reaches to the radial position of 0.75. This pressure wave and the current mesh configuration can be seen in Fig. 11. Note that no remeshing is applied to the Sedov and Noh tests. Fig. 12 compares the exact [41] and numerical solutions obtained for the two mesh resolutions.

4.3 Air–water system with big density jump

The two-phase flow with large density, pressure jumps and two different EOSs is considered. Air is modeled as a perfect gas with $\gamma = 1.4$ and the high pressure water is modeled by the Tait’s EOS with $k_1 = 2.7 \times 10^9$, $k_2 = 7.15$, as

$$\rho = \rho_0 \left( \frac{p + k_1/k_2}{P_0 + k_1/k_2} \right)^{\gamma}$$

The same setting as the one proposed by Farhat et al. [13] is used. A tube is filled with air on the left side, a membrane positioned at $x = 0.3$ and water on the right side (Fig. 5). Both fluids are initially at rest. A density jump of order 1000 and a pressure jump of order 100 is produced. The discontinuous pressure scheme explained in Section 3 allows us to initialize interface nodes for both the water and air initial pressures. The computations are done in a 1 m long tube discretized by 200 three-noded triangles. Fig. 13 depicts the results obtained at $t = 4 \times 10^{-4}$ s. The structure of the solution consists of a shock wave traveling in the air, a contact discontinuity, manifested in the density graph, and a rarefaction wave propagating in the water.
4.4. Underwater implosion of cylindrical bubble

In this section we study the implosion of an infinitely long cm size cylindrical bubble which is initially at atmospheric pressure and is being compressed by the surrounding pressurized water at 70 atmospheres. The initial radius of the bubble is 3.81 cm and the numerical domain is a $1 \times 1$ m square (Fig. 6). As we are interested to study the possible instabilities that may occur during the collapse, a full model is considered. The inside air is modeled as a perfect gas with $\gamma = 1.4$. Water is modeled by Tait’s equation of state presented in Section 4.3.

Adaptive mesh refinement is considered in three levels and with respect to the distance calculated from the air–water interface. We seek to obtain a constant number of elements at the inter-

Fig. 10. 1D Sedov test. Results for 60, “•”, 120, “•”, element meshes. Figures (a)–(d) show the exact solution in solid line for pressure, density, internal energy and velocity in comparison with the numerical ones. Artificial viscosities are shown in figures (e) and (f).

Fig. 11. 2D Sedov test. Pressure field and mesh configuration at time $t = 1.0$ s.
face level to cope with the dramatic change of bubble radius, variations up to 90% of the initial diameter, during the collapse and rebound phases. In this way starting from a fine mesh at the beginning of the solution, the quality of the results is guaranteed during the simulation.

Fig. 14(a) shows the time histories of the $x$ and $y$ values of the bubble radius along with the pressure at the center of the bubble. The set up of the test is the same as the underwater implosion at a depth around 700 m and therefore the behavior expected for the bubble is the violent collapse that consists of one or two pulses leading to the collapse. The variation of the length parameter, $H$, of a sample node on the interface in time is shown in Fig. 14(b). This parameter is used to refine the mesh in the vicinity of the interface, as explained on Section 3.3. Interface values of the $H$ have to be a function of the bubble radius to provide enough refinement during the contraction and expansion phases. As the minimum element size is required at the interface level, the minimum length parameter, $H_{min}$, is assigned to the first layer near the interface and then two more levels of refinement, based on the distance from the interface, are considered. $H_{max}$ is chosen in a way that the mesh in far distant zones, where no considerable movement occurs, remains unchanged. Fig. 15(a) and (b) show the mesh near the interface at the beginning of the simulation and at the moment of maximum contraction. The same level of zoom is chosen for both cases for depicting the variation of the bubble radius during the collapse. Note that a refined mesh is generated near the interface and derefinement occurs in the zones far from the interface.

The behavior of the bubble can be divided into three phases. The first one is the collapse of the air bubble due to the large external pressure exerting on it. Fig. 16 depicts the pressure distribution along the radius and the interface position for different instances prior to the collapse. These converging pressure waves induce the shock at the center of the bubble. The appearance of the instabilities and deviation from the circular shape can be also seen in the evolution of the velocity in the $x$ and $y$ directions shown in Fig. 17. The velocity profiles have the same shape till some instances prior to the collapse but then a different behavior appears at the collapse moment when the instabilities occur.

The collapse phase ends up with the maximum reduction in the radius up to 90% of its initial value and the drastic increase in the air pressure (Fig. 14(a)). Considerable change in the bubble radius and the appearance of instabilities in the air–water interface dur-

\[
H(d) = \begin{cases} 
H_{min}, & d < 10H_{min}, \\
H_{min} + 0.25(d - 10H_{min}), & 10H_{min} < d < 50H_{min}, \\
10H_{min} + (d - 50H_{min}) \frac{H_{max} - 10H_{min}}{250}, & 50H_{min} < d < 300H_{min}, \\
H_{max}, & d > 300H_{min}, 
\end{cases}
\]

where $d$ is the distance from the interface and $H_{max}$ is the element size far from the interface zone. Here $H_{min}$ is assigned to the first layer near the interface and then two more levels of refinement, based on the distance from the interface, are considered. $H_{max}$ is chosen in a way that the mesh in far distant zones, where no considerable movement occurs, remains unchanged. Fig. 15(a) and (b) show the mesh near the interface at the beginning of the simulation and at the moment of maximum contraction. The same level of zoom is chosen for both cases for depicting the variation of the bubble radius during the collapse. Note that a refined mesh is generated near the interface and derefinement occurs in the zones far from the interface.
ing a very short instance of the maximum pressure are quite noticeable (Fig. 18). The second phase starts with the shock wave, for an intensity of around 2500 times the initial air pressure, moving outward from the center of the bubble. The radius of the bubble starts to increase and the acceleration of the high pressure air into the water produces instabilities (Fig. 18). Note that the surface instabilities just appear in a very short period, less than 0.1 ms, and then the bubble recovers its circular shape with some oscillations remaining in its interface. The next expansion of the bubble starts at the final part of the second stage, as the result of the reduction in water pressure at the post shock zone. At the end of this stage the bubble has recovered 50% of its initial radius (Fig. 14(a)), and air pressure has been reduced due to the rapid expansion of the bubble volume (Fig. 19). Last phase of the simulation is again an implosion similar to the first phase, but this time ends up with the rupture of the bubble as the result of the shape

Fig. 13. Air–water tube. Results for pressure, density and velocity are shown in figures (a)–(c) and are compared with the exact ones depicted in solid lines. Artificial viscosities are shown in figures (d) and (e).

Fig. 14. Time histories of (a) “X”, “Y” measures of bubble radius and pressure at the center of the bubble. (b) Length parameter, H.
instabilities excited at this stage, (Fig. 19). Note that the bubble almost recovers a circular shape at the end of the second stage. The parameters of this simulation correspond to the underwater implosion of glass spheres provided by Terner [12]. He detected the pressure pulse being created as the response of the implosion and traveling in water at 10.16 cm from the center of the glass and then studied the effect of the glass sphere on the
pressure peak. The absence of a structure that separates air and water can cause up to 40% of error on the pressure peak [12]. The same setting is used by Farhat et al. [13]. The water pressure computed in our simulation is compared with the experiment in Fig. 20. The typical signature of all implosions that consists of a low flat negative-pulse followed by a sharp positive-pressure spike of 0.1 ms duration can be seen in this figure. Experimental results [12] report a peak value in the range of 27–32 MPa for the pressure pulse and depending on the failure rate of the structure. The same range is obtained by the simulations done using DYSMAS [12]. Numerical results presented in [13], using the AERO-F solver, report a maximum value of 27 MPa. Our solution (labeled as PFEM in Fig. 20) suggests a maximum peak of 35 MPa that is in good agreement with the experimental results both in the value and the signature. A more detailed study of the underwater implosion of cylindrical containers, accounting for fluid-air-structure interactions, is provided in [46].

5. Conclusion

We have presented a Lagrangian analysis framework for the simulation of fast dynamic compressible multi-flows with special interest in underwater implosion modeling. A variationally consistent form of the hydrodynamic set of conservation equations is considered to model the compressible air and water media. A nodally matched definition of the interface is used to follow its evolution. This facilitates the duplication of the pressure DOF at the interface to easily deal with the large initial pressure jumps typical of implosion problems. The severe deformation of the interface, in addition to the large local motion of the fluids, requires an adaptive mesh generation that respects the interface connectivities at each time step. The formulation has been successfully tested for several benchmarks and then has been applied to model the underwater implosion of cm size cylindrical bubbles. Numerical results show that the hydrodynamics of the bubble has three different stages. The first one is the collapse of the bubble that results in the reduction up to 90% in the bubble radius and an increase in the internal pressure up to 2500 times the initial pressure. At this point a shock wave is created at the center of the bubble and RT instabilities appear at the air–water interface. The pressure wave traveling in water is detected at 10.16 cm from the center of the bubble. Results for this phase have been successfully compared with experimental and numerical data. The second stage then starts by the expansion of the bubble, due to the high internal pressure, and the vanishing of the RT instabilities till the bubble recovers 50% of its initial radius and gains an almost circular shape. The third stage is again a collapse of the bubble volume that ends up with the rupture of the bubble initiated from a fifth mode instability that appears at the air–water interface.

Acknowledgments

This work was partially supported by the Office of Naval Research (ONR) under contract N00014-09-1-0969. Support for this research was also Granted by projects REALTIME and SAFECON of the European Research Council (ERC) of the European Commission.

References