A numerical strategy for gas cavity-body interactions from acoustic to incompressible liquid phases.

G. Colicchio[1,2] **M. Greco**[1,2,3] **O.M. Faltinsen**[2,3]
giuseppina.colicchio@cnr.it **marilena.greco@cnr.it** **odd.faltinsen@ntnu.no**

1 **CNR-INSEAN, Italian Ship Model Basin, Roma – Italy.**
2 **CeSOS and AMOS, NTNU, Trondheim – Norway.**
3 **Dept. of Marine Technology, NTNU, Trondheim – Norway.**

The gas-cavity problem with initially high pressure, evolving in a surrounding liquid, and interacting with a near body, is a very interesting research topic because it involves several physical phenomena and is of practical interest in different contexts. For instance, underwater explosions represent an important issue for ships and offshore structures. Therefore it is necessary to predict structural effects and try to improve vessel design. To this purpose, physical tests were performed along the years and theories were developed (Cole 1948). Another important application is in medical field. Implosion of micro-bubbles with ultrasound in biological flows is used within a noninvasive technique to remove calculi in human bodies (Lingeman *et al*. 2009).

Here we first focus on the first application. When an underwater explosion occurs, a chemical reaction and a detonation process cause the formation of a hot gas with high pressure and the release of a shock wave traveling in the surrounding fluid. Then a superheated, spherical, bubble is formed which will first expand while the high pressure reduces in time and propagates in the surrounding liquid. Eventually the bubble starts to oscillate and affect the local pressure. In the first stage (shock wave) both the gas and the surrounding liquid behave as compressible, in the later stages (gas bubble) the acoustic wave will disappear and the water can be considered incompressible. The interaction of this two-phase fluid with a body will then depend on the vicinity of the body from the explosion zone and by the presence or not of other boundaries, *e.g.* the sea floor, the free surface.

The solution strategy: a time-space domain decomposition

We first assume that the explosion occurs very far from other boundaries and that hydrostatic pressure does not affect the explosion phenomenon leading to a radial symmetry of the bubble evolution. Initial values of bubble radius, density and pressure, can be obtained from physical tests. A compressible '1D' solver along the radial direction r is then used to simulate the flow evolution until the first shock wave from the explosion becomes close to the bottom of a vessel assumed infinitely extended within the local analysis. This can be done because the problem equations are hyperbolic and so the presence of the structure will not affect the fluid behind the shock wave. As the shock wave becomes close to the body, a time-space Domain-Decomposition (DD) strategy is switched on, where a compressible 3D solver is initiated by the simplified '1D' solution in an inner region affected by the body and used to investigate the fluid-body interactions. The '1D' solution is still applied far from the structure and provides the boundary conditions to the 3D solver along a control surface bounding the inner domain. This implies a one-way coupling. The DD limits the computational costs which are quite high if a compressible 3D solver is used for the whole simulation and everywhere due to the limits in the time step connected with the local speed of sound in the fluid. In case of an explosion very close to boundaries, this DD cannot be applied. The main features of the methods involved in the DD are briefly described next.

'1D' compressible solver for multi-phase flows Assuming radial symmetry, the problem (in general governed by the later equation (3)) can be studied as one-dimensional in the r direction with formal Euler equation

$$\frac{\partial \boldsymbol{U}}{\partial t} + \frac{\partial \boldsymbol{F}}{\partial r} = \boldsymbol{S}\,, \tag{1}$$

with $\boldsymbol{U} = [\rho, \rho u, E]^T$, $\boldsymbol{F} = [\rho u, \rho u^2, (E+p)u]^T$ and $\boldsymbol{S} = 2[\rho u/r, \rho u^2/r, u(E+p)/r]^T$. Here u is the radial velocity, p the pressure and E the total energy $\rho(e+u^2/2)$. For the closure of the problem we need an equation of state (EOS) for the specific internal energy e. Here this is assumed of the form $\rho e = f_f(\rho)p + g_f(\rho)$, with the functions f_f and g_f depending on the fluid properties. In particular, the JonesWilkinsLee EOS is used for the gas (Dobratz and Crawford 1985) and an isentropic Tait relation for the water (Cole 1948), *i.e.*

$$\begin{array}{ll} f_g = 1/\omega & g_g = [-A_g(1-\omega\rho_g/(R_1\rho_{0g}))e^{-R_1\rho_{0g}/\rho} - B_g(1-\omega\rho_g/(R_2\rho_{0g}))e^{-R_2\rho_{0g}/\rho_g}]/\omega \\ f_w = 1/\gamma_w & g_w = (B_w - A_w)\gamma_w/(\gamma_w - 1) \end{array} \tag{2}$$

Here the subscripts g and w stand for gas and water, respectively, ρ_{0g} is the initial gas density, γ_w is the ratio of specific heats for water and the other parameters are given later. The problem is solved in time with a first order scheme using the HLL approximate Riemann solver (Toro 1999) to estimate the fluxes $\boldsymbol{F}$ in each fluid and enforcing a two-shock approximation to the Riemann problem at the interface as proposed by Liu *et al.* (2003). The latter provides an exact solution when a shock wave is reflected and is reliable for gas-gas or gas-water flow. The related equation system is nonlinear and is solved iteratively with a Newton-Raphson method giving u_i, p_i, ρ_i^L and ρ_i^R, respectively, the radial velocity and pressure at the interface and the left and right density. To avoid possible instability of the solution, the left and right densities are corrected by enforcing an isobaric condition across the interface. This interface algorithm is inserted into a ghost fluid method (Liu *et al*. 2003) providing the conditions across the interface to each fluid. In particular, say that the interface is between node i and i+1 of the computational grid and that we need to solve for the fluid on the left. Here we consider that for nodes $\geq$ i the density, velocity and pressure are, respectively, ρ_i^L , u_i and p_i, and the other needed quantities are obtained subsequently. Similarly is done for the fluid on the right. At this stage the fluxes $\boldsymbol{F}$ can be calculated in each fluid and the problem can be stepped forward in time. The location of the interface is updated using the velocity u_i. The solver has been satisfactorily verified against several numerical solutions, for fully 1D problems (in this case $\boldsymbol{S} = 0$ in equation (1)) and problems with radial symmetry.

3D compressible multi-phase solver In 3D, the compressible inviscid flow is represented by the equation

$$\frac{\partial \boldsymbol{U}}{\partial t} + \frac{\partial \boldsymbol{F}_x}{\partial x} + \frac{\partial \boldsymbol{F}_y}{\partial y} + \frac{\partial \boldsymbol{F}_z}{\partial z} = 0, \tag{3}$$

with $\boldsymbol{U} = [\rho, \rho u, \rho v, \rho w, E]^T$, $\boldsymbol{F}_x = [\rho u, \rho u^2 + p, \rho uv, \rho uw, (E+p)u]^T$, $\boldsymbol{F}_y = [\rho v, \rho uv, \rho v^2 + p, \rho vw, (E+p)u]^T$, $\boldsymbol{F}_z = [\rho w, \rho uw, \rho wv, \rho w^2 + p, (E+p)u]^T$. Here u, v, w are the velocity components. As in '1D', these equations are completed by the equations of state described in the previous paragraph. The equations are integrated with a 3^{rd} order Runge-Kutta scheme in time and they are discretized with a 2^{nd} order finite differences scheme in space. A level set function ϕ is used to represent implicitly the interface between the two fluids and it is advected in time using the equation

$$\frac{\partial \phi}{\partial t} + \boldsymbol{V}_i \cdot \nabla \phi = 0 \tag{4}$$

where $\boldsymbol{V}_i$ is the interface velocity calculated as in Liu *et al.* (2003). To make the solution efficient in time, an adaptive mesh refinement is used according to MacNeice *et al.* (2000). The grid is halved either close to the interface between the two fluids or in proximity of high gradients of the fluid variables $\boldsymbol{U}$. An example of mesh refinement is shown in figure 1, at the starting time of the explosion, close to the interface, the grid size is extremely refined. The refinement grades to a coarse mesh far from the interesting region.

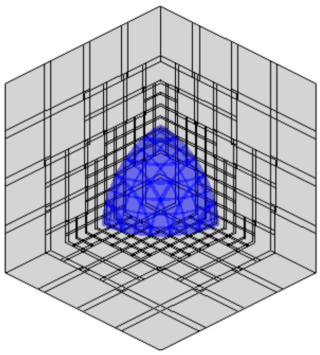

Figure 1: Example of adaptive mesh refinement close to the gas-water interface.

Fluid-structure interaction problem To estimate the local effects on the bottom of a vessel, as first attempt this has been modelled as an infinitely extended orthotropic plate, see *e.g.* Faltinsen (1999). The stresses and strains will be first evaluated using a quasi-static approach, then the structural and hydrodynamic problems will be coupled to assess excitation of hydroelasticity.

Preliminary results

The underwater explosion documented by Smith (1999) has been used as a test case to develop and assess the DD approach. The initial radius of the gas cavity is $r_0 = 0.16$ m and the parameters for the EOS of the fluids, using SI system, are: $\rho_{0g} = 1630.0$, $p_{0g} = 8.381 \cdot 10^9$, $A_g = 3.712 \cdot 10^{11}$, $B_g = 3.23 \cdot 10^9$, $R_1 = 4.15$, $R_2 = 0.95$, $\omega = 0.30$,

$\rho_{0w} = 1025.0$, $p_{0w} = 1.0 \cdot 10^6$, $A_w = 1.0 \cdot 10^6$, $B_w = 3.31E8$ and $\gamma_w = 7.5$. First the problem has been studied fully by the '1D' solver within the radial-symmetry assumption. A convergence analysis has been performed using a compu-

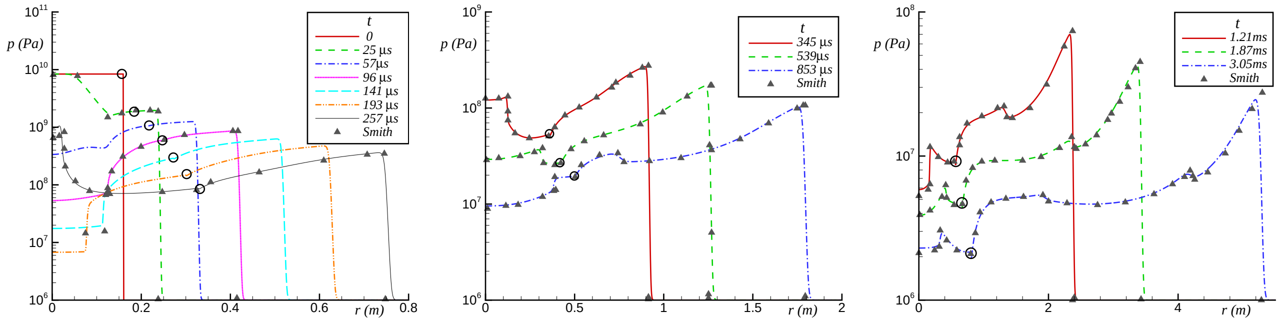

Figure 2: '1D' solution in the radial direction for the studied underwater explosion: pressure distributions at different time instants. The empty circles indicate the instantaneous radial location of the interface. The triangles are the numerical results by Smith (1999). For sake of clarity, in the left plot only the Smith's solutions at 25, 96 and 257μs, are shown.

tational domain long 10 m and a uniform discretization Δr. The order of accuracy OA (Colicchio 2004) was adopted as measure, which involves the time integral of the selected variable calculated with three discretizations and should be one for a solver accurate to the first order. In our case, using $\Delta r = 0.00125$ m, $1.5\Delta r$ ant $2.25\Delta r$ and studying the evolution up to 0.05 s, at a location $r \simeq 44 r_0$, OA was found 1.45, 1.44, 1.32 and 1.56, respectively for u, p, ρ and E. At the interface, where very complicated phenomena occur, OA is more limited and was found 0.39, 0.80 and 1.10, for the position of the interface, r_i, and for u_i and p_i, respectively. The evolution for the finest grid is shown in figure 2 in terms of pressure distribution and interface location at different time instants during the initial shock-wave phase. This stage involves a cavity expansion and is typically associated with a release of more than 50% of the energy from the explosion (Keil 1961). In this example, at first (left) a primary shock wave is caused by the detonation and moves rightwards while an expansion wave moves toward the bubble center and is later reflected from it leading to a low pressure at the core of the cavity. Later on the inner pressure rises and moves as a shock wave towards the interface. There, it is partially reflected and partially transmitted into the liquid phase (center). As a consequence of these repeated reflections, the intensity of the involved shock waves is reduced bringing toward an incompressible behavior (right). The described results fit well those by Smith (1999), based on an arbitrary Lagrangian-Eulerian version of the advective upstream-splitting shock-capturing scheme, also given in the figure. On a longer time scale the cavity reaches a maximum radius of about 2.2 m ($\simeq 13.8 r_0$) at about 0.066 s, this is consistent with the values reported by Smith (1999). Then, within the gas-bubble phase, the cavity starts to oscillate with smaller amplitudes as shown in the left of figure 3. Both pressure and velocity at the interface (center and right plots) are highest at the beginning. When the bubble is compressed the pressure tends to a peak and the velocity becomes negative, the magnitude of both of them decreases in time. According to studies by Keil (1961), most of the remaining energy from the explosion is released during the first bubble pulsation.

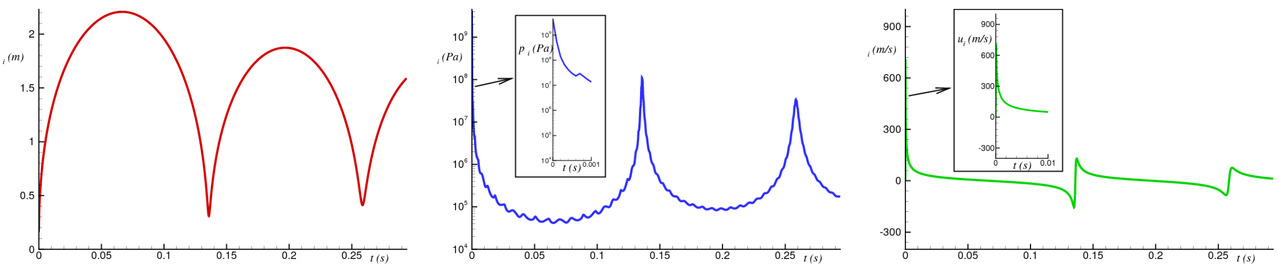

Figure 3: '1D' solution in the radial direction for the studied underwater explosion: location (left), pressure (center) and radial velocity (right) of the interface as a function of time. The results were obtained using a computational domain long 100 m, with constant $\Delta r = 0.00125$ m within 10 m and then stretching exponentially outwards.

The positive verification of the '1D' solver represents the first step of the solution strategy. The 3D method has also been built as compressible solver with one fluid and the exchange of information from the '1D' domain with the evolving cavity to the 3D domain with the compressible water is presently under assessment. This part of the research activity and the further developments, including the possibility to have the gas-water interface in the 3D solver sub-domain and the structural analysis, will be discussed at the workshop.

This research activity is supported by the Centre for Ships and Ocean Structures (CeSOS), NTNU, Trondheim, within the ”Violent Water-Vessel Interactions and Related Structural Loads” project and presently ongoing also within the Centre for Autonomous Marine Operations and Systems (AMOS), NTNU, Trondheim.

References

COLE, R. H. (1948). *Underwater Explosions*. Princeton, New Jersey: Princeton University Press.

COLICCHIO, G. (2004). *Violent disturbance and fragmentation of free surfaces*. Ph. D. thesis, University of Southampton, Southampton, UK.

DOBRATZ, B. M. AND P. C. CRAWFORD (1985). *LLNL Explosives HandbookProperties of Chemical Explosives and Explosive Simulants*. Lawrence Livermore National Laboratory.

FALTINSEN, O. (1999). Water entry of a wedge by hydroelastic orthotropic plate theory. *J. Ship Research 43*(3), 180–193.

KEIL, A. (1961). The response of ships to underwater explosions. Technical Report 1576, David Taylor Model Basin.

LINGEMAN, J., J. MCATEER, E. GNESSIN, AND A. EVAN (2009). Shock wave lithotripsy: advances in technology and technique. *Nat. Rev. Urol. 6*, 660–670.

LIU, T., B. KHOO, AND K. YEO (2003). Ghost fluid method for strong shock impacting on material interface. *J. Comput. Phys. 190*, 651–680.

MACNEICE, P., K. M. OLSON, C. MOBARRY, R. DEFAINCHTEIN, AND C. PACKER (2000). PARAMESH : A parallel adaptive mesh refinement community toolkit. *Computer Physics Communications 126*, 330–354.

SMITH, R. (1999). AUSM(ALE): A Geometrically Conservative Arbitrary LagrangianEulerian Flux Splitting Scheme. *J. of Computational Physics 150*, 268–286.

TORO, E. (1999). *Riemann Solvers and Numerical Methods for Fluid Dynamics*. Springer-Verlag.