

Development of Compressible-Incompressible Link to Efficiently Model Bubble Dynamics near Floating Body

Chao-Tsung Hsiao¹, Georges L. Chahine²

¹ DYNAFLOW, INC. 10621-J Iron Bridge Road, Jessup, MD 20794, USA, ctsung@dynaflow-inc.com

² DYNAFLOW, INC. 10621-J Iron Bridge Road, Jessup, MD 20794, USA, glchahine@dynaflow-inc.com

Keywords: Fluid Structure Interaction, boundary element method, bubble dynamics, free surface, piercing body

Abstract. The interaction between an explosion bubble and a floating body involves several physical phenomena that an accurate numerical simulation needs to capture. These include proper description of the initial shock wave and its propagation and interaction with the floating body and free surface and the subsequent growth, dynamics, collapse, and interaction of the resulting bubble with the structure. A numerical procedure which links a compressible finite difference flow solver with an incompressible boundary element method was applied to capture both shock and bubble phases efficiently and accurately. Both codes solved the fluid dynamics problem and coupled with a finite element structure code to simulate fluid-structure interaction. This numerical approach was validated by comparing the numerical results with available experimental measurements. The effects of including or neglecting some modeling aspects on the solution were considered.

Introduction

Underwater explosion interaction with a structure has long been an area of interest in the fluid and structures community [1-2]. The complex dynamics results from a shock wave created by the explosion, which leaves behind it a very dynamic underwater bubble. The shock wave and the bubble strongly interact with the structure. The dynamics of the bubble result in long duration loads with time scales, which could match the structure natural frequency. Modeling of the initial times in the dynamics requires a compressible flow solver to capture shock wave emission and propagation. However, such a flow solver usually uses a finite difference method (FDM) which requires very fine spatial resolution and small time step sizes. This makes them not very efficient for resolving the relatively long duration bubble dynamics. To overcome this, an incompressible potential flow solver based on a boundary element method (BEM) can be used to model the bubble dynamics once the shock phase has been described properly.

In this study we present the numerical procedures to transfer from the FDM compressible flow solver the flow field variables required to proceed with the BEM incompressible potential flow computations. In addition we describe coupling procedures between the BEM and a finite element method (FEM) based structure code, which are used to enable simulation of the full fluid-structure interaction between the bubble and the floating body. Unlike for a fully submerged body, a floating object pierces the free surface and only the wetted area of the body and the water free surface need to be discretized and solved for the BEM code. Since the body waterline changes dynamically, meshing, re-meshing and interpolation between BEM and FEM meshes needs to be done at each communication step for proper information exchange between the fluid and the structure codes. In this study we compare the numerical results with experimental measurements to validate the developed numerical procedure and the importance of various parameters which influence the accuracy of numerical simulations is addressed by isolating their effects.

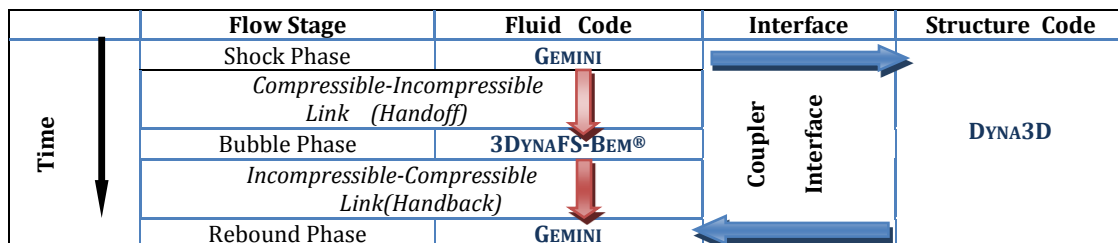


Figure 1. Schematic diagram of the numerical approach used to simulate fluid structure interaction problem between an underwater explosion and a structure.

Numerical Approach

Overview of the Approach. In the present study a hybrid scheme using three solvers was utilized to simulate the Fluid Structure Interaction (FSI) problem between an underwater explosion bubble and a floating body. This hybrid scheme combines the advantages of a compressible code, GEMINI [3], and an incompressible solver,

3DYNAPS-BEM[®] [4], to capture the shock and bubble phases. Both GEMINI and 3DYNAPS-BEM[®] are coupled with the structure dynamics code, DYNA3D [5], through an interface coupler. The schematic diagram shown in Figure 1 illustrates the hybrid numerical approach used in this paper. As illustrated in Figure 1, after the shock phase the flow field is “handed off” from GEMINI to 3DYNAPS-BEM[®] [6], which is then used to simulate most of the bubble period until the end of the bubble collapse where due to high speeds or to reentrant jet impact compressible flow effects prevail again. To continue the simulation into the rebound phase the solution is “handed back” [7] to GEMINI. In the current study, we will only focus on the shock and bubble phase during the bubble first period.

FDM-Based Compressible Flow Solver. GEMINI is a multi-component compressible Euler equation solver developed by the Naval Surface Warfare Center, Indian Head division based on a finite difference scheme [3]. The code solves continuity and momentum equations for a compressible inviscid liquid in Cartesian coordinates. These can be written in the following format:

$$\frac{\partial Q}{\partial t} + \frac{\partial E}{\partial x} + \frac{\partial F}{\partial y} + \frac{\partial G}{\partial z} = S, \quad (1)$$

$$Q = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ \rho w \\ \rho e_t \end{bmatrix}, \quad E = \begin{bmatrix} \rho u \\ \rho u^2 + p \\ \rho uv \\ \rho uw \\ (\rho e_t + p)u \end{bmatrix}, \quad F = \begin{bmatrix} \rho v \\ \rho vu \\ \rho v^2 + p \\ \rho vw \\ (\rho e_t + p)v \end{bmatrix}, \quad G = \begin{bmatrix} \rho w \\ \rho wu \\ \rho wv \\ \rho w^2 + p \\ (\rho e_t + p)w \end{bmatrix}, \quad S = \begin{bmatrix} 0 \\ 0 \\ 0 \\ \rho g \\ \rho gw \end{bmatrix}, \quad (2)$$

where ρ is the liquid density, p is the pressure, u , v , and w are the velocity components in the x , y , z directions respectively (z is vertical), g is the acceleration of gravity, and $e_t = e + 0.5(u^2 + v^2 + w^2)$ is the total energy with e being the internal energy. Eq (1) is applied to solve gas, liquid, or other media with an equation of state relating pressure, density, and energy for a given material or phase.

GEMINI uses a high order Godunov scheme. It employs the Riemann problem to construct a local flow solution between adjacent cells. The numerical method is based on single material higher order MUSCL scheme and tracks each material. To improve efficiency, an approximate Riemann problem solution replaces the full problem. The MUSCL scheme is augmented with a Lagrange re-map treatment of mixed cells. The code has been extensively validated against experiments [3].

BEM-Based Incompressible Flow Solver. 3DYNAPS-BEM[®] is a potential flow solver based on a boundary element method. The code solves the Laplace equation, $\nabla^2 \phi = 0$, for the velocity potential, ϕ , with the velocity vector defined as $\mathbf{u} = \nabla \phi$. A boundary integral method is used to solve the Laplace equation based on Green’s theorem:

$$\int_{\Omega} (\phi \nabla^2 G - G \nabla^2 \phi) d\Omega = \int_S \mathbf{n} \cdot [\phi \nabla G - G \nabla \phi] dS. \quad (3)$$

In this expression Ω is the domain of integration having elementary volume $d\Omega$. The boundary surface of Ω is S , which includes the surfaces of the bubble and the nearby boundaries with elementary surface element dS . \mathbf{n} is the local normal unit vector. $G = -1/|\mathbf{x} - \mathbf{y}|$ is Green’s function, where \mathbf{x} is a fixed point in Ω and \mathbf{y} is a point on the boundary surface S . Eq (3) reduces to Green’s formula with $a\pi$ being the solid angle under which \mathbf{x} sees the domain, Ω :

$$a\pi \phi(\mathbf{x}) = \int_S \left[\phi(\mathbf{y}) \frac{\partial G}{\partial n}(\mathbf{x}, \mathbf{y}) - G(\mathbf{x}, \mathbf{y}) \frac{\partial \phi}{\partial n}(\mathbf{y}) \right] dS, \quad (4)$$

where $a\pi$ is the solid angle. To solve eq (4) numerically, the boundary element method, which discretizes the surface of all objects in the computational domain into panel elements, is applied.

Eq (4) provides a relationship between ϕ and $\partial \phi / \partial n$ at the boundary surface S . Thus, if either of these two variables (e.g. ϕ) is known everywhere on the surface, the other variable (e.g. $\partial \phi / \partial n$) can be obtained. For the current problem, at each time step ϕ on the bubble and free surface will be obtained through the integration of Bernoulli equation:

$$\frac{d\phi}{dt} = \frac{1}{\rho} (p_{\infty} - p_l) + gz + \frac{1}{2} |\nabla \phi|^2, \quad (5)$$

where p_l is the liquid pressure at the bubble surface or free surface. For the floating body surface, $\partial\phi/\partial n$ is provided when the code is coupled with a structure code. The velocity potential, ϕ , and $\partial\phi/\partial n$ derived from eq (5) on the bubble and free surfaces are used to compute the velocity vector, \mathbf{u} . The locations of the bubble and free surfaces nodes are then updated with the coordinates of surface nodes, \mathbf{x} , advanced according to $d\mathbf{x}/dt = \mathbf{u}$.

FEM-Based Structure Code. DYNA3D is a non-linear explicit structure dynamics code based on finite element method developed by the Lawrence Livermore National Laboratory. DYNA3D is used here to handle the floating body motion and deformation with the loading provided by the fluid solver. DYNA3D uses a lumped mass formulation for efficiency. This produces a diagonal mass matrix \mathbf{M} , to express the momentum equation as:

$$\mathbf{M} \frac{d^2 \mathbf{x}}{dt^2} = \mathbf{F}_{\text{ext}} - \mathbf{F}_{\text{int}}, \quad (6)$$

where \mathbf{F}_{ext} represents the applied external forces, and \mathbf{F}_{int} the internal forces. The acceleration, $\mathbf{a} = d^2 \mathbf{x} / dt^2$, for each element is obtained through an explicit temporal central difference method. Additional details on the general formulation can be found in [5].

Compressible-Incompressible Link. A key aspect of the hybrid numerical procedure used here is the need of BEM flow field initial conditions at the time of “handoff”. These are based on the FDM compressible flow solution which provides all flow properties such as the velocity, pressure and density at the volume grids and need to be interpolated to the BEM grid as illustrated in Fig. 2 and Fig. 3. The steps of the procedure can be described as follows:

- The bubble and free surface liquid/gas interfaces are determined and the boundary element mesh generated based on the density distribution computed by the compressible solver (see Fig. 3).
- The velocities on the boundary element surface nodes are interpolated from the compressible flow field solution (volume grid) and the normal components of the velocities, $\partial\phi/\partial n$, are deduced according to the normal vectors at the discretized surface.
- The velocity potential values, ϕ , on the boundary element mesh including bubble, free surface, and body surfaces are computed after solution of eq (4) using the known $\partial\phi/\partial n$ at all surface nodes.
- The values of ϕ on the bubble and free surface nodes are used to initialize the BEM computation, which is then marched in time using eq (5).

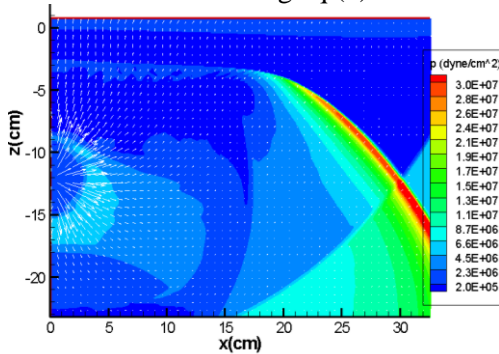


Fig. 2: Pressure contours and velocity vectors from the compressible flow solution at the time of handoff.

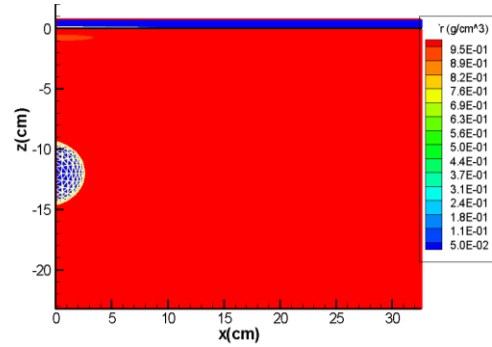


Fig. 3: Boundary element mesh generated on the gas/liquid interfaces according to the density distribution of compressible flow solution.

Fluid Structure Interaction Computation. In order to include fluid/structure interaction (FSI) effects in the simulations both fluid codes are coupled with DYNA3D through a coupler interface. The coupling is achieved through the following steps:

- Each of the fluid codes provides the structure code with the pressures at the structure surface nodes.
- The structure code then computes deformations and velocities in response to the loading.
- The coordinates and the velocities of the structure surface nodes are then sent back to the fluid code.
- The fluid code then solves the flow field and deduces the pressures at the surface using the body boundary nodes positions and normal velocities.

Additional details on the procedure concerning the exchange of information between the GEMINI volume grid and DYNA3D finite element mesh can be found in [3]. Concerning the coupling between 3DYNAFS-BEM[®] and DYNA3D, the information is exchanged using interpolations between values at the boundary element mesh and

values at the surface elements of the structure finite element model. Since the water contact line of the floating body changes dynamically, an automatic re-meshing scheme is implemented to modify the boundary element mesh on the floating body and adapt it at each time step to the position of the waterline. The following procedure is as follows:

- A “Slicer” is used to cross-cut the structure at selected locations (see Fig. 4a). The intersection of the cutting planes and the body surface elements provides a set of nodes in each cutting plane. These nodes are then ordered and connected with straight lines to form a contour of the structure section (see Fig. 4b).
- The position of the water-atmosphere contact points (blue nodes in Fig. 4c) are determined and all nodes above the water contact point except for the top corner point are removed. The nodes below the water contact point are coarsened, if necessary, and redistributed evenly (see Fig. 4c).
- The contours of all sections are connected to form a boundary element mesh (see Fig. 4d).

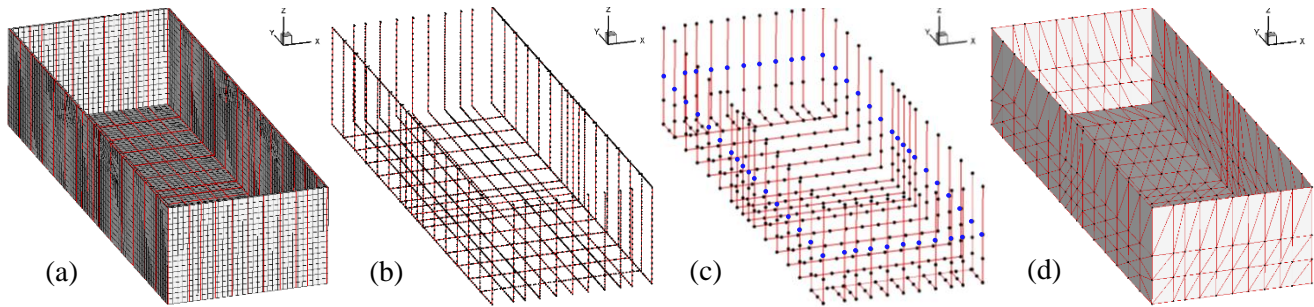


Fig. 4: Illustration of the automatic meshing scheme for generation of BEM grid on the floating body from an FEM body grid.

Validation

Problem Setup. In the present study we demonstrate the numerical scheme introduced above on the problem of the interaction of an explosion bubble and a Floating Shock Platform (FSP) which has a rectangular base of $4.9 \text{ m} \times 8.5 \text{ m}$ and an initial draft of 1.3 m . The problem setup is shown in Fig. 5. An explosion bubble is simulated using sudden release of gas bubble of initial radius 0.34 m and gas pressure 120 MPa . The bubble center is initially located at 7.3 m below the free surface and 6.1 m horizontally away from the floating body.

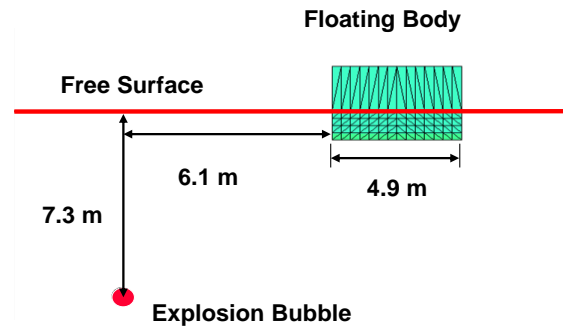


Fig. 5: Geometrical setup of an explosion bubble located at a depth of 7.3 m and at 6.1 m away from a floating body

Compressible-Incompressible Link. One important factors which affects solution accuracy of the link procedure is the time to handoff. Ideally, the time to execute handoff should be when all compressible effects have become negligible. Away from boundaries, compressible effects on the flow field due to an explosion bubble die out after the initial shock propagates away from the bubble center. However, the presence of a free surface and a nearby body result in the reflection of expansion and shock waves back towards the bubble. This expansion wave can also cause bulk cavitation near the free surface. To investigate the optimal handoff time we compare the solutions when the compressible-incompressible link is executed at four different times: 50, 60, 70 and 80 ms. The GEMINI pressure fields obtained by a grid of 6.1 million points with far field boundaries located at 400 m away from the bubble center at these four times are shown in Fig. 6. In the pressure contours, the white color regions indicate bulk cavitation (i.e. pressure below vapor pressure). It is seen that the water is still filled with large cavitation regions at 60 ms while only small cavitation regions near the bubble exist after 70 ms. This leads to selecting a handoff time selection larger than 70 ms and can be confirmed by comparing the time history of the floating body motion for these four handoff times. Fig. 7 compares the vertical velocity of the bottom center node on the FSP for the four handoff times with the velocity when the compressible code GEMINI is allowed to do the full computation and no link to incompressible is made (blue curve). It is seen that the results converge and become close to the no-handoff case for the handoff times 70 and 80 ms.

Evaluation of Different Modeling Aspects. With the numerical approach described above, we are able to isolate some modeling aspects such as inclusion of shock phase and structure deformation easily to study their effects on

the accuracy of the solution. To isolate the shock wave effects, we compare the above FSI results with the computations using 3DYNAFS-BEM[®] and DYNA3D only after by neglecting the initial compressible shock phase. Fig. 8 compares the vertical velocity and displacement of the floating body obtained from the two numerical simulations to test measurements [8]. It is seen that, as expected, the inclusion of the shock phase is essential to accurately predict the initial response of the floating body to the initial explosive bubble growth but is less critical for the later motion of the FSP.

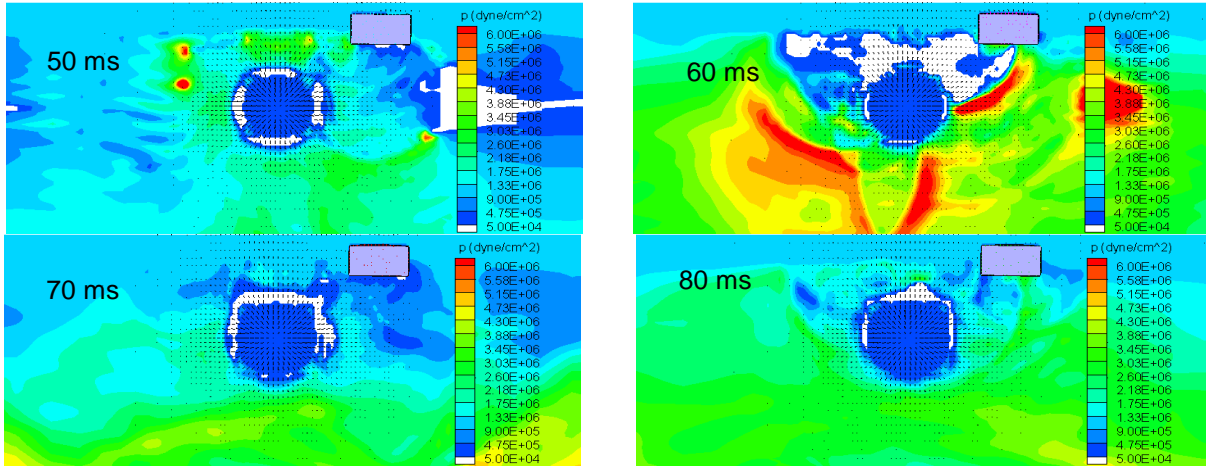


Fig. 6: Pressure contours obtained from a GEMINI simulation at four different times.

To study the effect of the structure deformation on the FSI simulation, we also conducted a simulation by treating the floating body as a rigid body. This is illustrated in Fig. 9 which compares the results of the two computations. One can see that ignoring the structure deformation shortens the bubble period and results in a smaller response (velocities) of the floating body during the bubble collapse than in the case of the deformable body.

The numerical predictions of the vertical velocity obtained from the different computations are compared to the experimental observations in Fig. 10. It is seen that the hybrid scheme with handoff at 70 ms predicts the floating body response the best. The CPU time requirement for this case is 1/10 of that using only the compressible code for the whole bubble period.

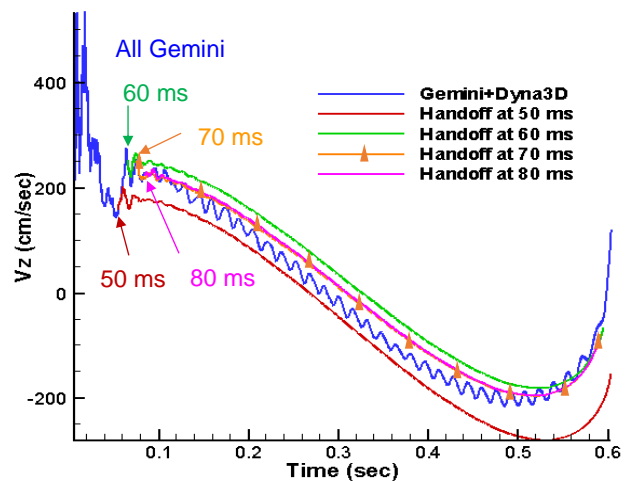


Fig. 7: Effect of handoff time on the vertical velocity monitored at the bottom center node of the floating body

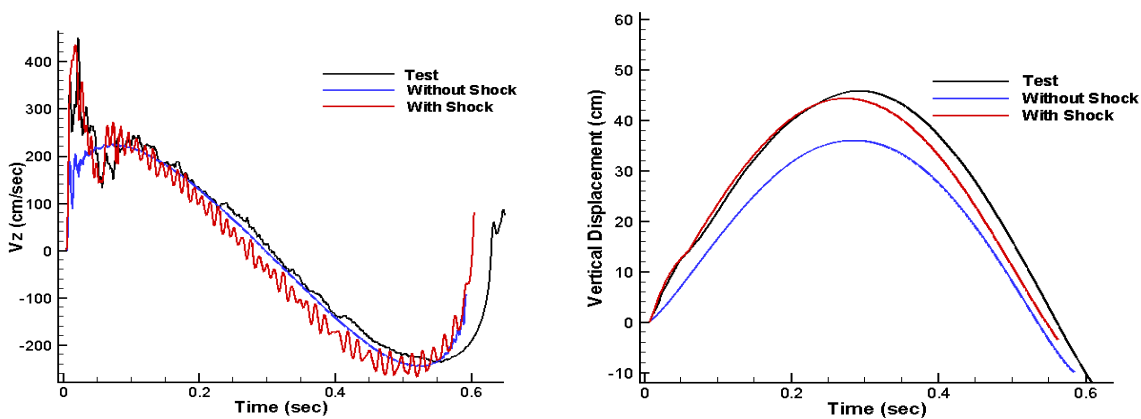


Fig. 8: Effect of the inclusion or neglect of the shock phase on the vertical velocity (left) and the floating body bottom center node displacement.

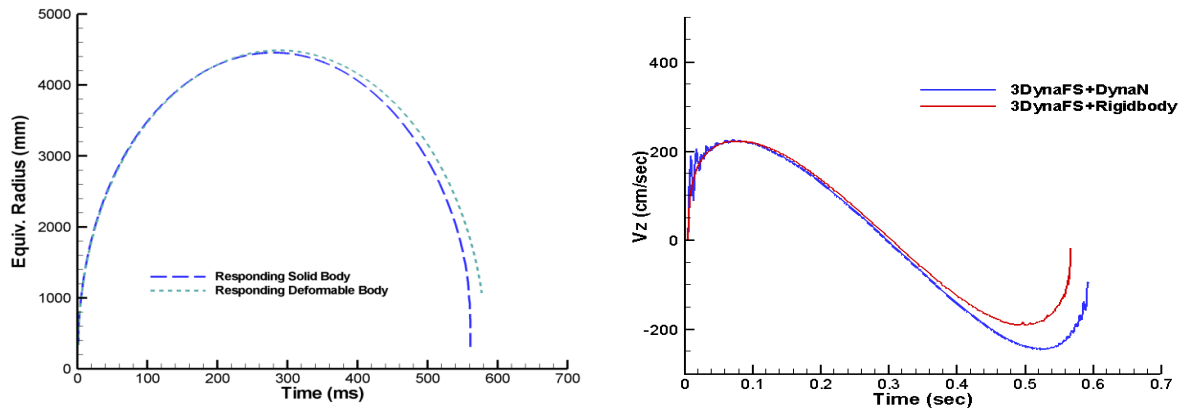


Fig. 9: Effect of the inclusion or neglect of the structure deformation on the bubble dynamics (left) and the floating body bottom center node vertical velocity.

Conclusions

A hybrid numerical procedure integrating three different solvers was developed to accurately and efficiently predict the response of a floating body to an underwater explosion bubble. It was found that shifting from the compressible to the incompressible simulation (link) should be performed after bulk cavitation has disappeared. Modeling the shock wave and the induced bulk cavitation is important for the simulation of the early history of the floating body response. Inclusion of the structure deformation was also shown to have a significant influence on the response of the floating body during the bubble expansion and collapse phases.

Acknowledgments

This study was conducted during the execution of an SBIR contract for the development of an alternative ship shock testing method. The support of The Naval Surface Warfare Center, Carderock Division (Dr. Fred Costanzo) and Indian Head Division (Mr. Greg Harris) is gratefully acknowledged.

References

1. G.L. Chahine, and K.M. Kalumuck, "BEM Software for Free Surface Flow Simulation Including Fluid Structure Interaction Effects," Int. Journal of Computer Application in Technology, Vol. 3/4/5, 1998
2. K.M. Kalumuck, G.L. Chahine, G.L. and C.-T. Hsiao, "Simulation of Surface Piercing Body Coupled Response to Underwater Bubble Dynamics Utilizing 3DYNAFS[®], a Three-Dimensional BEM Code," Computational Mechanics, 32, 319-326, 2003.
3. A.B. Wardlaw, J.A. Luton, J.R. Renzi, K.C. Kiddy, and R.M. McKeown, "The Gemini Euler Solver for the Coupled Simulation of Underwater Explosions," Indian Head Technical Report 2500, November 2003.
4. G.L. Chahine, R. Duraiswami, and K.M. Kalumuck, "Boundary Element Method for Calculating 2-D and 3-D Underwater Explosion Bubble Loading on Nearby Structures," Naval Surface Warfare Center, Weapons Research and Technology Department, Report NSWCDD/TR-93/46, September 1996.
5. DYNA3D, User manual "A nonlinear explicit, three dimensional finite element code for solid and structural mechanics", January 2005.
6. C.-T. Hsiao, J-K Choi, J-K and G.L. Chahine, "Analysis of the Phenomena influencing the Accurate Modelling of a Surface Vessel Response to an UNDEX," 78th Shock and Vibration Symposium Philadelphia, PA, November 4-8, 2007.
7. C.-T. Hsiao, G.L. Chahine, "Incompressible-Compressible Link to Accurately Predict Wall Pressure," 81th Shock and Vibration Symposium, Orlando, FL, October 24-28, 2010.
8. G.L. Chahine, K.M. Kalumuck, M. Tanguay, J.P. Galambos, M. Rayleigh, R.D. Miller and C. Mairs, "Development of a Non-Explosive Ship Shock Testing System-SBIR Phase I Final Report," Dynaflo, Inc. Report 2M3030-1 DTRC, March 15, 2004.

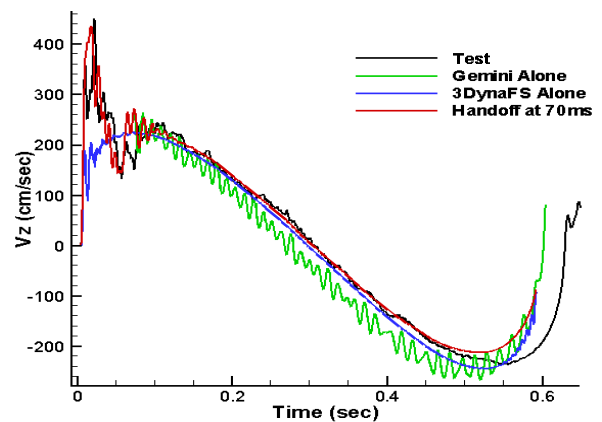


Fig. 10: Boundary element mesh generated on the gas/liquid interfaces determined according to the density distribution of compressible flow solution at the handoff time.