HAL
open science

# Study of the capabilities of an ALE bi-material fluid simulation for solving cavity expansion and collapse during an Hydrodynamic Ram event

Thomas Fourest, Jacques Dupas, Eric Deletombe, Jean-Marc Laurens, M. Arrigoni

## To cite this version:

Thomas Fourest, Jacques Dupas, Eric Deletombe, Jean-Marc Laurens, M. Arrigoni. Study of the capabilities of an ALE bi-material fluid simulation for solving cavity expansion and collapse during an Hydrodynamic Ram event. 11e colloque national en calcul des structures, CSMA, May 2013, Giens, France. hal-01717089

**HAL Id: hal-01717089**

**https://hal.science/hal-01717089v1**

Submitted on 25 Feb 2018

**HAL** is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire **HAL**, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.

Public Domain

**CSMA 2013**
11e Colloque National en Calcul des Structures
13-17 Mai 2013

# Study of the capabilities of an ALE bi-material fluid simulation for solving cavity expansion and collapse during an Hydrodynamic Ram event

Thomas Fourest[1] *, Jacques Dupas[1], Eric Deletombe[1], Jean-Marc Laurens[2], Michel Arrigoni[2]

[1] Département Aéroélasticité et Dynamique des Structure/Conception et Résistance Dynamique, ONERA, France, {thomas.fourest, jacques.dupas, eric.deletombe}@onera.fr
[2] ENSTA Bretagne/Laboratoire Brestois de Mécanique et des Système, France {jean-marc.laurens, michel.arrigoni}@ensta-bretagne.fr
* Corresponding Author

**abstract** — The hydrodynamic ram phenomenon generally referred to as HRAM, constitutes an important threat to modern aircraft, especially lightweight aircraft such as UAV that cannot be armoured due to the weight penalty. The aim of the present feasibility study is to determine whether an ALE solver using bi-material fluid permits a qualitative simulation of all the phases of the HRAM event, in particular the oscillations (growth and collapse) of the cavity created during the impact of a non-trivial geometry high speed projectile on a water filled tank. The present study examines the closure of the wake cavity, its shape and the deceleration of the projectile.
**keywords** — Hydrodynamic Ram, Fluid/Structure interaction, cavitation, ALE bi-material.

## 1 Aircraft vulnerability due to a ballistic impact on a fluid filled tank

In the field of aircraft safety, the vulnerability considerations are of utmost importance. The study of the effects identified as important factors in aircraft vulnerability implies solving the fluid/structure interaction problem which has been for the past decades, one major focus of research. In particular during ballistic impacts on fuel filled tanks, the phenomenon referred as Hydrodynamic Ram (HRAM) can occur.

During the impact of a projectile on a liquid filled tank, the kinetic energy of the projectile is transferred to the fluid in the form of thermal energy (tangential friction) and of kinetic energy (drag). This transfer gives rise to several phenomena. The HRAM is generally characterized by five phases (see Figure 1) : the shock phase ($t = 0\ ms$), the drag phase ($t = 0.2\ ms$), the surface closure (its time of occurrence is not easily determined in this experiment), the cavity oscillation phase ($t = 2.5\ to\ 6.5\ ms$), and the potential exit phase (that does not appear in this experiment). Those phases could induce important loads on the tank structure with potentially destructive results.

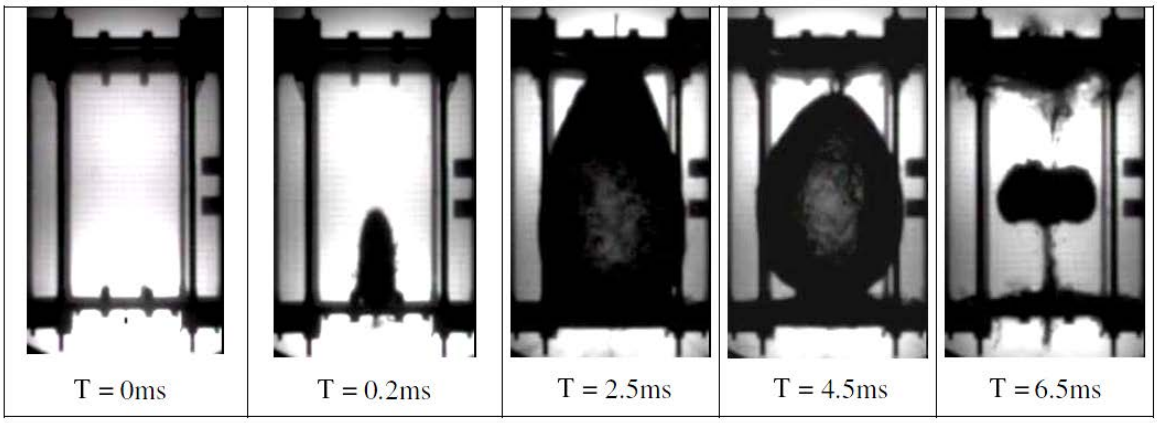



Fig. 1 – ONERA ballistic test of 7.62 mm ammunition at 790 m/s on a generic EADS-IWF tank [4].

The physics of the problem are quite complex when dealing with "non-trivial geometry" high speed deformable projectiles. It depends on many highly non-linear aspects (fluid, material, geometry, etc) that no analytical model or expert predictions can anticipate. Thus, the use of reliable numerical simulations appears necessary to correctly take this threat into account at the design stage.

## 2 State of the art of HRAM modelling

In the studies carried out at ONERA, the numerical simulation of the cavity oscillations (growth and collapse) is of particular interest. The aim is to demonstrate that the ability to qualitatively and quantitatively simulate the oscillations of the cavity during HRAM events is necessary to determine the structural loads in general cases of ballistic impact on liquid filled structures. This part summarises the state of the art of the HRAM modelling, presented in the open literature.

The first numerical studies of the complete phenomenon (projectile, liquid and gas phase and structure) were conducted using Lagrangian explicit finite elements in the seventies [7]. These studies focused on the simulation of the propagation of the shock wave and the drag phase. This was achieved by deleting fluid elements that exceed a maximum deformation criterion. It clearly shows that this method did not permit to accurately represent the cavity growth and collapse due to the erosion of fluid elements and to the simple equation of states used for the fluid.

Improvements of this method were proposed by using classical Lagrangian explicit finite elements in [5] and [3]. In these studies, the fluid elements were not discarded but were supposed to flow smoothly around the projectile. Unfortunately, the simulated time was very short due to the fluid mesh deformation which greatly limited simulation versus experimental comparisons.

More recently, new numerical methods were used to try to simulate the hydrodynamic ram phenomenon. The main studies in this area have been carried out by Varas et al. [12], [13], [11] and [1]. They presented results in which they demonstrated the capabilities of a commercial explicit finite element software (LS-Dyna) to simulate the shock phase, drag phase and the beginning of the cavity expansion phase generated by the impact of an indeformable sphere without tumbling, in an aluminium water filled tank. These different phases involve different time steps are actually minimized by the smallest one. To deal with the very large amount of deformation in the fluid during HRAM events, they used both an Coupled Euler Lagrange (CEL) solver with "bi-material" fluid and a mesh free Smooth Particles Hydrodynamics (SPH) approach. For the CEL simulation, they used an Eulerian mesh composed of two zones, the first one full of water and the second empty (air is not modelled in the simulation) ; multi-material elements were used to allow the water to fill the void elements. Both formulations appear to be able to qualitatively simulate the shock phase, the drag phase and the beginning of the cavity growth phase. Nevertheless they concluded that the CEL solver is more efficient than the SPH method to simulate the phenomenon in terms of computation time.

In the open literature no numerical simulations of HRAM events are performed up to the time of cavity closure, and the following evolution of the cavity bubble has still not be approached.

## 3 Importance of the cavitation phase simulation

The cavitation phase is the more complex phase and the least understood of the phases that occur during the HRAM phenomenon. Generally, during the water entry of a projectile, the wake created by the drag of the projectile separate from the surface, either by a surface seal or by a deep seal (closing that occurs under the surface of the fluid) as could be seen in [9]. In both cases a cavity bubble will be created. This cavity may continue to expand if the momentum associated with the velocity field is sufficiently high to separate the flow from the projectile and to continue to expand the cavity. Anyway the cavity will eventually collapse.

In his experiments, Disimile [6] found that greater pressures would occur either during the drag phase or during the cavitation phase depending on the projectile's material (mass). Deletombe [4] performed the same type of pressure measurements for 7.62 mm bullet impacts on water filled tanks ; he observed greater pressure during the drag phase than during the cavitation phase but during a shorter time period. He concluded that none of those phases could be neglected in the dimensioning of structure because

they could both carry a significant amount of energy. The energy partition between the drag phase and cavitation growth also depends on the shape of the projectile, on the time of its eventual tumbling and on its material (mass) [6].

# 4 Numerical simulation of the water entry of the projectile

The aim of this study is to determine the capability of a CEL solver with an Eulerian (air + water) fluid multi-material law to simulate the growth and collapse of cavities created by ballistic impacts on fluid filled tanks. Hence the correct simulation of the cavity closure is of key importance. During the closure of the cavity the phenomenon changes from the drag of a projectile in liquid to the evolution of a bubble of gas in the same liquid. The time of closure determines the amount of gas trapped in the bubble. The amount of gas is an important quantity, that influences the following behaviour of the bubble and the intensity of the pressure pulses emitted during the collapse of the gas bubble.

The complete simulation of the penetration of a high speed projectile in water filled tanks is a very complex task. In the present study great simplifications were made in order to concentrate on the cavity formation in the wake of the projectile and its closure. Firstly the water entry of a projectile is considered here instead of impacts on water filled tanks. The confinement effect of the tank is also studied, without taking its penetration by the projectile into account (a pre-pierced tank is used). In addition, due to great numerical instabilities in a bi-phasic simulation of high speed projectiles impacts, observed by Thevenet in [10], and because the observation of cavity closure in high speed projectiles impacts on fluid filled tanks is not clear, impacts at lower speed are examined in this study. It is assumed that the process of closure of the cavity is the same regardless of the speed of the projectile and that only its content evolves with speed due to a drop in cavity pressure that leads to the vaporisation of the water during cavity growth. The classic water entry experiment at low speed compared to ballistic speeds (10.6 m/s) presented by May in 1952 [8], has been identified to fullfil the above conditions. In experiments presented by Shi [9], the same type of cavity closing is observed for non-spherical projectiles at 350m/s. And even though it is not clear, it seems that in the ballistic speed (785 m/s) water entry of an ammunition experiment carried out at ONERA and presented in [2], the cavity closure respects the same scenario.

# 5 Finite Element model

The simulation of the HRAM phenomenon has been performed using the EUROPLEXUS explicit research code, which has been developed by the French CEA and the EU CRC-ISPRA to accurately solve fast dynamic problems for coupled fluid/structure systems. It enables the use of multi-material fluids in CEL solvers.

## 5.1 Description

Due to the symmetry of the problem and in order to reduce calculation time, an axisymmetric calculation is used to simulate these experiments. A CEL solver is used. In Europlexus, the Eulerian part is defined as ALE without any grid motion. Thus the mesh consists of two Eulerian fluid zones, water and air and of a Lagrangian projectile and tank. The projectile is a sphere of 1 inch (12.5 cm) diameter. Each fluid material mesh is composed of two parts, a fine part for the interaction (1mm side square elements) with the projectile, and a second part where the mesh becomes coarser at a further distance from the projectile. In May's simulation experiment, the fluid domain is assumed to be of infinite dimensions. Absorbing boundaries conditions have been used to prevent wave reflection that would disturb the simulation. As the upward jet in May's experiment proved to be dependent on gravity effects, the gravity acceleration is taken into account in our simulation.

## 5.2 Material laws

### Fluid model

The Europlexus code permits the modelisation of the behaviour of a liquid-gas mixture. This mixture is assumed to be homogeneous and includes two phases. Modelling the air material permits to take the depression in the wake of the projectile and the variation of the bubble pressure into account.

The behaviour of the gas is assumed to be perfect and the state variables are linked by the following equation :

$$P_g = P_{g0}\left(\frac{\rho_g}{\rho_{g0}}\right)^{n_g} \tag{1}$$

Here $P_{g0}$ and $\rho_{g0}$ are the initial values of gas pressure and density, and $n_g$ is the polytropic exponent. This exponent may be an arbitrary real number. The value $n_g = 1$ corresponds to an isothermal transformation while $n_g = \gamma_g$ corresponds to an adiabatic transformation.

The liquid is assumed to be compressible and isothermal. The equation of state is the following one :

$$P_l = P_{l_0} + (\rho_l - \rho_{l_0})c_l^2 \tag{2}$$

Here $P_{l_0}$ and $\rho_{l_0}$ are the initial values, and $c_l$ the sound speed is assumed constant. It is also possible to take the viscosity of the liquid $\mu_l$ into account with this bi-phasic law.

This law is used in ALE or Eulerian formulation ; the mixture in the elements is modelled with a volume of fluid approach. The interface between the fluids is not defined, and the content of the elements is calculated using the mass fraction in gas $x_l$ and liquid $x_g$.

$$x_l + x_g = 1 \tag{3}$$

### Projectile and tank model

The projectile and tank are modelled using a simple linear elastic law as the projectile does not undergo notable damages at this speed of impact (10.6 m/s). The Young modulus of steel is noted $E$ its density $\rho$, and its Poisson coefficient $\nu$.

### Numerical values used

The table 1 presents the numerical values used in the following simulations.

| | | | | |
|---|---|---|---|---|
| Steel | $\rho = 7.72.10^{-3}\ kg/mm^{-3}$ | $E = 2.01.10^5\ Mpa$ | $\nu = 0.26$ | |
| Water | $\rho_{l0} = 1.10^{-3}\ kg/mm^{-3}$ | $c_l = 1500\ m/s$ | $P_{l0} = 0.1\ MPa$ | $\mu_l = 1.006.10^{-3}\ Pa.s$ |
| Air | $\rho_{g0} = 1.10^{-6}\ kg/mm^{-3}$ | $n_g = \gamma_g = 1.4$ | $P_{g0} = 0.1\ MPa$ | |

Tab. 1 – Numerical values used for the simulation

## 5.3 Fluid structure interaction

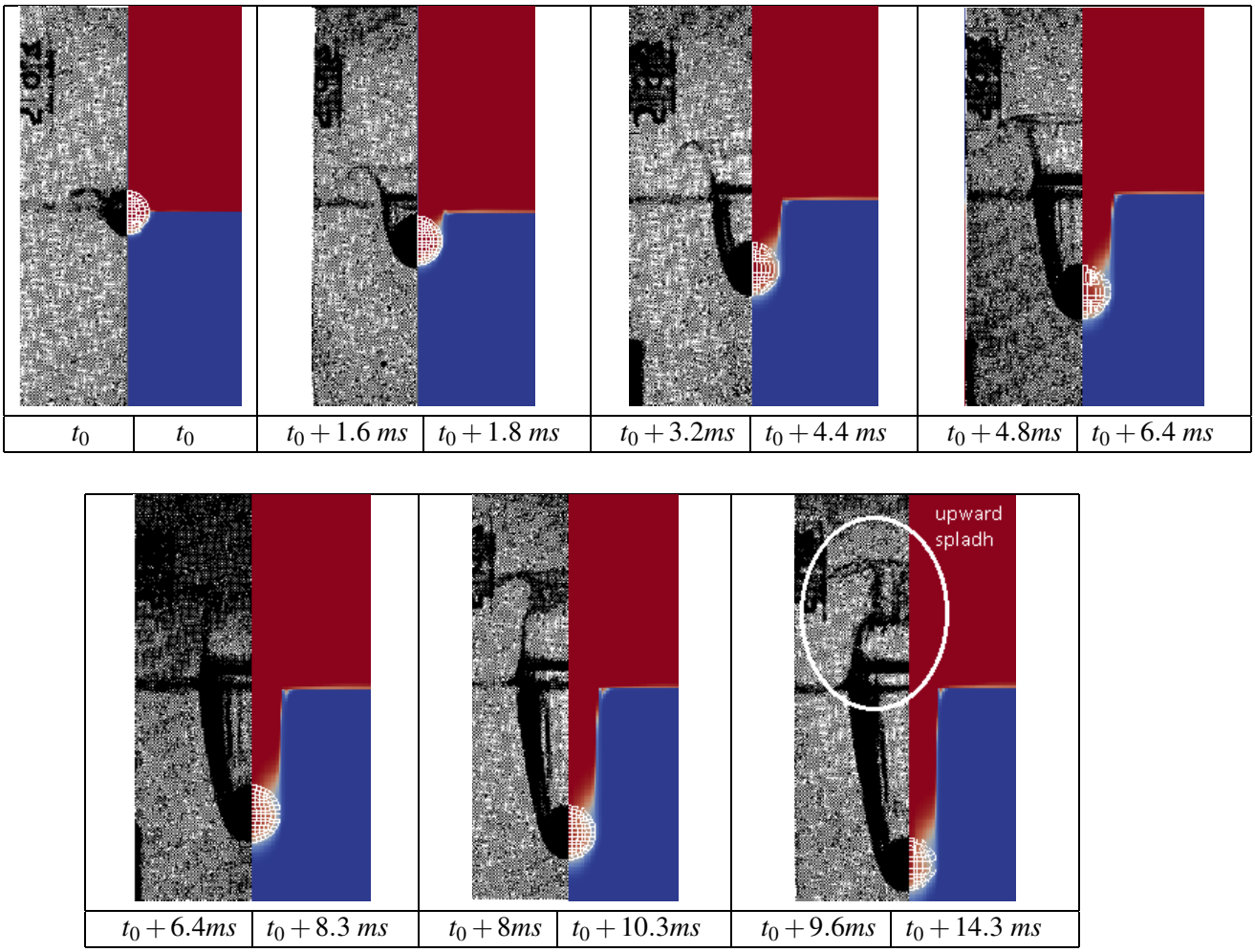
In the Europlexus code it is possible to model fluid structure interactions using a formulation based on Lagrange multipliers. In this case, suitable fluid structure interaction constraints are written between the fluid and the structure velocities, and are imposed exactly (and costly) (in an implicit manner) by the Lagrange multipliers method.

# 6 Results

## 6.1 May's experiment (water entry 10.6 m/s)

Figures 2 and 4 show the comparison between the experimental results and the simulation. The comparison is carried out at the same depth of penetration of the sphere in the liquid.

At first similarity could be observed in the drag created cavity shape between experiment and simulation, however the projectile is more decelerated in the simulation than in the experiment. It has been




| $t_0$ | $t_0$ | $t_0+1.6\ ms$ | $t_0+1.8\ ms$ | $t_0+3.2ms$ | $t_0+4.4\ ms$ | $t_0+4.8ms$ | $t_0+6.4\ ms$ |
|---|---|---|---|---|---|---|---|

| $t_0+6.4ms$ | $t_0+8.3\ ms$ | $t_0+8ms$ | $t_0+10.3ms$ | $t_0+9.6ms$ | $t_0+14.3\ ms$ |
|---|---|---|---|---|---|

Fig. 2 – Comparison of the cavity shape between the experimental shape and mass fraction of gas calculated in the simulation (Blue : $x_g = 0$ , Red : $x_g = 1$)

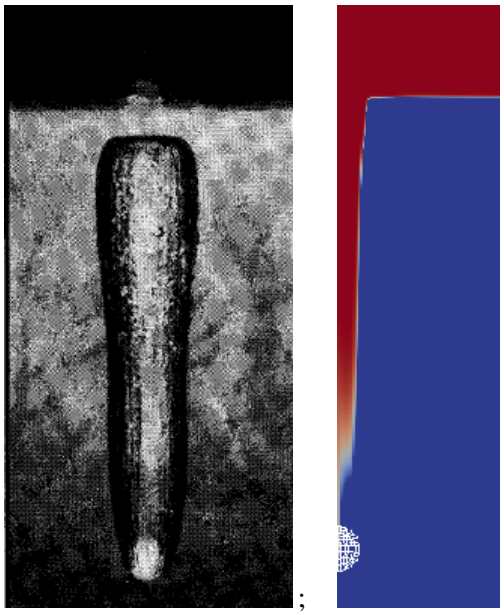


Fig. 3 – Photograph taken shortly after the cavity pulled away from the surface and the simulation result at the same depth of penetration of the sphere (Blue : $x_g = 0$ , Red : $x_g = 1$)

observed in this study that it is probably due to the fluid mesh size and that the drag of the projectile in the liquid could be better represented by decreasing this size. However a thinner mesh density would lead to very long calculation times particularly in three dimensional calculations, therefore the convergence in mesh size has not been examined in the present study.

In May's experiments the surface closure of the cavity created by the projectile comes from the upward splash that closes above the water level. In this simulation no upward splash appears. In the simulation no closure of the drag created cavity is observed and would lead to difficulties to perform a simulation of the ballistic speed water entry of the projectile. It is obvious that if the cavity does not close, no bubble would be properly created and the cavity growth and collapse phases would not occur in the simulation, and as explained previously those phases should be taken into account in the dimensioning of the structure.

### 6.2 Water entry with confinement tank

It is difficult to observe the process of the closure of the wake cavity during a ballistic impact as little information on this subject is available. The simulation aims to observe the influence of the presence of the tank walls on the cavity closure scenario. The speed of penetration is still 10.6 m/s. Figure 4 shows the cavity evolution observed in the simulation.

The confinement due to the tank influences the cavity closure scenario. The pressure wave reflected from the tank walls tends to compress the cavity in the wake of the projectile ; this leads to a surface seal of the cavity that would occur around 20 ms after the impact where the opening is 4 mm large. In this simulation, complete closure is not reached, it has been verified with another simulation (not shown here) that this is due to the limit condition on the symmetry axis, and that this problem does not occur when using three dimension simulations.

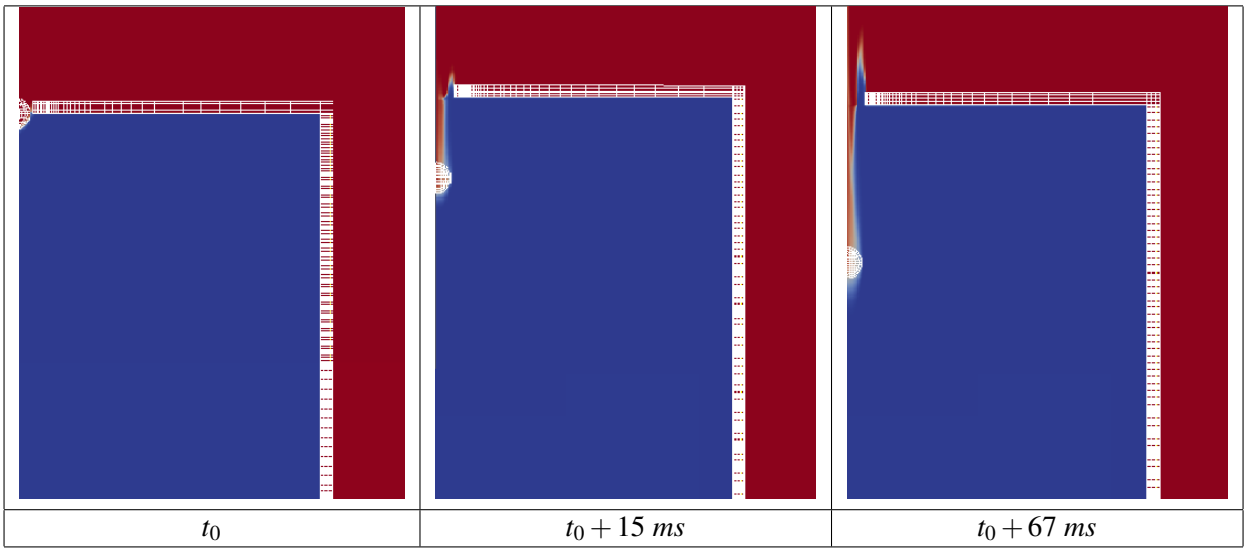



Fig. 4 – Cavity evolution in a fluid filled tank penetration simulation (10.6 m/s), mass fraction of gas calculated in the simulation (Blue : $x_g = 0$ , Red : $x_g = 1$)

## 7 Conclusion

The capabilities of an CEL bi-material solver to simulate high speed water entry have been examined in this study through the simulation of a classic experiment at slow speed (10.6 m/s) compared to the ballistic speed in May's experiment. The simulation of this experiment has shown that the actual Coupled Euler Lagrange solver with bi-material fluid law, does not permit to numerically simulate the closing of the cavity created in the wake of a projectile in the case of a free surface and infinite medium, but

it does permit the simulation of the closing of the wake cavity in the case of tank penetration due to its confinement. It would be interesting to determine the lack of any confinement effect in May's experiment, but the dimensions of the water container used for the experiment are not presented in his paper. It is also believed that even at a low speed impact (10 m/s) the closure of the cavity might be governed by surface tension effects and not by gravity acceleration. A further study should be carried out to take this phenomenon into account for water entry.

This study demonstrates that the closure of wake cavity can be achieved for a liquid filled tank penetration simulation using an CEL bi-material solver. This is an important problem since correctly simulating the closing of the surface is of key importance to the study of the cavity oscillations phase created by ballistic impacts in water. However, difficulties have been observed in the simulation of high speed projectile penetration in water. The simulation stopped due to a rapid drop of the elemental time step to ensure the convergence of the pressure calculation in the wake of the projectile. It is believed that the computed drop in pressure in the wake cavity at this speed of impact is not physical and that it would be limited by the vaporisation of the fluid medium that is not taken into account in this simulation. New material laws which take the liquid vaporisation into account, should be developed to avoid these difficulties. Only then will it be possible to compare the time of cavity closing and the evolution of the cavitation bubble in the tank between simulation and experiment.

# 8 Acknowledgement

This study has been carried out as part of the PhD thesis of the main author. The authors would like to thank the French Ministery of Defense and DGA (French Armament Procurement Directorate), for their financial support in the thesis preparation.

## Références

[1] J.A. Artero-Guerrero, J. Pernas-Sánchez, D. Varas, and J. López-Puente. Numerical analysis of cfrp fluid-filled tubes subjected to high-velocity impact. *Composite Structures*, 2012.

[2] A. Charles, E. Deletombe, and J. Dupas. A numerical study on cavity expansion in water : Hydraulic ram under ballistic impacts. In *International Conference on Structures Under Shock and Impact*, Kos, Greece, September 2012.

[3] E. Deletombe, D. Delsart, J. Dupas, J.L. Charles, and J.F. Sobry. Methodology for f.e. resolution of coupled fluid/structure problems. In *International Crashworthiness and Design Symposium 2003*.

[4] E. Deletombe, J. Fabis, and J. Dupas. Vulnerability of a/c fuel tanks with respect to hydrodynamic ram pressure. interpretation of 7.62 mm experiments. In *Colloque National en Calcul des Structures (CSMA)*.

[5] E. Deletombe and B. Malherbe. Fluid-structure simulation of hydraulic ram pressure in fuel tanks. In Civil-Comp Press, editor, *Advances in computational Structural Mechanics*, pages 271–281, Edinburgh, Scotland, 1998.

[6] P.J. Disimile, L.A. Swanson, and N. Toy. The hydrodynamic ram pressure generated by spherical projectiles. *International Journal of Impact Engineering*, 2009.

[7] K.D. Kimsey. Numerical simulation of hydrodynamic ram. Technical Report ARBRL-TR-02217, US Army Ballistic Research Laboratory, 1980.

[8] A. May. Vertical entry of missiles into water. *Journal of applied physics*, 23(12), 1952.

[9] H.H. Shi and M. Itoh. High-speed photography of supercavitation and multiphase flows in water entry. In *Proceedings of the 7th International Symposium on Cavitation*, Ann Arbor, Michigan, USA, August 2009.

[10] P. Thevenet. Impact balistique d'une balle otan 7.62 sur réservoir : expérimentation et simulation du coup de bélier. In *19 ème Congrès Français de Mécanique*, 2009.

[11] D. Varas, J. López-Puente, and R. Zaera. Numerical analysis of the hydrodynamic ram phenomenon in aircraft fuel tanks. *AIAA JOURNAL*, 50(7), 2012.

[12] D. Varas, R. Zaera, and J. López-Puente. Numerical modelling of the hydrodynamic rem phenomenon. *International Journal of Impact Engineering*, 36 :363–374, 2009.

[13] D. Varas, R. Zaera, and J. López-Puente. Numerical modelling of partially filled aircraft fuel tanks submitted to hydrodynamic ram. *Aerospace Science and Technology*, 16 :19–28, 2012.