International Journal of Multiphysics
Volume 18, No. 2s, 2024
ISSN: 1750-9548

Numerical Investigation of a Mixed SPH-FEM Formulation for Fluid Structure Interaction Problems

[1]Lobosco, R; [2]Souli, M; [3]AlBahkali, E; [4]Moatamedi, M; [5]Erchiqui,F

[1]Rio de Janeiro Federal University, Brazil
[2]Université de Lille UML, France
[3]King Saud University, Saudi Arabia
[4]New York University in London, UK
[5]University of Quebec, Canada

Abstract

Simulation of Fluid Structure Interaction FSI, has become more and more the focus of computational engineering, where FEM (Finite element Methods) for structural mechanics and Finite Volume for CFD are dominant. For small deformation, FEM Lagrangian formulation can solve structure interface and material boundary accurately, the main limitation of the formulation is high mesh distortion for large deformation and moving structure. One of the commonly used approach to solve these problems is the ALE (Arbitrary Lagrangian Eulerian) formulation which has been used with success in the simulation of fluid structure interaction with large structure motion such as sloshing fuel tank in automotive industry and bird impact in aeronautic industry. For some applications, including bird impact and high velocity impact problems, engineers have switched from ALE to SPH method to reduce CPU time and save memory allocation.

In this paper a mixed SPH-FEM method is presented. The mathematical and numerical implementation of the FEM method using an ALE (arbitrary Lagrangian Eulerian) formulation is described. From different simulation, it has been observed that for the SPH-FEM method to provide similar results as ALE or Lagrangian formulations, the SPH meshing, or SPH spacing particles needs to be finer than the ALE mesh. To validate the statement, we perform a simulation of a hydrodynamic impact problem. For this application , the particle spacing of SPH method needs to be at least two times finer than ALE mesh. A contact algorithm is performed at the fluid structure interface for both SPH-FEM and SPH-Solid. Since VOF (Volume of Fluid) method is part of the general ALE method, in this paper we use either ALE or VOF terminology to describe the VOF method. Since contact algorithm is an important part of the fluid structure interaction, in this paper we describe the penalty contact us

Keywords: ALE, SPH, Fluid Structure Interaction.

1 Introduction

Theoretical and experimental analysis of hydrodynamic impact problems as well as underwater explosion have been considered by several researchers over the past decades, using empirical methods as CONWEAP (Conventional Weapon) code when the explosive charge is far away from the structure, and Lagrangian description of motion for near field. In its formulation, CONWEAP code does not represent the physical behavior of detonation. When a high explosive is detonated an inward wave is generated in the explosive material, at the same time, a shock wave moves through the air medium, which is at lower pressure and a contact discontinuity appears between the rarefaction wave and the shock wave. Experiments have shown, see Kingery and Bulmash (1984), that the resulting flow is quite complex, involving several physical phenomena as

burning effects and heat transfer. The detonation of high explosive material converts the explosive charge into gas at high pressure and temperature what leads to damage structures. Numerical simulation of high explosive detonation and expansion are very difficult for classical numerical methods, see Boyer (1960). During the process in the explosion, a very thin reaction zone divides the domain into inhomogeneous parts and produces large deformations. Numerical simulation using appropriate equation of state for high explosive detonation, helps to describe these phenomena, and also minimize the number of tests required that are very costly. Once simulations are validated by test results, it can be used as design tool for the improvement of the system structure involved. Initially FEM Lagrangian were used to simulate these problems, unfortunately classical Lagrangian methods cannot resolve large mesh distortion, runs are stopped before reaching termination time, due to negative Jacobian in highly distorted element. ALE multi-material description of motion developed in Aquelet, Souli and Olovson (2005) can be used as an alternative for the simulation of high explosive phenomena. The ALE formulations have been developed to overcome the difficulties due to large mesh distortion. For some applications, including underwater explosions and their impact on the surrounding structure, engineers have switched from ALE to SPH method to reduce CPU time and save memory allocation.

It is well known from previous papers, see Ozdemir, Souli and Fahjan (2010) that the classical FEM Lagrangian method is not suitable for most of the FSI (Fluid Structure Interaction) problems due to high mesh distortion in the fluid domain. In many applications the ALE formulation has been the only alternative to solve fluid structure interaction for engineering problems. For the last decade, SPH method has been used usefully for engineering problems to simulate hydrodynamic impact problems, high explosive detonation in soil, underwater explosion phenomena, and bird strike in aerospace industry. SPH is a mesh free Lagrangian description of motion, that can provide many advantages in fluid mechanics and also for modeling large deformation in solid mechanics. Unlike ALE method, and because of the absence of the mesh, SPH method suffers from a lack of consistency than can lead to poor accuracy.

In this paper, devoted to ALE and SPH formulations for fluid structure interaction problems, the mathematical and numerical implementation of the ALE and SPH formulations are described. From different simulation, it has been observed that for the SPH method to provide similar results as ALE formulation, the SPH meshing, or SPH spacing particles needs to be finer than the ALE mesh. To validate the statement, we perform a simulation of a hydrodynamic impact problem, a rigid cylinder with initial velocity impacting a free surface of water. For this problem, the particle spacing of SPH method needs to be at least two times finer than ALE mesh. A contact algorithm is performed at the fluid structure interface, SPH-Structure, and SPH-ALE interface.

In Section 2, the governing equations of the ALE formulation are described. In this section, we discuss the advection algorithms used to solve mass, momentum and energy conservation in the multi-material formulation. Section 3 describes the SPH formulation, unlike ALE formulation which based of the Galerkin approach, SPH is a collocation method. The last section is devoted to numerical simulation of an underwater explosion and its impact on a deformable structure using both ALE and SPH methods. To get comparable between ALE and SPH, the particle spacing of SPH method needs to be at least two times finer than ALE mesh.

2 VOF Formulation

A brief description of the VOF formulation used in this paper is presented, additional details can be provided in Aquelet et al. (2006). To solve fluid structure interaction problems, a Lagrangian formulation is performed for the structure and an ALE formulation for the fluid and explosive materials, where fluid and explosive materials can be mixed in the same element, this element is referred as mixed element, since it contains two varied materials fluid and explosive as described in Fig. 1. A mixture theory is used to partition the material inside the element and compute the volume weighted stress from the constitutive model of each material as described in Souli et al. (Souli, Erchiqui 2012).

International Journal of Multiphysics
Volume 18, No. 2s, 2024
ISSN: 1750-9548

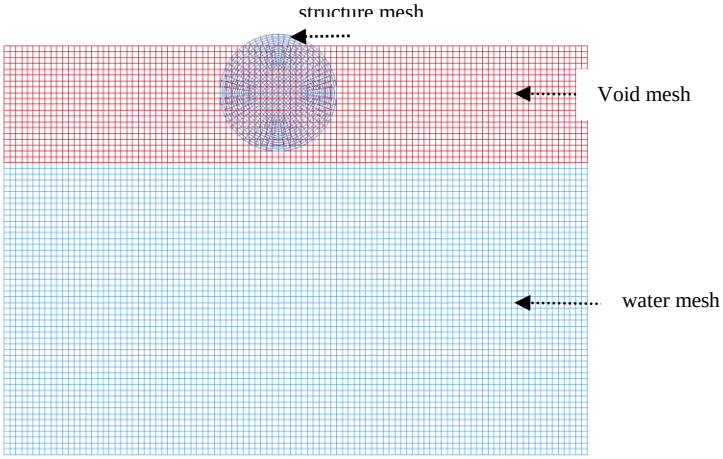


Fig. 1: Structure mesh and ALE mesh for water and Void

In the ALE description, an arbitrary referential coordinate is introduced in addition to the Lagrangian and Eulerian coordinates. The material derivative with respect to the reference coordinate can be described in Equation (2.1). Thus, substituting the relationship between material time derivative and the reference configuration time derivative leads to the ALE Equations in (2.1).

$$\frac{\partial f\left(X_i,t\right)}{\partial t}=\frac{\partial f\left(x_i,t\right)}{\partial t}+w_i\frac{\partial f\left(x_i,t\right)}{\partial x_i} \qquad (2.1)$$

where X_i is the Lagrangian coordinate, x_i the Eulerian coordinate, w_i is the relative velocity. Let denote by v the velocity of the material and by u the velocity of the mesh. In order to simplify the equations, we introduce the relative velocity $w = v - u$. Thus, the governing equations for the ALE formulation are given by the following conservation Equations (2.1) and (2.2).

(i) *Mass equation.*

$$\frac{\partial \rho}{\partial t}=-\rho\frac{\partial v_i}{\partial x_i}-w_i\frac{\partial \rho}{\partial x_i} \qquad (2.2)$$

(ii) *Momentum equation.*

$$\rho\frac{\partial v_i}{\partial t}=\sigma_{\mathrm{ij},j}+\rho b_i-\rho w_i\frac{\partial v_i}{\partial x_j} \qquad (2.3)$$

σ_{ij} is the stress tensor defined by $\sigma = -P.Id + \tau$, where τ is the shear stress from the constitutive model, and P the pressure. For fluid and explosive gas the pressure is computed through an equation of state.

For the structure, a classical elasto-plastic material model is used, where the shear strength is much higher than the volumetric strain.

(iii) *Energy equation.*

$$\rho\frac{\partial E}{\partial t}=\sigma_{\mathrm{ij}}v_{i,j}+\rho b_i v_i-\rho w_j\frac{\partial E}{\partial x_j} \qquad (2.4)$$

Note that the Eulerian equations commonly used in fluid mechanics by the CFD community, are derived by assuming that the velocity of the reference configuration is zero, $u = 0$ and that the relative velocity between the material and the reference configuration is therefore the material velocity, $w = v$. The term in the relative velocity in Equation (2.3) and Equation (2.4) is usually referred to as the advective term, and accounts for the

transport of the material past the mesh. It is the additional term in the equations that makes solving the ALE equations much more difficult numerically than the Lagrangian equations, where the relative velocity is zero.

There are two ways to implement the ALE equations, and they correspond to the two approaches taken in implementing the Eulerian viewpoint in fluid mechanics. The first way solves the fully coupled equations for computational fluid mechanics; this approach used by different authors can handle only a single material in an element as described for example in Ozdemir, Souli and Fahjan (2010). The alternative approach is referred to as an operator split in the literature, where the calculation, for each time step is divided into two phases. First a Lagrangian phase is performed, in which the mesh moves with the material, in this phase the changes in velocity and internal energy due to the internal and external forces are calculated. The equilibrium equations are:

$$\rho \frac{\partial v_i}{\partial t} = \sigma_{ij,j} + \rho b_i, \tag{2.5}$$

$$\rho \frac{\partial E}{\partial t} = \sigma_{ij} v_{i,j} + \rho b_i v_i. \tag{2.6}$$

In the Lagrangian phase, mass is automatically conserved since no material flows across element boundaries.

In the second phase, the advection phase, transport of mass, energy and momentum across element boundaries are computed; this may be thought as remapping the displaced mesh at the Lagrangian phase back to its original for Eulerian formulation or arbitrary position for ALE formulation using smoothing algorithms. From a discretization point of view of Equation. 2.5 and 2.6, one point integration is used for efficiency and to eliminate locking as it is mentioned by Erchiqui et al. (2015), Khan et al. (2008). The zero energy modes are controlled with an hourglass viscosity, see Hallquist (1998). A shock viscosity with linear and quadratic terms derived by Von Neumann and Richtmeyer (1950), is used to resolve the shock wave. The resolution is advanced in time with the central difference method, which provides a second order accuracy for time integration.

For each node, the velocity and displacement are updated as follows:

$$u^{n+1/2} = u^{n-1/2} + \Delta t.M^{-1}.\left(F_{\text{exl}} + F_{\text{int}}\right) \tag{2.7}$$

$$x^{n+1} = x^{n-1} + \Delta \text{tu}^{n+1/2}$$

Where F_{nt} is the internal vector force and F_{ext} the external vector force associated with body forces, coupling forces, and pressure boundary conditions, M is a diagonal lumped mass matrix. For each element of the mesh, the internal force is computed as shown in Equation 2.8.

$$F_{\text{int}} = \sum_{k=1}^{\text{Nelem}} - \int_k B^t.\sigma.\text{dv} \tag{2.8}$$

Where B is the gradient matrix and *Nelem* is the number of elements.

The time step size Δt, is limited by the Courant stability condition (see Benson (1992)), which may be expressed as

$$\Delta t \leq \frac{l}{c} \tag{2.9}$$

Where l is the characteristic length of the element, and c the speed of sound through the material in the element. For a solid material, the speed of sound is defined as:

$$c = \sqrt{\frac{K}{\rho}} \tag{2.10}$$

Where ρ is the material density, K is the module of compressibility.

3 SPH Formulation

The SPH method developed originally for solving astrophysics problem has been extended to solid mechanics by Libersky et al. (1993) to model problems involving large deformation including high velocity impact.

SPH method provides many advantages in modeling severe deformation as compared to classical FEM formulation which suffers from high mesh distortion. The method was first introduced by Lucy (1977) and Gingold and Monaghan (1977) for gas dynamic problems and for problems where the main concern is a set of discrete physical particles than the continuum media, see Fig 2. The method was extended to solve high velocity impact in solid mechanics, CFD applications governed by Navier-Stokes equations and fluid structure interaction problems.

(a) (b)

Fig. 2 : (a) FEM model, mesh and nodes (b) SPH model, particles.

3.1 SPH Formulation

It is well known from previous papers, Campbell et al. (2000) that SPH method suffers from lack of consistency, that can lead to poor accuracy of motion approximation. Unlike Finite Element, interpolation in SPH method cannot reproduce constant and linear functions.

A detailed overview of the SPH method is developed by Liu M.B. and Liu G.R. (2010), where the two steps for representing of function f, an integral interpolation and a kernel approximation are given by:

$$u(x_i) = \int u\,(y).\delta(x_i - y)\mathrm{dy} \qquad (3.1)$$

Where the Dirac function satisfies:

$$\delta(x_i - y) = 1, \text{ if } x_i = y \qquad (3.2)$$

$$\delta(x_i - y) = 0, \text{ if } x_i \neq y$$

$$\begin{cases} \delta(x_i\text{-}y) = 1 \text{if} x_i = y \\ \delta(x_i\text{-}y) = 0 \text{if} x_i \neq y \end{cases}$$

The approximation of the integral function Eq. 3.1 is based on the kernel approximation *W*, that approximates the Dirac function based on the smoothing length h.

$$W(d,h) = \frac{1}{h^{\alpha}}.\theta\left(\frac{d}{h}\right), \qquad (3.3)$$

that represents support domain of the kernel function, see Fig. 3.

International Journal of Multiphysics
Volume 18, No. 2s, 2024
ISSN: 1750-9548

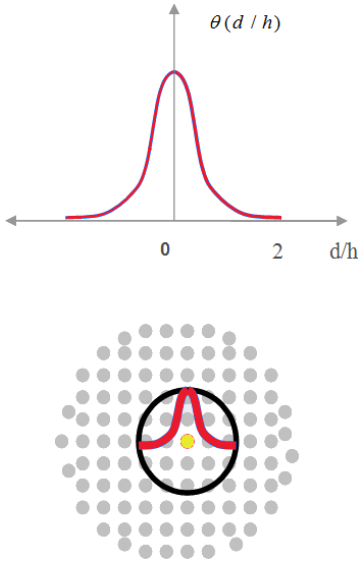


Fig. 3 : Kernel Function and its support domain for a 2D function.

So that Eq 3.1 becomes,

$$u(x_i) >= \int u\,(y).W(\|x-y\|,h)\mathrm{dy} \qquad (3.4)$$

Taking in consideration de support domain of the kernel function, the SPH approximation of a particle $\boldsymbol{x_i}$is obtained discretizing the integral into a sum over the particles that are within the kernel support domain as it is shown in Fig. 3.

$$u_{\mathrm{sph}}(x_i) = \sum \omega_j.u_j.W\left(\left\|x_i - x_j\right\|,h\right), \qquad (3.5)$$

where the weight $\omega_j = \frac{m_j}{\rho_j}$ is the volume of the particle.

Integrating by part Eq. 3.4 and considering the properties of the SPH interpolation and that $\nabla(u) = u.\nabla(1) - 1.\nabla(u)$, the SPH approximation for the gradient operator of a function is given by,

$$\nabla u_{\mathrm{sph}}(x_i) = \sum \omega_j.(u_i - u_j).\nabla W\left(\left\|x_i - x_j\right\|,h\right) \qquad (3.6)$$

Considering that

$$\frac{\nabla(P)}{\rho} = \frac{P}{\rho^2}\nabla(\rho) + \nabla\left(\frac{P}{\rho}\right), \qquad (3.7)$$

and applying the SPH interpolation on Navier-Stokes equations, one can derive a symmetric SPH formulation for Navier-Stokes equations such that the principle of action and reaction is respected and that the accuracy is improved. Finally, we have the following discretized set of equations :

(i) *Mass equation.*

$$\frac{D\rho_i}{\mathrm{Dt}} = \rho_i \sum \omega_j.\left(v_i^\beta - v_j^\beta\right).\frac{\partial W(\|x_i - x_j\|,h)}{\partial x_i^\beta} \qquad (3.8)$$

(ii) *Momentum equation.*

$$\frac{\mathrm{Dv}_i^\alpha}{\mathrm{Dt}} = \sum m_j.\left(\frac{\sigma_i^{\alpha\beta}}{\rho_i^2} + \frac{\sigma_j^{\alpha\beta}}{\rho_j^2}\right).\frac{\partial W(\|x_i - x_j\|,h)}{\partial x_i^\beta} + f_{\mathrm{ext}} \qquad (3.9)$$

(iii) *Energy equation.*

$$\frac{\mathrm{De}_i}{\mathrm{Dt}} = \frac{1}{2}\sum m_j.\left(\frac{P_i}{\rho_i^2} + \frac{P_j}{\rho_j^2}\right).\left(v_i^\beta - v_j^\beta\right)\frac{\partial W(\|x_i - x_j\|,h)}{\partial x_i^\beta} + \frac{\mu_i}{\rho_i}\varepsilon_i^{\alpha\beta}\varepsilon_i^{\alpha\beta} \qquad (3.10)$$

For constant and linear function, The standard SPH interpolation is not exact:

For $u(x_i) = 1$, $\sum \omega_j..W\left(\left\|x_i - x_j\right\|, h\right) \neq 1$ (3.11)

For $u(x_i) = x_i$, $\sum \omega_j.x_j.W\left(\left\|x_i - x_j\right\|, h\right) \neq x_i$ (3.12)

It is well know from previous studies (see Villa (1999,2005) and Oger (2006)), that Eq.3.11 and Eq.3.12 are exact only if the condition $\frac{\Delta x}{h} \to 0 \frac{\Delta x}{h} \to 0$ is fulfilled.

3.2 Constitutive material models for fluid

The nature of a particular fluid is represented into the statement of a constitutive law. Some of them, such as the Newtonian fluid relates the deviatoric stress to the local strain rate by establishing a linear equation for the evolution of the stress tensor and the deformation response.

If the rate of strain is linear and the fluid is isotropic the stress tensor is symmetric and the velocity gradient field is decomposed into two parts, the symmetric tensor $\dot{\gamma}$, and the vorticity tensor, ω, as written in Equation 3.1.

$$\bigtriangledown u = (\dot{\gamma} + \omega) / 2 \quad (3.13)$$

Where $\dot{\gamma}$ and ω are given by Equation 3.2 and 3.3 respectively.

$$\omega = (\bigtriangledown u - (\bigtriangledown u)^T), \quad (3.14)$$

$$\dot{\gamma} = (\bigtriangledown u + (\bigtriangledown u)^T) \quad (3.15)$$

In a newtonian fluid there is no preferred direction to the fluid deformation and the symmetric stress defines the isentropic flow. The deviatoric stress tensor reduces to a linear combination. Decomposing the isotropic part it is possible to achieve the constitutive equation for a Newtonian fluid given by Equation 3.16.

$$\tau = \mu\left(\dot{\gamma} - \frac{2}{3}(\bigtriangledown \cdot u) I\right) + \mu'(\bigtriangledown \cdot u) I \quad (3.16)$$

Where μ is the fluid viscosity and μ' is the dilational viscosity. For an incompressible fluid flow $\bigtriangledown \cdot u = 0$ and Equation 3.14 reduces to $\tau = \mu\dot{\gamma}$. In this case it is possible to write the stress tensor as Equation 3.17.

$$\sigma = -pI + \mu\dot{\gamma} \quad (3.17)$$

For a generalized fluid, Equation (3.5) becomes

$$\sigma = -pI + \eta\left(\dot{\gamma}\right)\dot{\gamma} \quad (3.18)$$

Where $\eta\left(\dot{\gamma}\right) = \mu$ in a newtonian fluid.

In the elastic solid the stress tensor depends locally on the material deformation; in a viscous fluid the stress tensor depends locally on the rate of fluid flow. For fluids with small molecules the Newtonian assumption is a good approximation and the energy dissipation mechanism is represented by viscosity in the constitutive law. If the applied flow can alter the fluid microstructure, the classical Newtonian approximation will fail in providing a dynamic model. A squematic sketch of the fluid structure interface is shown in Fig. 4.

International Journal of Multiphysics
Volume 18, No. 2s, 2024
ISSN: 1750-9548

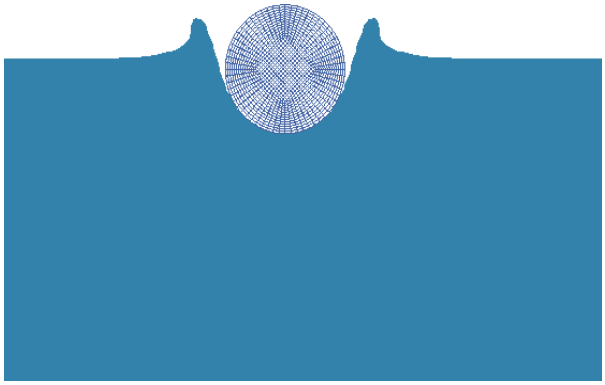


Fig. 4: Sketch of the structure interaction with the fluid.

Contact Algorithm

The coupling technique described in figure 4 is based on penalty contact algorithm used in structure mechanics, where a contact interface defined though the mesh separates the fluid mesh from the structure mesh. Thus contact forces applied from the fluid to the structure and conversely. In contact a master node is represented trough a fluid node whereas in coupling the master node is a fluid particle represented through its local coordinates in the fluid element. The penalty method imposes a resisting force to the slave node, proportional to the penetration through the master segment. This force is applied to both the slave node and the nodes of the master segment in opposite direction to satisfy equilibrium forces, as describe in figure 4.1.

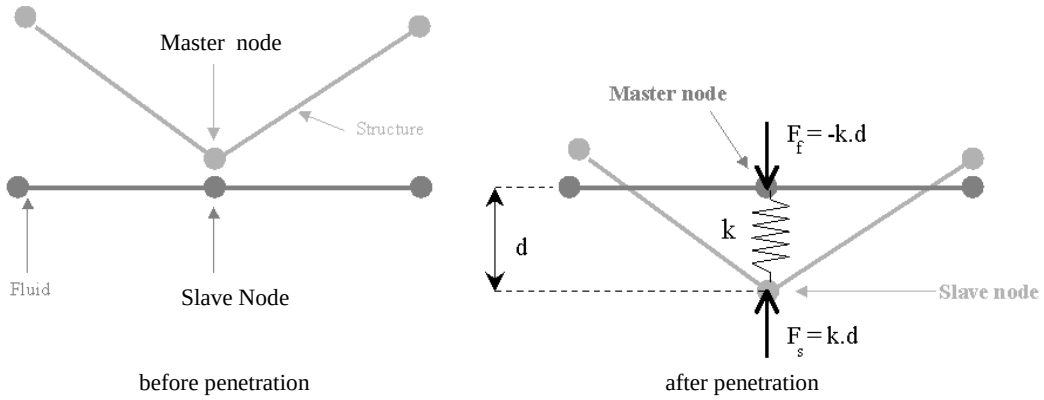


Figure-4.1: Penalty Contact algorithm before and after penetration

4 Numerical Simulations

4.1 VOF Model

In this example, we consider a simple structure in water subjected to prescribed velocity. The FEM Structure and ALE Fluid are modeled using eight nodes solid elements, while the SPH elements are generated at the center of the ALE elements. A sketch at Fig.5 illustrates the problem.

https://doi.org/10.52710/ijm.v18.i2s.979

International Journal of Multiphysics
Volume 18, No. 2s, 2024
ISSN: 1750-9548

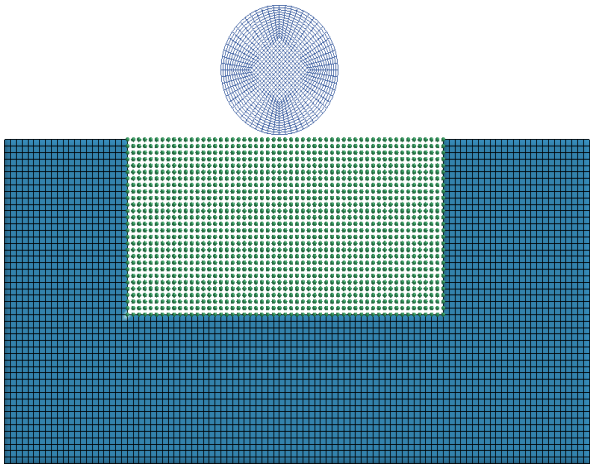


Fig. 5: Sketch of the mixed SPH-FEM model.

The fluid structure interaction is modeled for both ALE-FEM problems using kinematics type contact algorithm at the fluid structure interface. The kinematic constraints method is used where constraints are imposed on displacement and velocity of the contact interface, as well as impenetrability condition, as described Belytschko and Neal (1989). In addition, a smoothing algorithm is used with the ALE formulation, constraining the nodes to move uniformly along straight lines, to overcome high mesh distortion problems preserving mesh integrity.

Slip boundary conditions are applied to the ALE elements at the top, left and bottom boundaries, whereas the structure is fixed at the top and constrained to move in the X-direction at the bottom.

Definition of proper boundary conditions for SPH formulation is a challenge in the SPH theory. Several techniques have been developed in order to enhance the desired conditions, to stop particle from penetrating solid boundaries and also to complete the kernel function which is truncated by the physical domain for a particle close to the boundary. Among the different techniques, the ghost particle method (see Oger (2006), Colagrossi and Landrini (2003) and Doring et al (2006)) is known to be robust and accurate and is used in the simulations. When a particle is close to the boundary, it is symmetrised across the boundary with the same density, pressure, and temperature as its real particle such that mathematical consistency is restored. The ghost particles velocity is adjusted so that slip or stick boundary condition is applied.

In order to treat problem involving discontinuities in the flow variables such as shock waves, an additional dissipative term is added as an artificial pressure term. This artificial viscosity should be acting in the shock layer and should be neglected outside. This term is based on the classical Von Neumann and Richtmeyer (1950) artificial viscosity and is readapted to the SPH formulation as follow,

$$\pi_{ij} = \beta\mu_{ij}^2 - \alpha\mu_{ij} c_{ij}, \quad \text{if} \quad v_{ij}.r_{ij} < 0 \qquad \text{(In the shock layer)} \qquad (4.1)$$

$$\pi_{ij} = 0 \qquad \text{elsewhere} \qquad \text{(Outside the shock layer)}$$

$$\{\delta(x_i - y) = 1 \qquad if \qquad x_i = y\delta(x_i - y) = 0 if x_i \neq y$$

$$\pi_{ij}\begin{cases} \dfrac{B\mu_{ij}^2 - \alpha\mu_{ij} C_{ij}}{\rho_{ij}} & if \quad v_{ij} r_{ij} < 0 \\ 0 & elsewhere \end{cases}$$

$$\mu_{ij} = \frac{v_{ij}.r_{ij}}{r_{ij}^2 + \varepsilon h_{ij}^2},$$

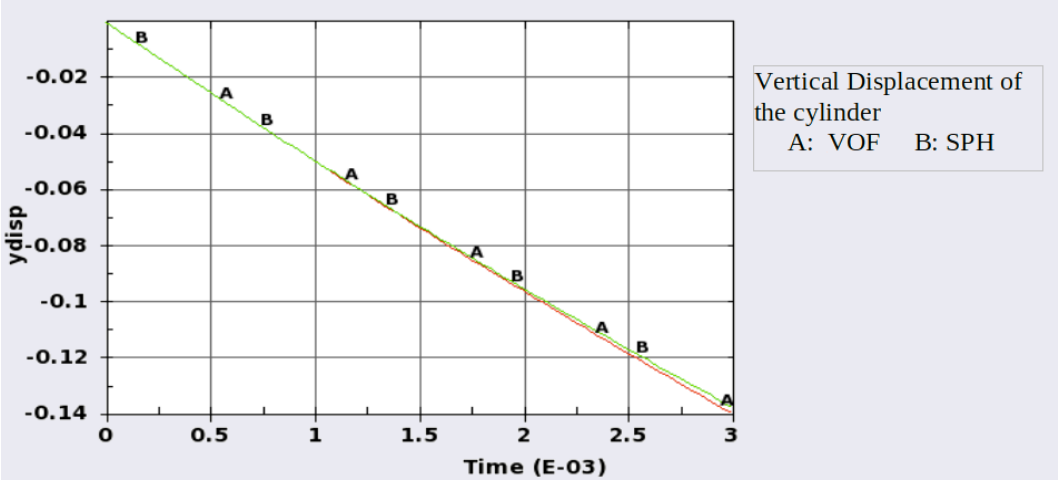
Where: $\rho_{ij} = \frac{(\rho_i + \rho_j)}{2}$ and $c_{ij} = \frac{(c_i + c_j)}{2}$

are respectively the average density and speed of sound, ε is a small perturbation that is added to avoid singularities, finally α and β are respectively the linear and quadratic coefficients.

5.2 Results Comparison and Mesh sensitivity analysis

For this problem ALE multi-material and SPH formulation are used to solve the problem up to physical termination time. As mentioned in the introduction, experimental tests for explosive detonation in fluid and the impact on surrounding structures, are costly to perform. The ALE formulation will be considered as reference solution to validate the SPH formulation since it has been validated against experiments in many applications involving explosions, blast impact and shock waves (see Puryear, Souli and Harrison, 2019).

In order to compare ALE and SPH formulations and to check the limits of the SPH formulation solving fluid structure interaction problems, two simulations were performed. A first one, using the same number of elements for both ALE and SPH methods, such that the space step dx separating two particles in the SPH case or the length of an element in the ALE case is the same. A second one, refining the SPH model by two in both direction in order to improve the accuracy of the simulation and to see how the SPH solution behave compared to the ALE one. In both simulations SPH particles are generated at the center of the ALE elements as it is shown in Fig.6.



A- VOF or ALE method
B- SPH-ALE method

Fig. 6: Vertical displacement of the cylinder.

To illustrate pressure wave propagation through water material, Fig.7 shows the pressure fringe at time t=60 microseconds from both ALE and non-refined SPH simulations. At this time of the simulation, the structure is not deformed, since the shock wave did not reach the structure yet, and it can be seen that SPH formulation represents well qualitatively the physics as the shock front and the expansion waves has the same shape in both ALE and non-refined SPH solutions. The Von_Mises Stress contour for ALE at time t = 60 ms are illustrated in Fig. 8.

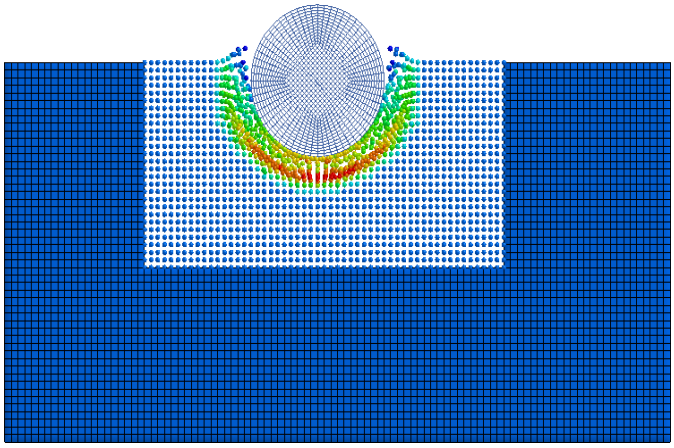


Fig. 7: Pressure contour for mixed formulation SPH-ALE

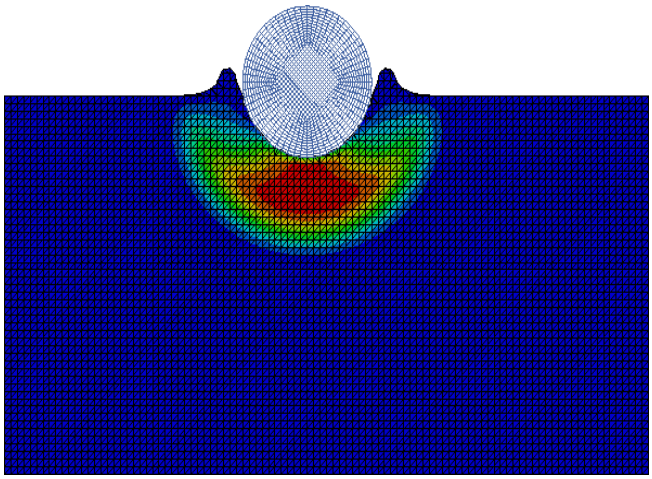


Fig. 8: Pressure contour for fluid at time t = 60 ms.

6 Conclusion

In this paper we present ALE methods as well as their limitations for specific problems. Underwater explosion is commonly solved using ALE formulation, in defence industry, some of these problems are solved using SPH method. For the last decade, SPH methods are gaining in accuracy numerical stability, and the use of SPH method is becoming more common in industry for solving fluid structure coupling problems. For instance, in aerospace, where bird impacts on aircraft are very common and cause significant safety threats to commercial and military aircraft. According to FAA (Federal American Aviation) regulations, aircraft should be able to land safely Souli and Erchiqui (2012). For decades engineers in aerospace industry were using ALE method to simulate bird impact on aircrafts, where a viscous hydrodynamic material is used for the bird. These applications require a large ALE domain for the coupling between the bird material and the surrounding structure, mainly when the bird is spread all over the space. According to technical reports from engineers in aerospace, ALE formulation is more CPU time consuming and requires more memory allocation that SPH method. In this paper, first we describe both ALE and SPH methods, and we compare numerical results between the two methods using similar mesh size, each ALE element is replaced by an SPH particle at the element center. Using a simple fluid structure interaction problem, it has been observed that using same mesh size for both methods, numerical results, displacement, velocity, and Von Mises stress on the structure, are under estimated with SPH method. When refining the SPH particles, where each ALE element is replaced by 4 SPH particles in two dimensional and 8 particles in three dimensions, numerical results from SPH method are in good correlation with those from ALE simulation; in terms of displacement, velocity, and Von Mises stress on the structure. Since the ultimate objective is the design of structure resisting to load blast, numerical simulations from ALE and SPH methods can be included in shape design optimization with shape optimal design

techniques, see Souli and Zolesio (1993), and material optimisation, see Erchiqui et *al* (2007). Once simulations are validated by test results, they can be used as design tool for the improvement of the system structure being involved.

References:

[1] Erchiqui, F ; Annasabi, Z ; Souli, M ; Slaoui-Hasnaoui, F (2015)
[2] 3D numerical analysis of the thermal effect and dielectric anisotropy on thawing frozen wood using microwave energy.
[3] International Journal of Thermal Sciences**,** Volume 89, March 2015, Pages 58-78
[4] Khan, MU ; Moatamedi, M ; Souli, M ; Zeguer, T (2008)
[5] Multiphysics out of position airbag simulation.
International Journal of Crashworthiness Volume 13, 2008 - Issue 2
[6] Erchiqui,F; Souli, M; Kanit,T; Imad,A; A Boudlal; El Moumen, A (2015)
[7] Characterization of Polymeric Membranes Under Large Deformations Using Fluid-Structure Coupling
[8] International Journal of Applied Mechanics 7(5); 2015
[9] Aquelet, N. Souli, M. Olovsson, L. (2006) Computational Methods Appl. Mech. Engrg. 195 110–132
[10] Belytschko, T.; Neal, MO. (1989) Contact-impact by the pinball algorithm with penalty, projection, and Lagrangian methods. Proceedings of the symposium on computational techniques for impact, penetration, and perforation of solids.
[11] AMD, vol.103, New York, NY: ASME, p. 97–140.
[12] Boyer, D.W. (1960) An experimental study of the explosion generated by a pressurised sphere, J. Fluid Mech. 9 401–429.
[13] Campbell, James & Vignjevic, Rade & Libersky, L.. (2000). A contact algorithm for smoothed particle hydrodynamics. Computer Methods in Applied Mechanics and Engineering. 184. 49-65. 10.1016/S0045-7825(99)00442-9.
[14] Colagrossi, A.; Landrini, M. (2003): Numerical simulation of interfacial flows by smoothed particle hydrodynamics. Journal of Computational Physics, 191:448.
[15] Doring, M. Alessandrini, B. Ferrant, P, (2006) Two-dimensional SPH simulations of wedge water entries. Journal of computational physics, Vol.213 (2), p.803-822
[16] Erchiqui, F.; Souli, M.; Ben Yedder, R. (2007): Non isothermal finite-element analysis of thermoforming of polyethylene terephthalate sheet: Incomplete effect of the forming stage. Polymer Engineering and Science, Vol.47, p.2129-2144.
[17] Hallquist, J.O. (1998) LS-DYNA Theory manuel. Livermore Software Technology Corporation.
[18] Kingery, C.N. and Bulmash G. (1984) Technical report ARBRL-TR-02555: air blast parameters from TNT spherical air burst and hemispherical burst. AD-B082 713, Aberdeen Proving Ground, MD: U.S. Army Ballistic Research Laboratory.
[19] Oger, G. (2006): Aspects théoriques de la méthode SPH et applications a l'hydrodynamique a surface.
[20] Ozdemir, Z.; Souli, M.; Fahjan Y.M. (2010) Application of nonlinear fluid–structure interaction methods to seismic analysis of anchored and unanchored tanks. Engineering Structures, vol.32, p.409-423
[21] Puryear, V.J. Souli M. Harrison B. (2019): velocity and Shape of Explosive Computationusing Multi-Material and ALE Formulations, Int. Jnl. of Multiphysics Volume 13 · Number 1, p 31-52
[22] Souli, M.; Erchiqui, F. (2011) Experimental and Numerical investigation of hyperelastic membrane inflation using fluid structure coupling. Computer Modeling in Engineering & Sciences, vol.77, p183-200.
[23] Souli, M.; Zolesio, J.P. (1993): Shape Derivative of Discretized Problems. Computer Methods in Applied Mechanics and Engineering, vol.108, p.187–199.
[24] Von Neumann, J.; Richtmeyer, R. D. (1950): A method for the numerical calculation of hydrodynamical shocks. Journal of Applied Physics, vol.21, p. 232.