Stratégie de couplage faible entre les méthodes FEM et SPH pour les simulations d'IFS

G. Fourey[a], G. Oger[a], D. Le Touzé[a], B. Alessandrini[a]

a. *LUNAM Université, Ecole Centrale Nantes, LHÉEA Lab. (CNRS UMR6598), 44300 NANTES*

Résumé :

Un couplage pour l'Interaction Fluide-Structure a été proposé par les auteurs entre la méthode Smoothed Particle Hydrodynamics (SPH) pour le fluide et la méthode Eléments Finis (EF) pour le solide. Ce couplage tire avantages des deux méthodes, à savoir la capacité de la méthode SPH à prendre en compte de grandes déformations du domaine fluide et la capacité éprouvée de prédiction du comportement des solides sous chargement instationnaire de la méthode EF. De plus, aucun algorithme spécifique n'est requis à l'interface solide-fluide pour éviter l'interpénétration des deux milieux. Tout ceci conduit à une implémentation relativement aisée du couplage. Des validations sont présentées en comparaison de résultats analytiques et expérimentaux. En particulier la conservation de l'énergie totale à travers le couplage est soigneusement suivie et analysée, démontrant la validité de ce couplage totalement explicite. Enfin le modèle est appliqué à un cas réaliste où les effets 3D ne peuvent être négligés.

Abstract :

A coupling strategy was proposed by the authors for solving Fluid-Structure Interactions by means of the Smoothed Particle Hydrodynamics (SPH) method for the fluid, and Finite Elements (FE) for the solid. This coupling takes advantage from both methods, namely from the capability of SPH to handle arbitrary large deformations of the fluid domain, and the known capability of FEs to predict the structural behavior of solids undergoing unsteady pressure loads. Moreover, no specific algorithm is required at the interface to prevent penetration of one medium into the other. All this permits a relatively easy implementation of the coupling. Validations are presented in comparison with analytical and experimental results. In particular, careful monitoring of the total energy evolution throughout the coupling is analyzed, demonstrating the effectiveness of this fully explicit coupling. Eventually, the solver is applied to a complex test case where 3D effects cannot be neglected.

Mots clefs : **interactions fluide-structure ; surface libre ; couplage explicite**

1 Introduction

In [1] an explicit parallel SPH-FEM coupling method was presented and first validation was performed by studying the impact at high velocity of an elastic solid on the free surface. Despite the use of a weak coupling strategy, the results obtained were in good agreement with analytical solution provided in [2]. One advantage of the coupling developed here is that no specific treatment such as sub-iterations is done to verify physical conditions at the fluid-structure interface. Even if good agreements between simulations and experimental or analytical data are found, it remained necessary to assess the coupling consistency. One way to do so is to monitor time conservation of the total energy of the whole system. In the present paper the evolution of the different flow energies are first analysed on simple fluid simulations without the presence of a structure. This permits to discuss of energy preservation in the SPH scheme stabilized with a Riemann solver. Then this monitoring is applied to FSI situations. In particular, another two-dimensional test case is presented in detail with evaluation of fluid and solid energies. It concerns the escape of a water column through an elastic gate [3]. Finally, first 3D validation of the coupling is performed, and the FSI model applicability to realistic complex situations is performed on an industrial test case.

2 SPH solver

In this section, the system of equations and the numerical scheme used in the SPH-Flow solver are briefly described. We then focus on the evaluation of kinetic, potential and internal energies.

2.1 Governing Equations

Locally, we have the following conservation equations of mass and momentum.

$$\frac{\partial \rho}{\partial t} = -\vec{\nabla}.(\rho \vec{v}) \tag{1}$$

$$\frac{\partial \rho \vec{v}}{\partial t} = -\vec{\nabla}.(\rho \vec{v} \otimes \vec{v} + pI_d) + \rho \vec{g} \tag{2}$$

The Tait's equation of state relating pressure to density of the barotropic fluid closes the system above.

$$p = \frac{\rho_0 a_0^2}{7} \left[\left(\frac{\rho}{\rho_0} \right)^{\gamma} - 1 \right] \tag{3}$$

2.2 Discrete scheme

In concrete terms, the field is described by a set of particles. The discrete scheme is written using the Lagrangian symmetrized form of equations, leading to, for the space discretization part :

$$\frac{d\vec{x_i}}{dt} = \vec{v_i} \quad ; \qquad \frac{d\omega_i}{dt} = -\omega_i \sum_j \omega_j \left(\vec{v_i} - \vec{v_j} \right) . \vec{\nabla} W_{ij} \tag{4}$$

and for Euler equations :

$$\frac{d\omega_i \rho_i}{dt} = -\omega_i \sum_j \omega_j 2\rho_e (\vec{v_e} - \vec{v}(x_{ij})).\vec{\nabla} W_{ij} \tag{5}$$

$$\frac{d\omega_i \rho_i \vec{v_i}}{dt} = \omega_i \rho_i \vec{g} - \omega_i \sum_j \omega_j 2[\rho_e \vec{v_e} \otimes (\vec{v_e} - \vec{v}(x_{ij})) + p_e I_d].\vec{\nabla} W_{ij} \tag{6}$$

where ρ_e and $\vec{v_e}$ are the solutions of the Riemann problem solved for each interaction at the interface x_{ij}. Velocity of this interface is given by (7).

$$\vec{v}(x_{ij}) = \frac{1}{2} \left(\vec{v_i} + \vec{v_j} \right) \tag{7}$$

Riemann problems are exactly solved using a Godunov numerical scheme [4]. A linear approximation combined to a limiter is used to extrapolate variables through the Monotone Upstream-centered Schemes for Conservations Laws (MUSCL) scheme [5].

2.3 Kinetic, potential and internal energies

Noting $m_i = \rho_i \omega_i$ the mass of a particle, the kinetic and potential energies can be calculated using respectively (7) and (8).

$$E_p = \sum_i m_i \vec{g}.\vec{x_i} \quad ; \qquad E_k = \frac{1}{2} \sum_i m_i ||\vec{v_i}||^2 \tag{8}$$

The internal energy u satisfies (9).

$$\rho \frac{du}{dt} = -p \vec{\nabla}.\vec{v} \tag{9}$$

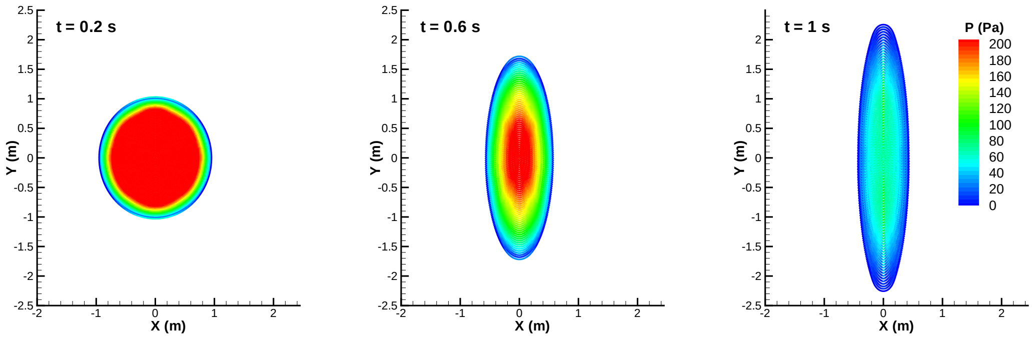

Figure 1 – Time evolution of the fluid for the patch test case

From (1) and (9) we get (10).

$$\left\{\begin{array}{l}\rho\frac{du}{dt}+p\vec{\nabla}.\vec{v}=0\\ \frac{d\rho}{dt}+\rho\vec{\nabla}.\vec{v}=0\end{array}\right. \Rightarrow \quad \frac{\rho}{p}\frac{du}{dt}=\frac{1}{\rho}\frac{d\rho}{dt}$$

$$\Rightarrow \int du=\int\frac{p}{\rho^2}d\rho \tag{10}$$

Replacing the pressure in (10) by its expression from Tait's equation we have :

$$u=\int\frac{\rho_0 c_0^2}{\gamma\rho^2}\left[\left(\frac{\rho}{\rho_0}\right)^{\gamma}-1\right]d\rho=\int\frac{\rho_0 c_0^2}{\gamma\rho_0^{\gamma}}\rho^{\gamma-2}d\rho-\int\frac{\rho_0 c_0^2}{\gamma}\frac{1}{\rho^2}d\rho \tag{11}$$

Finally, the internal energy can be computed using (12).

$$u=\sum_i\left(\frac{\rho_0^{1-\gamma}c_0^2}{\gamma}\frac{1}{\gamma-1}\rho_i^{\gamma-1}+\frac{\rho_0 c_0^2}{\gamma}\frac{1}{\rho_i}\right) \tag{12}$$

2.4 SPH simulations with energy evaluation

We first compute kinetic and internal energies on a simple test case. Velocity and pressure fields computed with (13) are initially imposed to a fluid disk of radius R centered at the origin [6]. Gravity is not modeled here.

$$\vec{v_0}(x,y)=(-A_0x,A_0y) \quad ; \qquad p_0(x,y)=\frac{\rho_0 A_0^2}{2}\left[R^2-\left(x^2+y^2\right)\right] \tag{13}$$

For this simulation, we assume $R = 1m$, $A_0 = 1s^{-1}$, and $\rho_0 = 1000kg.m^{-3}$. Speed of sound is fixed to ten times the maximum initial fluid velocity : $c_0 = 10m.s^{-1}$. As with all simulations presented in this paper, the ration $h/\Delta x$ is set to 1.23 and the cubic spline kernel proposed by Monaghan [7] is adopted. Pictures in figure 1 show time evolution of the fluid. Initial spacing of particles is set to $\Delta x = 0.02m$. As it can be seen in figure 2 and as expected, there is an exchange between internal and kinetic energies. Mass is strictly conserved. Total energy, i.e. the sum of internal and kinetic energies is plotted in figure 2 for three different discretizations. A slight numerical dissipation is observed, linked to the numerical diffusion introduced to stabilize the scheme (here through a Riemann solver). However, consistency is verified with a convergence order visibly higher than linear. This preliminary study permits to highlight the energetic behavior of the SPH scheme for the fluid. When no violent impact occur in the simulation, consistency of the total energy is obtained, which will enable us to monitor the coupling quality through global energy preservation when interacting with a structure. In case of violent impacts this analysis remains possible only up to the impact.

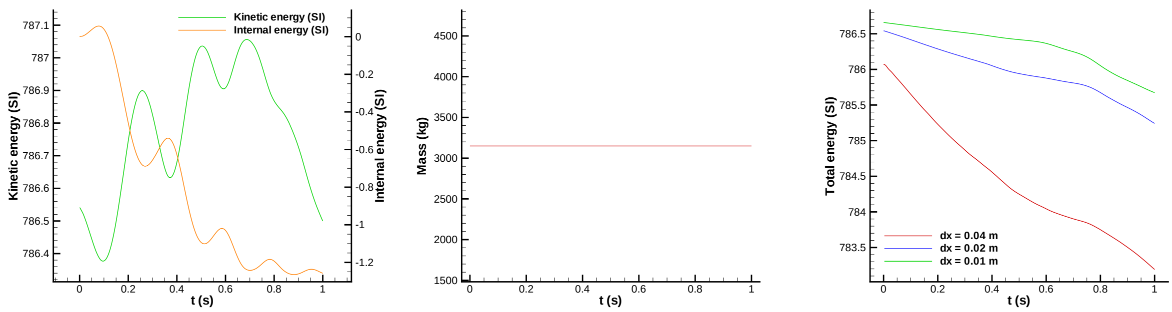

FIGURE 2 – Left : evolution of internal and kinetic energies for $\Delta x = 0.02m$. Middle : evolution of total mass. Right : convergence on total energy

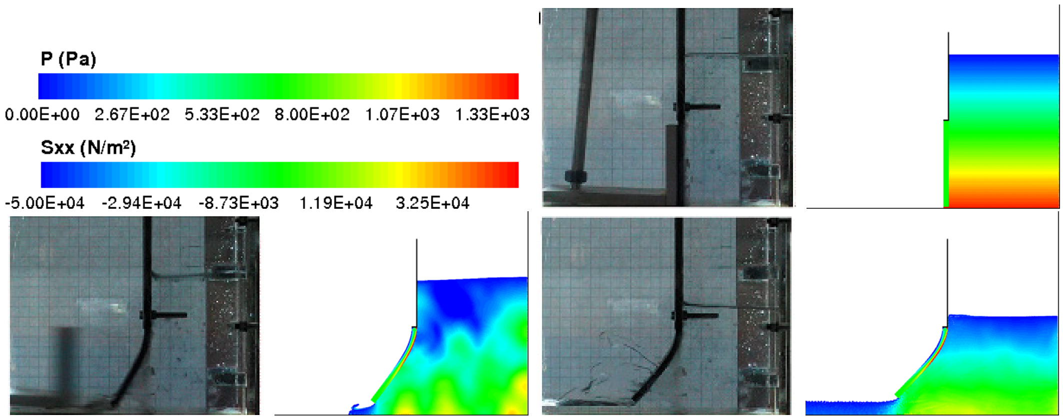

FIGURE 3 – Time evolution of the fluid for the dambreak test case (experiment taken from [3])

3 SPH-FEM coupling

The structure simulations showed in this paper are all performed with a standard FEM solver, namely here the open source *Code_Aster* solver. The coupling algorithm adopted is straightforward. At each time step, the two solvers exchange informations at the interface between fluid and structure. When it is done, the codes perform their calculations in parallel. The time step used for the coupling is based on the timestep needed for fluid computation. Of course, if the FEM solver needs a smaller time step to reach convergence then a subdivision of time steps can be applied ; in practice it is not needed in the following simulations. Regarding the boundary conditions applying at the interface, they are enforced through a ghost particle technique in the fluid SPH solver. As for the FEM solver, the pressure loading is computed by averaging the pressure of particles located next to the solid boundary.

3.1 2D validation test case

We presented in [1] a first validation of this coupling method between SPH and FEM on the impact at high velocity of an aluminium beam on free surface [1], recovering accurately beam deformation and local pressures in comparison to the analytical solution. Here we focus on another test case described in [3]. It deals with the escape of a water column initially locked by a deformable body. The width of the reservoir is $0.1m$ and the water level is set to $0.14m$. The plate is in rubber material with density $1100kg.m^{-3}$. The Poisson coefficient has been fixed to 0.499 but has no influence on the deformations of the solid. The dimensions of the plate are $79mm$ in length to $5mm$ in thickness. The experimental stress-strain curve of the deformable body used for the simulation can be found in [8]. The initial distance between particles is set to $1mm$ while the finite element mesh of the plate is composed of four

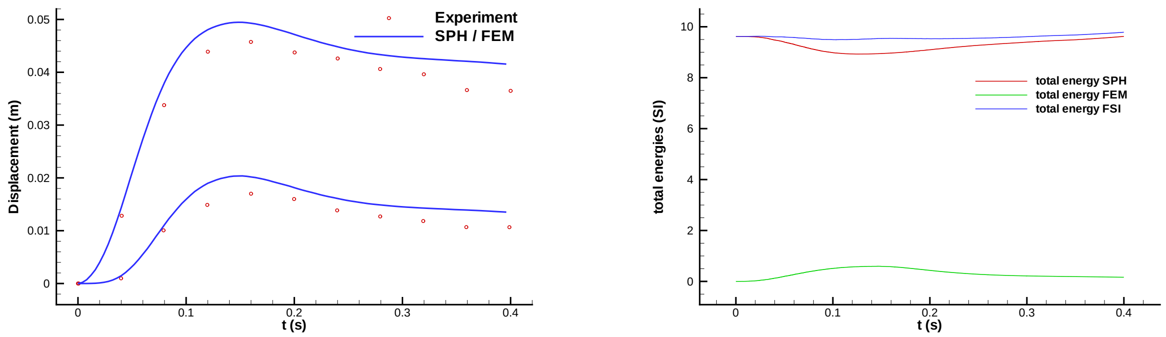

Figure 4 – Left : displacement of the free end of the plate. Right : evolution of SPH and FEM total energies

quadrangles in thickness to forty in length. The speed of sound is fixed to $c_0 = 30m.s^{-1}$, a sufficient value to comply with a Mach number below 0.1 throughout the simulation. Figure 3 show a comparison between the experiment and the results given by our simulation. A close agreement is visible. It can be noticed presence of an oscillating pressure field in the fluid. These high frequency acoustic variations are transmitted to the solid, and can generate instabilities in the coupling. However, one can note that the finite element solver is not affected by this high-frequency low energetic loading. This is thanks to the use of a Hilber-Hugues-Taylor (HHT) time advance scheme in the solid which enables a controlled dissipation of high frequency oscillations.

To obtain a more precise evaluation of the simulation quality, we can compare the displacement of the free end of the plate obtained by SPH-FEM to the experimental one, see figure 4. Despite a slight overestimation of displacements on the two directions a very good agreement is found between simulation and experiment. In any case the experiment is not strictly 2D and the friction on the wall of the rubber gate is not null. The evaluations of total energies for this FSI simulation are plotted in figure 4. As expected we have an exchange between fluid and solid energies. A slight decreasing of the total system energy is observed at the beginning and can be interpreted as a slight numerical dissipation. After some time, the increase of total energy for the fluid is due to the increase of internal energy, as already noted for the dambreak test case, and thus mainly corresponds to a slight residual contraction of the fluid domain volume. Nonetheless, the variation of the total energy throughout the simulation is very low, of the order of a few percents. This permits to verify the effectiveness of the simple explicit coupling methodology adopted. It is likely that the small time steps used, driven by the explicit nature of the weakly-compressible SPH fluid scheme used, permit to provide accuracy to this weak coupling, whereas iterating methods are usually employed for implicit solvers using larger time steps.

3.2 3D demonstrative test case

In order to illustrate the capabilities of the coupling method we performed a simulation of rubber material slipping on a rough ground in presence of fluid. This solid represents in reality only a small part of the tyre of a wheel, whose velocity is imposed by its upper part. The ground is fixed and solid-solid contact is performed within the FEM solver. As it can be seen in figure 5, the deformable solid first reaches the ground and then a translation movement is applied to its upper part. At this stage, the solid undergoes the loadings coming from contact and from fluid pressure and largely deforms. After its passage it is visible how it has dried the ground modeling the surface of a road. This problem of adhesion of a wheel on a wet rough ground is known to be numerically challenging very challenging. This permits to highlight the benefits of the coupling developed : the SPH capability to deal with complex fragmentations and reconnections of a free surface, and the FEM capability to perform contact simulations.

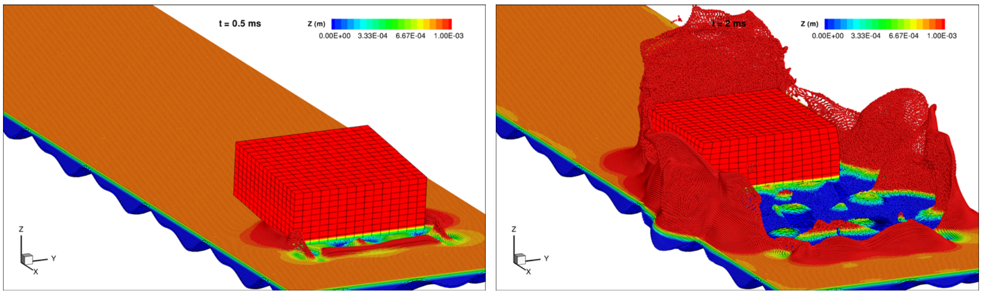

Figure 5 – Evolution of a rubber material slipping on a rough ground in presence of fluid

4 Conclusion

In previous communications we showed first validations on the simulation of FSI with a coupling between SPH and FEM methods. In this paper this coupling is further validated on different test cases, and extended to 3D complex applications. A monitoring of the energies in the fluid domain is introduced and permits to investigate the behavior of the SPH scheme in this respect. From this analysis it is then possible to monitor the energy preservation of the whole coupling, showing that the explicit weak coupling strategy adopted is effective and permits to obtain accurate results in comparison to experiments. This is in particular the case for the validation problem of a water column escaping through an elastic gate. The effectiveness of the method in 3D is then demonstrated, by comparison with 2D simulations on this same test case first, and then through the simulation of a complex test case. The latter one consists in the interaction between a piece of tyre and the wet ground when the wheel is rolling, which constitutes a challenging test case of industrial interest. On this test case the association of the meshless nature of the method adopted in the fluid enabling to capture the complex deformation of the water film, and the possibilities of the established FEM in the structure able to model the contact between the rough ground and the tyre, proves to be an effective strategy.

Acknowledgment

Authors wish to acknowledge DGA (French Ministry of Defense) for financing the PhD thesis of G. Fourey. The research leading to these results has also received funding from the European Community's Seventh Framework Programme (FP7/2007-2013) under grant agreement n.225967 "NextMuSE". Finally, authors want to thank Michelin for the authorization to publish the tyre illustrative test case.

Références

[1] Fourey G, Oger G, Le Touzé D and Alessandrini B 2010 SPH/FEM coupling to simulate Fluid-Structure Interactions with complex free-surface flows *5th International Spheric Workshop* 369-74

[2] Scolan Y-M 2003 Hydro-elastic behaviour of a conical shell impacting on a quiescent-free surface of an incompressible liquid *Journal of Sound and Vibration* **277** 163-203

[3] Antoci C, Gallati M and Sibilla S 2007 Numerical simulation of fluid-structure interaction by SPH *Comput. Struct.* **85** 879-90

[4] Vila J-P 1999 On particle weighted methods and smooth particle hydrodynamics *Mathematical Models and Methods in Applied Sciences* **9** 161-209

[5] Van Leer B 1979 Towards the ultimate conservative different scheme. V A second-order sequel to Godunov's method *Journal of Computational Physics* **32** 101-36

[6] Colagrossi A 2005 *A meshless lagrangian method for free-surface and interface flows with fragmentation*, PhD thesis, University of Rome La Sapienza

[7] Monaghan J J 1992 Smoothed particle hydrodynamics *Annu. Rev. Astron. Astrophys.* **30** 543-74

[8] Antoci C 2006 *Simulazione numerica dell'interazione fluido-struttura con la tecnica SPH*, PhD Thesis, University of Pavia