VII International Conference on Computational Methods for Coupled Problems in Science and Engineering
COUPLED PROBLEMS 2017
M. Papadrakakis, E. Oñate and B. Schrefler (Eds)

A FULLY EXPLICIT FLUID-STRUCTURE INTERACTION APPROACH BASED ON THE PFEM

SIMONE MEDURI*, MASSIMILIANO CREMONESI* AND UMBERTO PEREGO*

*Department of Civil and Environmental Engineering
Politecnico di Milano
Piazza Leonardo da Vinci, 32, 20133 Milano, Italy
e-mail: simone.meduri@polimi.it; massimiliano.cremonesi@polimi.it; umberto.perego@polimi.it

Key words: Fluid-Structure Interaction; PFEM-FEM coupling; domain decomposition; explicit dynamics.

Abstract. The efficient numerical simulation of Fluid-Structure Interaction (FSI) problems is of growing interest in many engineering fields. Staggered approaches are particularly interesting because they allow for the reuse of existing softwares. In this work we propose a staggered scheme based on the weakly compressible PFEM for the fluid domain and SIMULIA Abaqus/Explicit for the solid domain. The coupling is treated with a domain decomposition approach based on the Gravouil-Combescure algorithm. The main goal is to show the possibility of a fully explicit coupling with different time step size on the two phases (fluid and solid) and incompatible mesh at the interfaces. 2D test-cases will be presented to validate the proposed coupling technique. The explicit time integration scheme for both the fluid and solid subdomains, together with the explicit treatment of the coupling, makes this method appealing for applications in a variety of engineering problems with fast dynamics and/or a high degree of non-linearity.

1 INTRODUCTION

In the present work, a staggered approach for the solution of the FSI problem is proposed. The fluid domain is discretized with a Particle Finite Element Method (PFEM) [1], while the solid domain with a standard Finite Element Method. The weakly compressible formulation of the fluid flow, originally proposed in [2] for an explicit PFEM, is here used for the fluid domain. The PFEM has shown its capability in simulation of free surface flows in many applications (e.g. [3, 4, 5, 6]). Thanks to the Lagrangian formulation, the free surface is directly defined by the current position of the particles, while the governing equations are imposed like in standard FEM. When the mesh becomes too distorted, a fast remeshing algorithm is used to redefine the connectivities. SIMULIA Abaqus/Explicit has

been used for the solution of the structural domain. This allows to perform simulations with a complete and advanced description on the structural domain, including advanced structural material models and contact. The GC Domain Decomposition method [7] is here used for the coupling: the problem is solved independently on each subdomain and then linked at the interface using a Lagrange multiplier technique. The proposed method allows for different time-steps in the two subdomains and for non-conforming meshes at the interfaces between the solid and fluid domains. Moreover, this approach guarantees an explicit coupling at the interfaces.

2 GOVERNING EQUATIONS

The fluid domain Ω_f^t has been modeled through the weakly compressible Navier-Stokes equations:

$$\frac{d\rho_f}{dt} + \rho_f(\nabla_{\mathbf{x}} \cdot \mathbf{v}_f) = 0 \quad \text{in } \Omega_f^t \times [0,T] \tag{1}$$

$$\rho_f \frac{d\mathbf{v}_f}{dt} = \nabla_{\mathbf{x}} \cdot \boldsymbol{\sigma}_f + \rho_f \mathbf{b_f} \quad \text{in } \Omega_f^t \times [0,T] \tag{2}$$

where $\mathbf{x}$ are the coordinates in the current configuration, ρ_f is the fluid density, $\mathbf{v}_f$ the fluid velocity and $\mathbf{b_f}$ the external forces. The Cauchy stress tensor $\boldsymbol{\sigma}_f$ can be decomposed in its deviatoric and isotropic parts: $\boldsymbol{\sigma}_f = -p_f\mathbf{I} + \boldsymbol{\tau}_f$. In the weakly compressible framework, the pressure field p_f can be directly related to the density ρ_f through the Tait equation:

$$p_f(\rho_f) = p_{0,f} + K_f \left[\left(\frac{\rho_f}{\rho_{0,f}} \right)^{\gamma} - 1 \right] \tag{3}$$

where $p_{0,f}$ is the reference pressure, $\rho_{0,f}$ the reference density, $\gamma = 7$ the specific heat ratio and K_f the bulk modulus.

The solid domain Ω_s^t is governed by the momentum conservation equation:

$$\rho_s \frac{d\mathbf{v}_s}{dt} = \nabla_{\mathbf{x}} \cdot \boldsymbol{\sigma}_s + \rho_s \mathbf{b}_s \quad \text{in } \Omega_s^t \times [0,T] \tag{4}$$

where ρ_s is the solid density, $\mathbf{v}_s$ the solid velocity, $\mathbf{b_s}$ the external forces on the solid domain and $\boldsymbol{\sigma}_s$ represents the stress tensor on Ω_s^t. Standard Dirichlet and Neumann boundary conditions are applied on both the domains.

3 SPACE AND TIME DISCRETIZATION

Following a standard Galerkin finite element approach the semidiscretized equations of motion appear:

$$\mathbf{M}_f \frac{d\mathbf{V}_f}{dt} = \mathbf{F}_{ext,f} - \mathbf{F}_{int,f} = \mathbf{F}_f \quad \text{in } \Omega_f^t \times [0,T] \tag{5}$$

$$\mathbf{M}_s \frac{d\mathbf{V}_s}{dt} = \mathbf{F}_{ext,s} - \mathbf{F}_{int,s} = \mathbf{F}_s \quad \text{in } \Omega_s^t \times [0,T] \tag{6}$$

where $\mathbf{M}$ are the mass matrices, $\mathbf{V}$ the vector of nodal velocities and $\mathbf{F}_{int}$ and $\mathbf{F}_{ext}$ the vectors of internal and external nodal forces, respectively.

The discretized form of fluid mass conservation (2) can be obtained starting from the Lagrangian strong form, leading to:

$$\mathbf{M}_\rho \mathbf{R}_f = \mathbf{R}_0 \tag{7}$$

where $\mathbf{R}_f$ contains the nodal values of the density field (details can be found in [2]).

Equations (5-6) are integrated in time by means of the Central Difference Scheme [8]. It is important to recall that performing a mass lumping in the mass matrices, a fully decoupled system of equations can be obtained and fluid and solid velocities can be computed explicitly node by node.

4 THE COUPLING SOLUTION SCHEME

A staggered approach is here proposed for the solution of the coupled fluid-solid problem. The fluid sub-problem is solved numerically through the weakly compressible PFEM (see [2]). The solid sub-problem is analyzed using the commercial software Abaqus/Explicit [9].

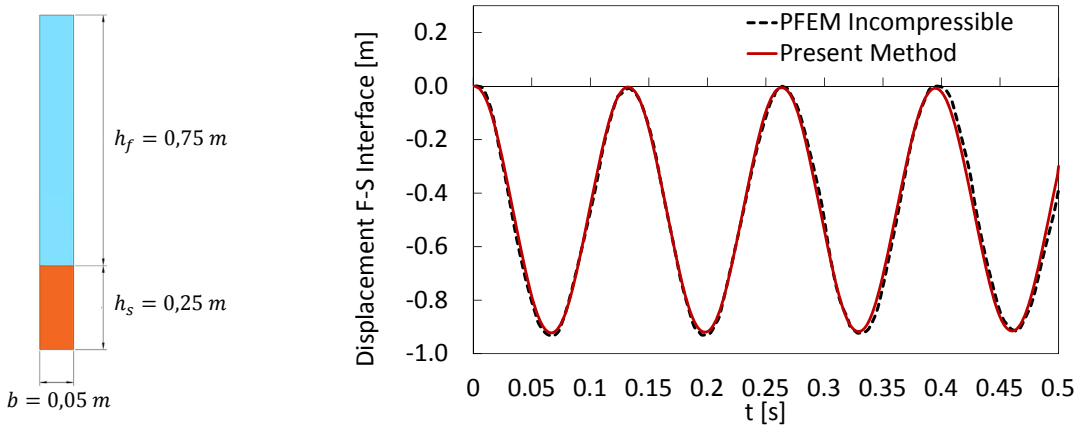
To couple the fluid and the solid domains the so-called GC method (Gravouil-Combescure) [7] has been selected. This algorithm, originally conceived for non-overlapping structural domains, has been recently extended to FSI problems [10]. The proposed approach splits the kinematic solution of each subdomain into two terms: *free* and *link*. The free solution is related to the free motion of each subdomain without considering the interaction with each other; the link solutions introduce a correction to account for the coupling. The correction terms are computed applying boundary tractions at the fluid-solid interface that play the role of Lagrange multipliers for the imposition of a kinematic constraint. The GC coupling algorithm allows for different time steps in the two sub-domains. Moreover, it allows for the use of incompatible meshes at the fluid-solid interface. This algorithm guarantees a stable strong coupling for the staggered solution of the FSI problem. In the present case of explicit fluid and structural solvers, the correction step consists in a small system of decoupled equations, resulting in a fully explicit coupled solver. A detailed description of the proposed approach can be found in [11].

5 NUMERICAL EXAMPLES

5.1 1D Fluid Structure Interaction

A first validation of the present method is obtained considering the 1D example presented in [12]. An elastic column, clamped at its bottom edge, is loaded by a column of water, as shown in Figure 1a. Due to the weight of the water, at the beginning of the analysis the system starts to oscillate vertically, as the horizontal displacement is constrained. The material parameters used for the water subdomain are:

- reference density $\rho_{0,f} = 1000\ kg/m^3$,



(a) (b)

Figure 1: 1D Fluid Structure Interaction. (a) Geometry of the problem. (b) Vertical Displacement of the fluid-structure interface: comparison with numerical results presented in [12]

- viscosity $\mu_f = 0\ Pa \cdot s$,
- bulk modulus $K_f = 2.2 \cdot 10^9\ Pa$;

while for the structural subdomain, the material parameters are:

- reference density $\rho_s = 1500\ kg/m^3$,
- Young Modulus $E_s = 2.3 \cdot 10^5\ Pa$,
- Poisson ratio $\nu_s = 0.4$.

Figure 1 shows the evolution in time of the vertical displacement at the fluid-structure interface. The results obtained with the present approach are compared with the ones presented in [12], showing a good agreement, both in terms of maximum displacement and frequency of the oscillations. It is worth noting that the structural parameters correspond to a very soft material, with a low Young's Modulus and density close to the fluid one. These conditions are known to be source of numerical difficulties [13, 12]. Consequently, despite its apparent simplicity, this example is a good test for the robustness of a FSI solver.

5.2 Deformable wedge impacting free surface water

In this example, the proposed method is tested on the challenging case of high speed impact of deformable structures against water at rest. The problem setting, presented in [14], is depicted in Figure 2. An aluminum wedge, with an inclination of 10°, is clamped at both its ends, where a vertical velocity of $V_{y,s} = -30\ m/s$ is imposed. The geometrical and mechanical parameters of the analysis are listed in table 1. The impact at high speed

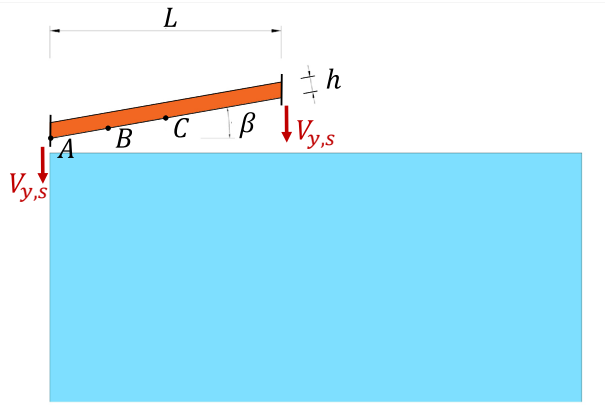
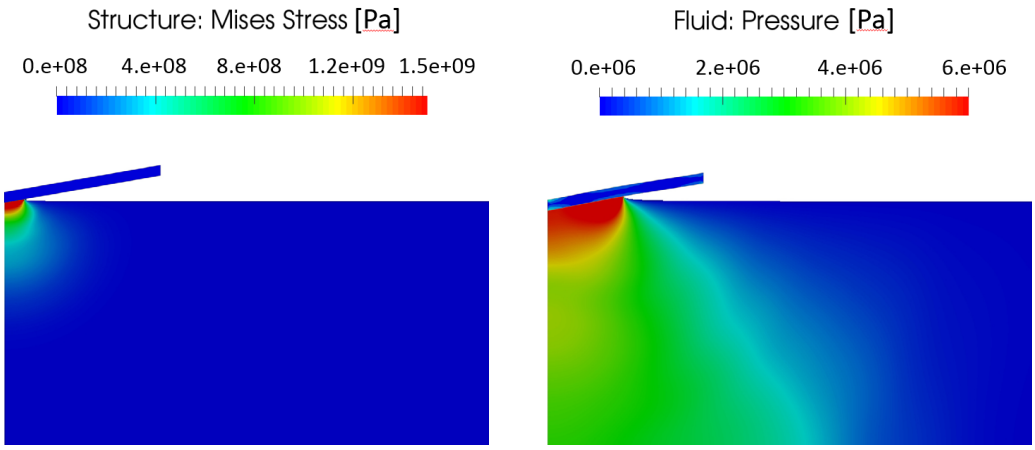


Figure 2: Deformable wedge impacting water free surface. Geometry of the problem

against water causes the deflection of the beam and the creation of lateral water jets. This is qualitatively shown in Figure 3, which collects some snapshots of the simulation. For a quantitative validation, Figure 4 shows the time evolution of the beam deflection at point B, as well as the time evolution of pressure at points A, B, and C, respectively. The results are compared with the numerical ones obtained with the SPH method presented in [14], and the semi-analytical solution presented in [15]. Once again, one can observe a very satisfactory agreement of the comparisons, considering the high level of complexity of the problem at hand.

Data	
L	$0.6\ m$
h	$0.04\ m$
β	$10°$
$V_{y,s}$	$-30\ m/s$
Fluid	
Density	$1000\ kg/m^3$
Viscosity	$0.001\ Pa \cdot s$
Bulk Modulus	$2.2 \cdot 10^9\ Pa$
Alluminium Beam	
Density	$2700\ kg/m^3$
Young Modulus	$67.5\ GPa$
Poisson ratio	0.34

Table 1: Deformable wedge impacting water free surface. Geometry and materials parameters.



(c) $t = 0.0003\ s$ (d) $t = 0.0013\ s$

(e) $t = 0.0023\ s$ (f) $t = 0.0028\ s$

Figure 3: Deformable wedge impacting water free surface. Snapshots of the simulation at different time instants. Contour plot of the pressure field in the fluid subdomain and Mises stress field in the structural one.

6 CONCLUSIONS

In the present work a fully explicit and fully Lagrangian PFEM-FEM coupling approach has been proposed for the solution of a fluid-structure interaction problem. The employment of a commercial software such as Abaqus/Explicit will allow to exploit all the available features for an advanced modelling of the structural part. The coupling has been performed through the GC domain decomposition method. This method synchronizes the independent solutions of the two subdomains, ensuring the strong coupling and the stability of the staggered approach. A fully explicit approach with different time step sizes and incompatible meshes at the interface has been proposed and validated. Two numerical examples have been used to assess the potential of the proposed numerical scheme.

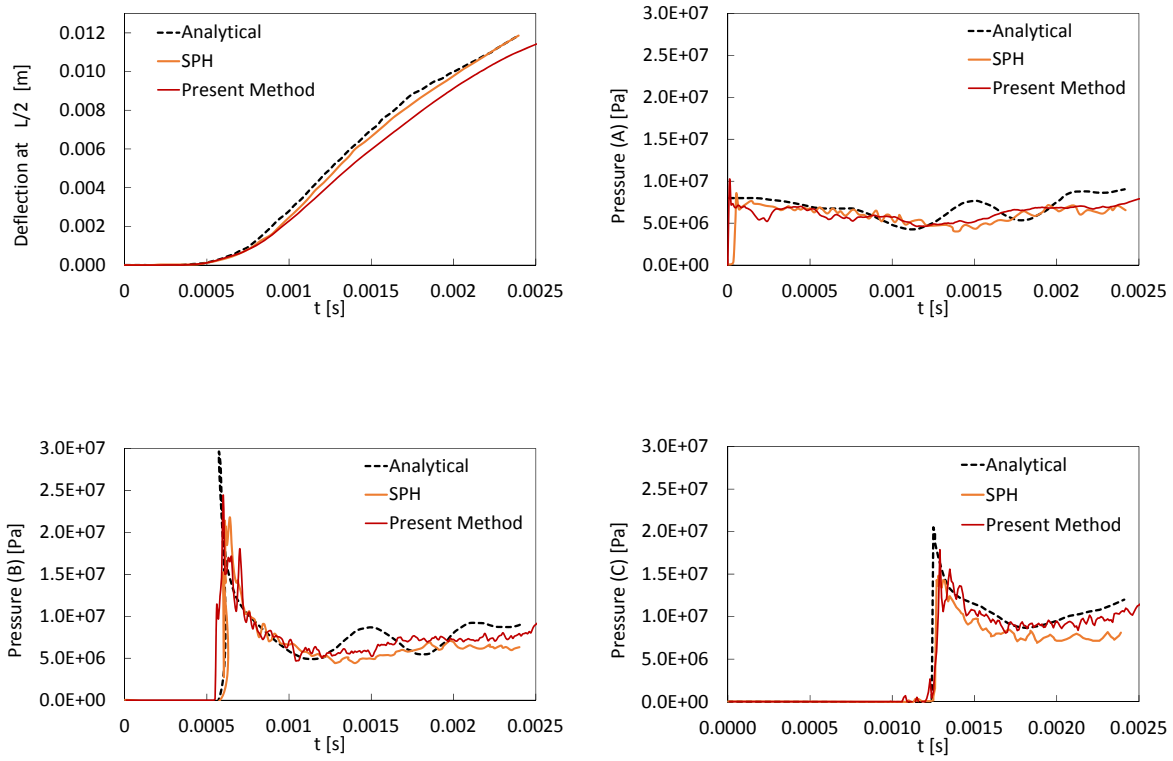


Figure 4: Deformable wedge impacting water free surface. Comparison with the analytical solution presented in [13] and the numerical one presented in [14]. (a): Time evolution of the deflection of the midpoint of the beam. (b)-(c)-(d) Time evolution of pressure at points A, B and C, respectively.

REFERENCES

[1] Oñate, E., Idelsohn, S.R., del Pin, F. and Aubry,R. The Particle Finite Element Method. An Overview. *Int J Computational Method* (2004) **1**:267-307.

[2] Cremonesi, M., Meduri, S., Perego, U. and Frangi, A. An explicit Lagrangian finite element method for free-surface weakly compressible flows. *Comp. Particle Mech.* (2017) DOI: 10.1007/s40571-016-0122-7

[3] Cremonesi, M., Ferri, F. and Perego U. A basal slip model for Lagrangian finite element simulations of 3D landslides. *Int J Numer Anal Met* (2017) **41**:30-53.

[4] Idelsohn, S. R., Mier-Torrecilla, M. and Oñate, E. Multi-fluid flows with the Particle Finite Element Method. *Computer Meth. in Appl. Mech. and Engng.* (2009) **198**:2750-2767.

[5] Idelsohn, S. R., Oñate, E., Del Pin, F. and Calvo, N. Fluid-structure interaction using the Particle Finite Element Method. *Computer Meth. in Appl. Mech. and Engng.* (2006) **195**:2100-2123.

[6] Zhu, M. and Scott, M. H. Direct differentiation of the quasi-incompressible fluid formulation of fluid-structure interaction using the PFEM. *Computer Particle Mech.* (2017) 1-13.

[7] Combescure, A. and Gravouil, A. A time-space multi-scale algorithm for transient structural nonlinear problems. *Mecanique et Industries* (2001) **2**:43-55.

[8] Hughes, T. J. R. The Finite Element Method: Linear Static and Dynamic Finite Element Analysis. Dover Publications. (1987)

[9] Dassault Systèmes SIMULIA. *Abaqus Documentation.* Providence, RI, USA, (2016).

[10] Nunez-Ramirez, J., Marongiu, J. C., Brun, M. and Combescure, A. A partitioned approach for the coupling of SPH and FE methods for transient non-linear FSI problems with incompatible time-steps. *Int. J. Numer. Meth. in Engng.*(2016) 1-46.

[11] Meduri, S., Cremonesi, M., Perego, U., Bettinotti, O., Kurkchubasche, A. and Oancea, V. A staggered fully explicit Lagrangian Finite Element Method for highly nonlinear Fluid-Structure-Interaction problems. *submitted.* (2017)

[12] Idelsohn, S. R., Del Pin, F., Rossi, R. and Oñate, E. Fluid-structure interaction problems with strong added-mass effect. *Int. J. Numer. Meth. Engng.* (2009) **80**:1261-1294.

[13] Causin, P., Gerbeau, J. F. and Nobile F. Added-mass effect in the design of partitioned algorithms for fluid-structure problems. *Computer Meth. in Appl. Mech. and Engng.* (2005) **194**:4506-4527.

[14] Oger, G., Guilcher, P. M., Jacquin, E., Brosset, L., Deu, J. B. and Le Touzé, D. Simulations of hydro-elastic impacts using a parallel SPH model. *Int. J. Offshore and Polar Engng.* (2010) **20**:181-189.

[15] Scolan, Y. M. Hydroelastic behaviour of a conical shell impacting on a quiescent-free surface of an incompressible liquid. *J. of Sound and Vibration* (2004) **277**:163-203.