V International Conference on Particle-based Methods – Fundamentals and Applications
PARTICLES 2017
P. Wriggers, M. Bischoff, E. Oñate, D.R.J. Owen, & T. Zohdi (Eds)

A FULLY EXPLICIT FLUID-STRUCTURE INTERACTION APPROACH BASED ON PFEM AND FEM

SIMONE MEDURI[1], MASSIMILIANO CREMONESI [1] AND UMBERTO PEREGO[1]

[1]Department of Civil and Environmental Engineering
Politecnico di Milano
Piazza Leonardo da Vinci, 32, 20133 Milano, Italy
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. Partitioned approaches for the solution of Fluid-Structure Interaction (FSI) problems are particularly interesting because, among other aspects, they allow for the reuse of existing software. In this work, we propose a partitioned 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. This approach allows for the use of different time step size on the two phases (fluid and solid) and incompatible mesh at the interfaces. The main goal of the proposed fomrulation is to show the possibility of a strong fluid-structure interaction coupling within af fully explicit sframework. 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 large scale applications in a variety of engineering problems with fast dynamics and/or a high degree of non-linearity.

1 INTRODUCTION

The numerical simulation of fluid-structure interaction (FSI) problems involving free-surfaces is of great interest in many engineering applications. Particle-based methods, like PFEM, are particularly suited for the analysis of free-surface flows and fluid-structure interaction with large displacements of the interface. In the current work, a partitioned approach for the solution of the FSI problem is proposed. The fluid domain is discretized using the Particle Finite Element Method (PFEM). The weakly compressible formulation of fluid flow, originally proposed in [1] for the PFEM, allows for a fully explicit solution scheme. Thanks to the Lagrangian formulation, the free surface is directly defined by the current position of the particles, coinciding with the nodes of the fluid finite element

mesh, while the governing equations are imposed like in the standard FEM (see e.g. [2, 3, 4, 5, 6]). When the mesh becomes too distorted, a fast triangulation algorithm is used to redefine the connectivities. The structural domain is instead analysed with a standard commercial explicit FEM (SIMULIA Abaqus-Explicit). The coupling between the fluid and solid domains is treated with the GC Domain Decomposition approach [9]. On each subdomain the problem is solved independently and then the two solutions are 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 allows for an explicit coupling, without iterations, between the two subdomains. 2D test-cases will be presented to validate the proposed coupling technique. The explicit, high parallelizable, 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 large scale engineering problems with fast dynamics and/or a high degree of non-linearity.

2 GOVERNING EQUATIONS

The weakly compressible Navier-Stokes equations are used to model the fluid domain Ω_f^t. Introducing the coordinates in the current configuration $\mathbf{x}$, the fluid density ρ_f, the fluid velocity $\mathbf{v}_f$ and the external forces $\mathbf{b_f}$, mass and momentum conservations read:

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

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

where $\boldsymbol{\sigma}_f$ is the Cauchy stress tensor which can be decomposed into its deviatoric and isotropic parts: $\boldsymbol{\sigma}_f = -p_f\mathbf{I} + \boldsymbol{\tau}_f$. Under the hypothesis of weak compressibility, the pressure field p_f can be expressed as a function of 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.

Introducing now the solid density ρ_s, the solid velocity $\mathbf{v}_s$, the external forces on the solid domain $\mathbf{b_s}$ and the stress tensor $\boldsymbol{\sigma}_s$, the momentum conservation equation can be written in the solid domain Ω_s^t as:

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

Standard Dirichlet and Neumann boundary conditions are applied on both the domains.

3 NUMERICAL APPROACH

A standard Galerkin finite element approach is applied to obtain the semidiscretized form of momentum conservation equations for the fluid and solid domains:

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

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

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.

Fluid mass conservation (1) is discretized 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 [1]).

A Central Difference Scheme [7] has been used to integrate equations (5-6) in time. 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. The proposed integration scheme is known to be conditionally stable, consequently an adaptive time step which guarantees the respect of the CFL condition has been used.

A partitioned approach is here proposed for the solution of the fluid-structure interaction problem. The fluid sub-problem is solved numerically through the weakly compressible PFEM (see [1]) while the solid sub-problem is analyzed using the commercial software Abaqus/Explicit [8].

4 THE COUPLING SCHEME

To couple the fluid and the solid subdomains, a partitioned approach, based on the so-called GC (Gravouil-Combescure) algorithm [9], has been used. This algorithm, originally conceived for non-overlapping structural domains, has been recently extended to FSI problems [10]. Applying the GC algorithm to FSI problems, the fluid and structural domain are solved independently, as if there was no interaction between them. The two separated analyses are then synchronized by considering a small system of constraint equations at the fluid-structure interface, ensuring the strong coupling of the partitioned approach. If explicit time integration is used for both the fluid and structural domain, as in the present case, the correction step consists in a small system of decoupled equations, resulting in a fully explicit coupled solver. The proposed algorithm allows for the use of incompatibles meshes at the fluid-solid interface and, moreover, guarantees the possibility of the use of different time steps in the two sub-domains. A complete description of the proposed approach can be found in [11].

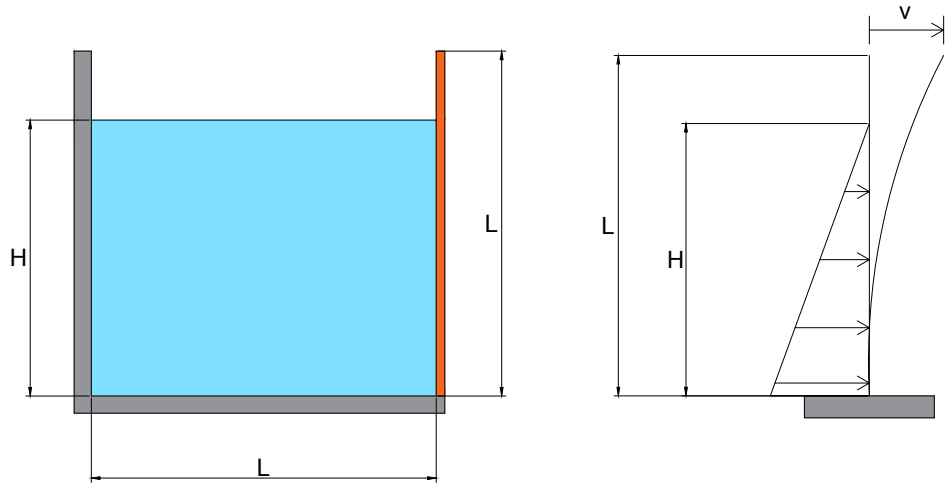


Figure 1: (a) Geometry of the numerical examples. (b) Equivalent Structural scheme.

5 NUMERICAL EXAMPLES

5.1 Beam under hydrostatic load

In the first validation case, the problem depicted in Fig. 1(a) and presented in [12] is considered. A tank full of water at rest has rigid walls on its left and bottom sides, while the wall on the right side is made of a deformable beam clamped at its bottom edge. This problem can be associated to a simple structural analysis scheme, i.e. a clamped beam subjected to hydrostatic load, as shown in Fig 1(b). The analytical solution in the framework of linear elasticity and small displacements provide a useful validation for the proposed method. The geometrical and mechanical parameters of the fluid and the structure are listed in Table 1. Conforming meshes of average size of 0.002 m are used, leading to 2400 linear triangles for the fluid and 50 two-node linear beam elements for the structure. The static analytical solution for the displacement of the upper edge of the beam represented in Fig 1(b) is given by:

$$v = \frac{p_{max} d}{E_s I_s} \left[\frac{H^4}{30} + (L - H) \frac{h^3}{24} \right]$$

where $p_{max} = \rho_f g H$ is the maximum value of the hydrostatic pressure at the bottom of the tank, g being the gravity acceleration, I is the moment of inertia of the beam cross-section and d is the out-of-plane thickness of the beam, which is assumed to be equal to the one of the fluid domain. Substituting the data of the problem at hand, one gets a horizontal displacement of $v = 0.09767\ mm$ and a maximum pressure at the bottom of the beam equal to $p_{max} = 784.8\ Pa$. These analytical values will be used for the validation of the numerical results obtained with the present approach.
Fig. 2(a) shows the time history of the horizontal displacement of the upper edge of the

Table 1: Beam under hydrostatic loading. Geometry and mechanical parameters.

Geometry	
L	$0.1\ m$
H	$0.08\ m$
d	$1\ m$
beam width	$0.012\ m$
Fluid	
Density	$\rho_f = 1000\ kg/m^3$
Viscosity	$\mu_f = 0.1\ Pa \cdot s$
Bulk Modulus	$K_f = 2.15 \cdot 10^9\ Pa$
Structure	
Density	$\rho_s = 2500\ kg/m^3$
Young Modulus	$E_s = 100 \cdot 10^6\ Pa$

beam, while Fig. 2(b) shows the time history of the pressure at the bottom edge. One can note that after the initial transient oscillations, the simulation reaches the steady state, which is in good agreement with the static analytical solution.

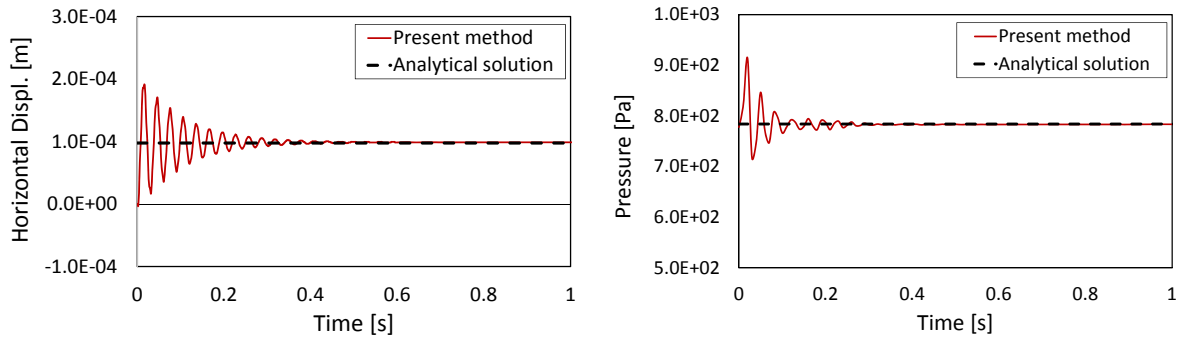
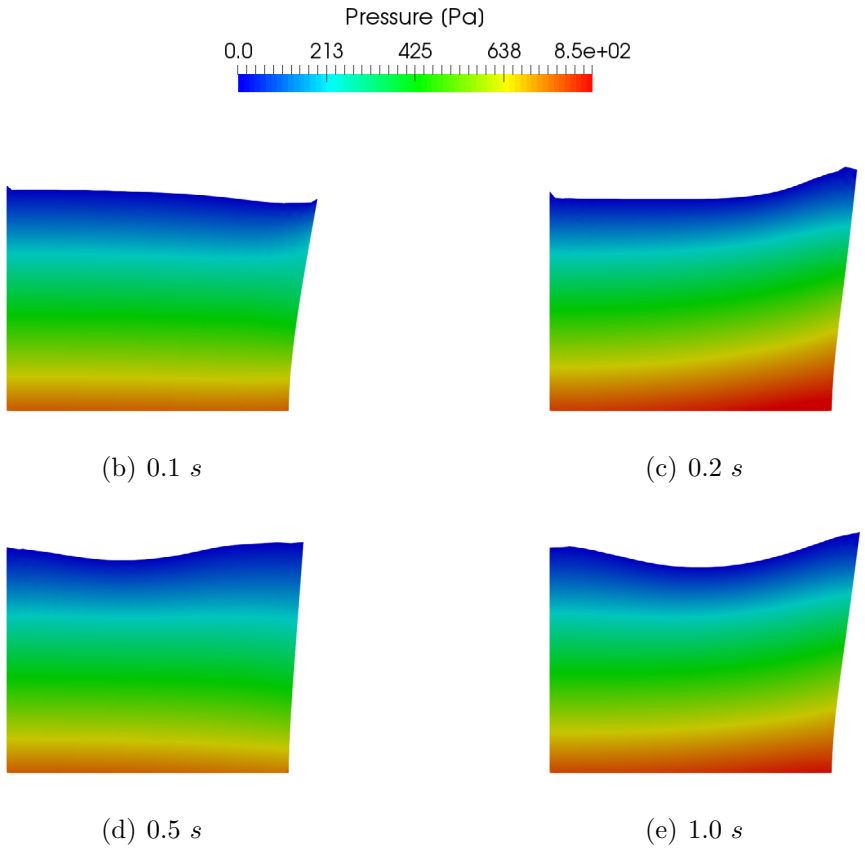


Figure 2: Beam under hydrostatic loading. (a) Time history of the horizontal displacement at the tip of the beam. (b) Time history of pressure at the bottom of the beam.

5.2 Tank interacting with a highly deformable beam

In this second example, a tank interacting with a highly deformable beam, originally proposed in [13], is considered. The geometry is the same of the previous case (Fig. 1(a)). The only different parameter is the beam Young's Modulus, which is $E_s = 1MPa$ in this case. Because of the lower beam stiffness, the analytical solution based on the hypothesis of small displacements and constant pressure is no longer valid. The beam deflection



(b) 0.1 s (c) 0.2 s

(d) 0.5 s (e) 1.0 s

Figure 3: Tank interacting with a highly deformable beam. Snapshots of the simulation at different time instants.

generates a sloshing wave which interacts dynamically with the beam itself, leading to their oscillations. Fig. 3 shows some snapshots of the simulation that are qualitatively in good agreement with the numerical results presented in [13]. For a more quantitative validation, Fig. 4 shows the time histories comparison of the horizontal displacement at the tip of the beam. One can observe that the frequency of oscillations is the same and, despite a small overestimation of their peaks, the comparison between the two curves can be considered very satisfactory.

6 CONCLUSIONS

In the present work a fully explicit and fully Lagrangian PFEM-FEM coupling scheme has been proposed for the solution of fluid-structure interaction problems. The GC domain decomposition method has been used to couple the two subdomains. The GC algorithm ensures the strong coupling and the stability of the partitioned approach. A fully explicit approach with different time step sizes and incompatible meshes at the interface has been proposed and validated. The comparison of 2D-examples with analytical and numerical

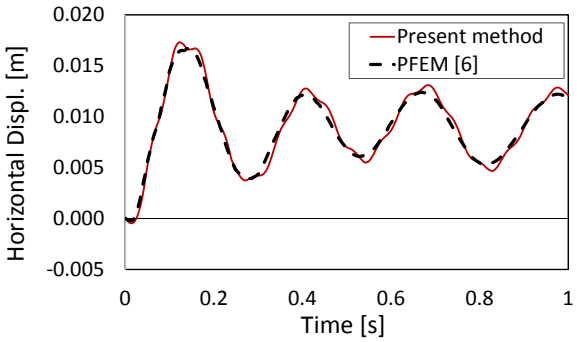


Figure 4: Tank interacting with a highly deformable beam. Time history of the horizontal displacement at the tip of the beam.

results presented in the literature shows the effectiveness and accuracy of the proposed coupling scheme. The resulting fully explicit solver is appealing for its possible application in a large variety of large scale engineering problems with fast dynamics and/or a high degree of non-linearity.

REFERENCES

[1] 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

[2] 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.

[3] 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.

[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., 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.

[6] 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.

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

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

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

[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 partitioned fully explicit Lagrangian Finite Element Method for highly nonlinear Fluid-Structure-Interaction problems. *International Journal for Numerical Methods in Engineering.* (2017) DOI:10.1002/nme.5602

[12] Zhu, M. and Scott, M. H. Direct differentiation of the Particle Finite-Element Method for fluid-structure interaction using the PFEM. *J Struct. Eng.* (2016) **142**(3).

[13] 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.