

AN EXPLICIT LAGRANGIAN APPROACH FOR 3D SIMULATION OF FLUID-STRUCTURE-INTERACTION PROBLEMS

S. Meduri¹, M. Cremonesi¹ and U. Perego¹

¹ Politecnico di Milano, Piazza Leonardo da Vinci 32, 20133 Milan, Italy.
massimiliano.cremonesi@polimi.it

Key words: FSI, PFEM-FEM Coupling, Explicit Solver, Lagrangian Formulation

Abstract. A Lagrangian fully explicit approach for the co-simulation of three-dimensional problems of Fluid-Structure Interaction (FSI) is here presented. The fluid domain is modelled as a weakly compressible material through an explicit version of the Particle Finite Element Method (PFEM). The structure is modelled with standard FEM through the commercial software Abaqus/Explicit. The strong coupling of the partitioned approach is ensured by the Gravouil and Combescure (GC) algorithm. The GC scheme allows for the use of incompatible space-time discretizations in the two subdomains. The coupling problem leads to a small explicit system of decoupled equations when conforming meshes are used at the interface, while, in the case of non-conforming meshes, a weakly coupled system is obtained.

A novel and efficient mesh smoothing procedure is proposed to remove bad quality tetrahedra that may arise in the frequent remeshing framework of the PFEM, since they can lead to an overly small stable time step size. The fully Lagrangian description of the present method is particularly effective in problems characterized by strong variations in the fluid boundaries. Moreover, the highly parallelizable and fully explicit nature of the equations of the global solver is appealing for real-scale engineering applications with fast dynamics and/or a high degree of non-linearity.

1 INTRODUCTION

The efficient numerical simulation of large scale three-dimensional Fluid-Structure Interaction (FSI) problems is of growing interest in many engineering fields. Partitioned approaches base on the co-simulation concept are particularly effective because they allow for the reuse of existing software. The present work presents a partitioned approach coupling an explicit version of the Particle Finite Element Method (PFEM) [18, 10, 12] based on the assumption of a weakly compressible fluid [3, 21] and a standard commercial FEM (SIMULIA Abaqus/Explicit) [1] for the structural domain. This allows for the use of all its advanced features, such as the wide library of constitutive laws and finite elements, contact interactions and large deformations. The Gravouil and Combescure (GC)

algorithm [9] is here employed for structure-to-fluid coupling: this method allows for different time-steps in the two subdomains and for non-conforming meshes at the interface. The resulting interface problem consists of a system of fully decoupled equations, in the case of conforming meshes, of and only weakly coupled equations otherwise, leading to an efficient global explicit solver. The explicit time integration can be an appealing choice for many engineering applications where the time step size is intrinsically small, namely in large scale real applications with fast dynamics and/or high degree of non-linearity. Furthermore, the fully Lagrangian formulation of this coupled approach is particularly effective for the description of free surface flows and FSI problems with large structural displacements, i.e., when the fluid boundaries can vary significantly. The fluid boundaries are indeed automatically defined by the position of the mesh nodes, with no need for interface tracking algorithms. Unfortunately, the Lagrangian motion of the fluid nodes leads very quickly to overly distorted fluid meshes: according to the PFEM approach, this can be overcome generating new meshes whenever the current one gets too distorted. This feature of the PFEM approach becomes particularly demanding in the case of an explicit 3D solver. On the one hand it must be underlined that the Delaunay Tassellation may generate bad tetrahedra in 3D meshes that can lead to a vanishing stable time step size in an explicit solver. This problem has been widely addressed in the literature, and many advanced algorithms have been proposed to improve the mesh quality [6, 7, 13, 5]. On the other hand, these advanced algorithms are too expensive to be frequently applied during the simulation as required in the PFEM remeshing framework, because they are designed to improve only the initial fixed mesh in for standard FEM. For these reasons a novel efficient smoothing technique has been developed to produce a regular mesh, with a reasonably large stable time increment for the explicit solver. This smoothing algorithm is fully explicit and parallelizable, because it exploits the same architecture of the fluid solver thanks to an elastic analogy. The proposed approach is here validated through a test case proposed in the literature. The comparison with the available solutions confirms the accuracy of the partitioned solver. Moreover, an assessment of the computational costs of the numerical analyses shows the effectiveness of the smoothing technique.

2 GOVERNING EQUATIONS

Let us consider a weakly compressible fluid occupying the evolving domain Ω_f^t . The motion of the fluid in the time interval $[0, T]$ is governed by the momentum equation and the mass equation, respectively:

$$\rho_f(\mathbf{x}, t = 0) = \rho_f(\mathbf{x}, t) J_f(\mathbf{x}, t) \quad \text{in } \Omega_f^t \times [0, T] \quad (1)$$

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

where \mathbf{x} are the coordinates in the current configuration, ρ_f is the fluid density, \mathbf{v}_f is the fluid velocity and \mathbf{b}_f the external body forces. Moreover, $J_f(\mathbf{x}, t)$ is the determinant of the deformation gradient, so that $d\Omega_f^t = d\Omega_f^0 J_f(\mathbf{x}, t)$. Finally, the Cauchy stress tensor $\boldsymbol{\sigma}_f$ is the sum of its hydrostatic and deviatoric parts: $\boldsymbol{\sigma}_f = -p_f \mathbf{I} + \boldsymbol{\tau}_f$.

In the framework of weakly compressible fluids, the pressure field can be related to the density field through the Equation of State [2]:

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

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

As far as the solid domain Ω_s^t is concerned, the momentum conservation equation reads:

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

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 Cauchy stress tensor on Ω_s^t . Standard boundary and initial conditions are applied on both domains to complete the formulation of a well posed problem.

3 SPACE AND TIME DISCRETIZATION

Let us introduce a standard Galerkin finite element approach with linear interpolation of velocities, fluid density and pressure. The semi-discrete version of the equations of motion (2)-(4) is obtained:

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

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

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 equivalent nodal forces, respectively. The discretized version of the fluid mass conservation (1) is obtained introducing the vector of the current fluid nodal densities \mathbf{R}_f (more details can be found in [3]):

$$\mathbf{M}_\rho \mathbf{R}_f = \mathbf{R}_0 \quad (7)$$

where \mathbf{R}_0 are the initial fluid nodal densities.

The explicit Central Difference Scheme (CDS) has been used to integrate equations (5)-(6) in time. It is remarkable that the mass lumping performed on the matrices \mathbf{M}_f , \mathbf{M}_s , \mathbf{M}_ρ leads to a global system of fully decoupled equations. On the other hand, the CDS is only conditionally stable and an adaptive time step is computed in order to fulfil the CFL stability condition:

$$\Delta t^{n+1} = C_N \min_e \left(\frac{r_e^{in}}{c_e} \right) \quad (8)$$

where C_N is a safety parameter, c_e is the speed of dilational waves in the fluid depending on the element density and r_e^{in} is a characteristic size of the the e -th deformed element: in the present work, the radius of the tetrahedron insphere is considered.

4 COUPLING SCHEME

A partitioned strategy is employed in the present approach to solve the global system of governing equations of the FSI problem: the explicit version of the PFEM [3] is used for the fluid subdomain, while the commercial software SIMULIA Abaqus/Explicit [1] is used for the structural subdomain. The two solvers are coupled with the Gravouil and Combescure (GC) algorithm [9], which is a Domain Decomposition method initially proposed for structural dynamics problems and then extended to FSI problems for example in [14]. The key idea behind the algorithm is to split the kinematic solution of each subdomains into two terms, denoted as *free* and *link* solutions, respectively. The former is related to the free motion of each subdomain as if they were not interacting with each other. The latter evaluates the correction to this solution. This is done applying boundary tractions at the fluid-structure interface that play the role of Lagrange multipliers for the imposition of the kinematic constraint. These tractions restore the compatibility at the interface and ensure the strong coupling of the partitioned approach.

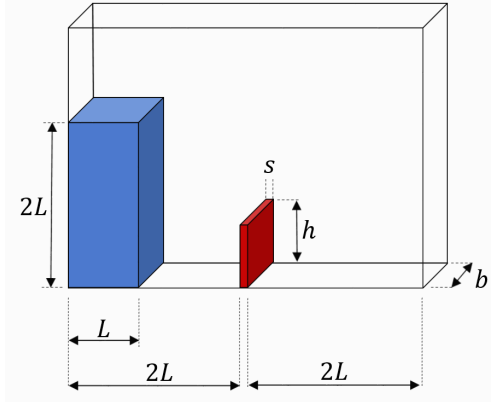
The GC algorithm allows for the use of different time integration schemes in the two subdomains, as well as non conforming meshes at the interface. When two explicit solvers are used for the fluid and structure with conforming meshes, the *link* solution consists of a small system of decoupled equations, leading to an overall fully explicit global solver. On the contrary, when non conforming meshes are used, a small system of weakly coupled equation is needed at the interface. A complete description of the proposed approach can be found in [15].

5 ELASTIC ANALOGY FOR 3D MESH SMOOTHING

The problem of mesh distortion related to the adoption of a Lagrangian description for viscous fluid flows is solved in the PFEM approach through the introduction of a new mesh whenever the current one gets too distorted. This is done exploiting the effectiveness of the Delaunay Tassellation, combined with the α -shape method [4] to recover the correct domain boundaries. However, while in 2D the Delaunay algorithm ensures excellent properties in terms of mesh quality, in 3D bad tetrahedra may be generated. More in details, tetrahedra with almost coplanar nodes called “slivers” can be created by the Tassellation. This represents a strong limitation for the use of explicit solvers, as the consequent small insphere radii lead to vanishing stable time step size, because of Equation (8).

For this reason, a novel efficient mesh smoothing approach based on an elastic analogy has been developed to improve the worst elements insphere radii, with a computational cost compatible with the frequent remeshing procedure required by the PFEM. The mesh regularization is the result of the deformation produced in a fictitious elastic problem under the action of a suitable distribution high fictitious pressures in high distorted tetrahedra.

The smoothing step is applied to a domain Ω_s , which is constituted by all the elements having the insphere radius smaller than an *a priori* fixed threshold, together with all the surrounding ones which share at least one node with them.



Mesh	
n. fluid elem.	450 <i>k</i>
n. struct. elem.	3800
Analysis	
Time simulated	1 <i>s</i>
Total duration	32 <i>h</i>
Solver duration	77.8 %
Remeshing duration	11.4 %
Smoothing duration	8.9 %
n. Threads	8

Figure 1: Dam Break with elastic obstacle. Geometry and parameters of the analysis.

This elastic based smoothing has been combined with the sequential Geometric Element Transformation (GETMe) approach presented in [19] to obtain a strategy effective on a wider range mesh configurations that usually occur upon remeshing of of complex 3D fluid flows. The resulting final smoothing algorithm is explicit and parallelizable and exploits the same structure of the fluid solver, simplifying the overall architecture of the method. Its efficiency allows to obtain remarkable improvements on the stable time steps of the explicit fluid solver in a reasonable computational time that is compatible with its frequent application due to the remeshing procedure of the PFEM approach. A more detailed description of the smoothing algorithm can be found in [16].

6 EXAMPLES

6.1 DAM BREAK WITH ELASTIC OBSTACLE

Let us consider the dam break with obstacle proposed in [20, 11] and reproduced with various approaches in the literature [8, 14, 15]. Among them [15] refers to the 2D version of the same PFEM co-simulation here described, [11, 8] use monolithic PFEM schemes for FSI, [20] presents a FEM approach, while [14] presents an SPH-FEM partitioned approach with a coupling scheme that is similar to the present method. The initial geometry is depicted in Figure 1 and the following values for the geometrical parameters have been used: $L = 0.146 \text{ m}$, $h = 0.08 \text{ m}$, $s = 0.012 \text{ m}$, $b = 0.15 \text{ m}$. The fluid material parameters are: density $\rho_f = 1000 \text{ kg/m}^3$ and viscosity $\mu = 0.001 \text{ Pa} \cdot \text{s}$; the solid domain is modelled with a linear elastic material with density $\rho_s = 2700 \text{ kg/m}^3$, Young's Modulus $E_s = 1 \text{ MPa}$ and Poisson ratio $\nu_s = 0$. The fluid domain has been discretized with 120 *k* nodes and an initial mesh of 450 *k* tetrahedra, while the structural domain has been discretized with 3800 8-nodes brick elements.

On the left side of the tank, a column of water is initially sustained by a rigid wall that is instantaneously removed at the beginning of the analysis, letting the water flow inside the tank and hit an elastic deformable body clamped in the middle of the tank. The impact bends the obstacle, while the resulting long wave hits the right rigid wall of the tank and rebounds back impinging for the second time on the obstacle, which starts

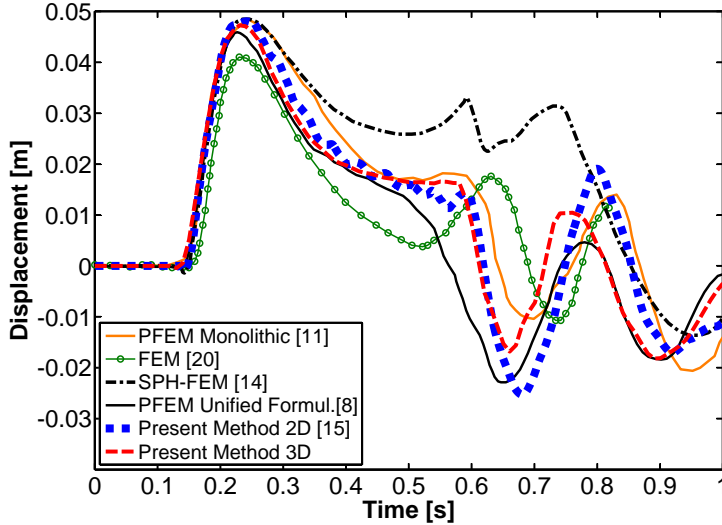


Figure 2: Dam Break with elastic obstacle. Time evolution of the horizontal displacement of the tip of the deformable structure.

to oscillate. Some snapshots of the simulation are reported in Figure 4, while the time evolution of the horizontal displacement of the top of the elastic body is plotted in the graph in Figure 2. One can observe a very good agreement in terms of timing and peak value of the structural displacement with the results presented in the literature up to 0.5 s, which proves the capability of the model to represent the impact of a mass of fluid with a flexible structure and its deformation. The beam oscillations after 0.5 s have some discrepancies but it's difficult to uniquely define a trend among the presented solutions because of the high level of complexity of the dynamics of the problem after the second impact.

Let us now consider the two graphs in Figure 3 to provide a quantitative evaluation of the effectiveness of the smoothing procedure. The first one plots the stable time step computed directly on the Delaunay mesh whenever a new grid is generated during the analysis. The second one represents the corresponding stable time step size after the action of the combined Smoothing procedure. One can observe how the mesh generated by the Delaunay Tessellation could not be used in an explicit solver, because the consequent stable time step would lead to an unacceptable analysis duration. The smoothing procedure here presented shows a remarkable improvement: the mean value of the stable time step is increased of almost 2 orders of magnitude from $3.05 \cdot 10^{-8}$ s to $1.45 \cdot 10^{-6}$ s, while the minimum value of $4.9 \cdot 10^{-10}$ s gains almost three orders of magnitude to $2.71 \cdot 10^{-7}$. This great improvement is obtained in an acceptable amount of time, as it can be noticed from the table in Figure 1: the overall smoothing duration is less than the remeshing one and their sum takes around the 20 % of the overall time duration of the analysis, which is comparable to standard the performances of 3D PFEM approaches.

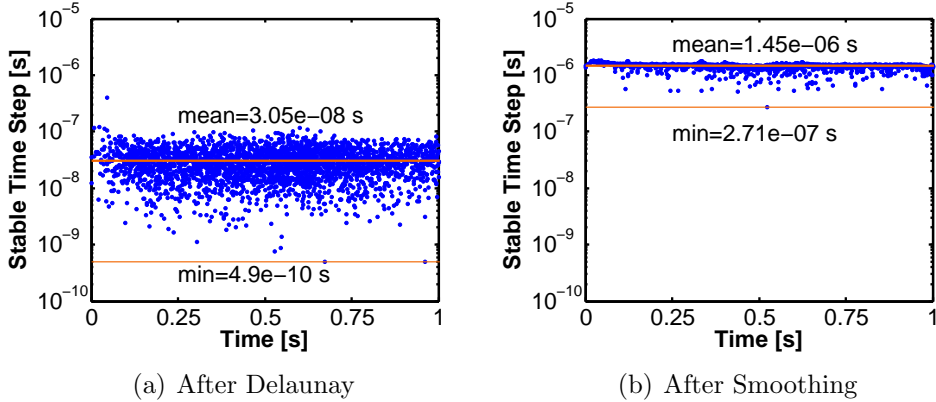


Figure 3: Dam Break with elastic obstacle. Plot of the stable time step at each remeshing computed after: (a)Delaunay Triangulations, (b)Combined Smoothing Procedure

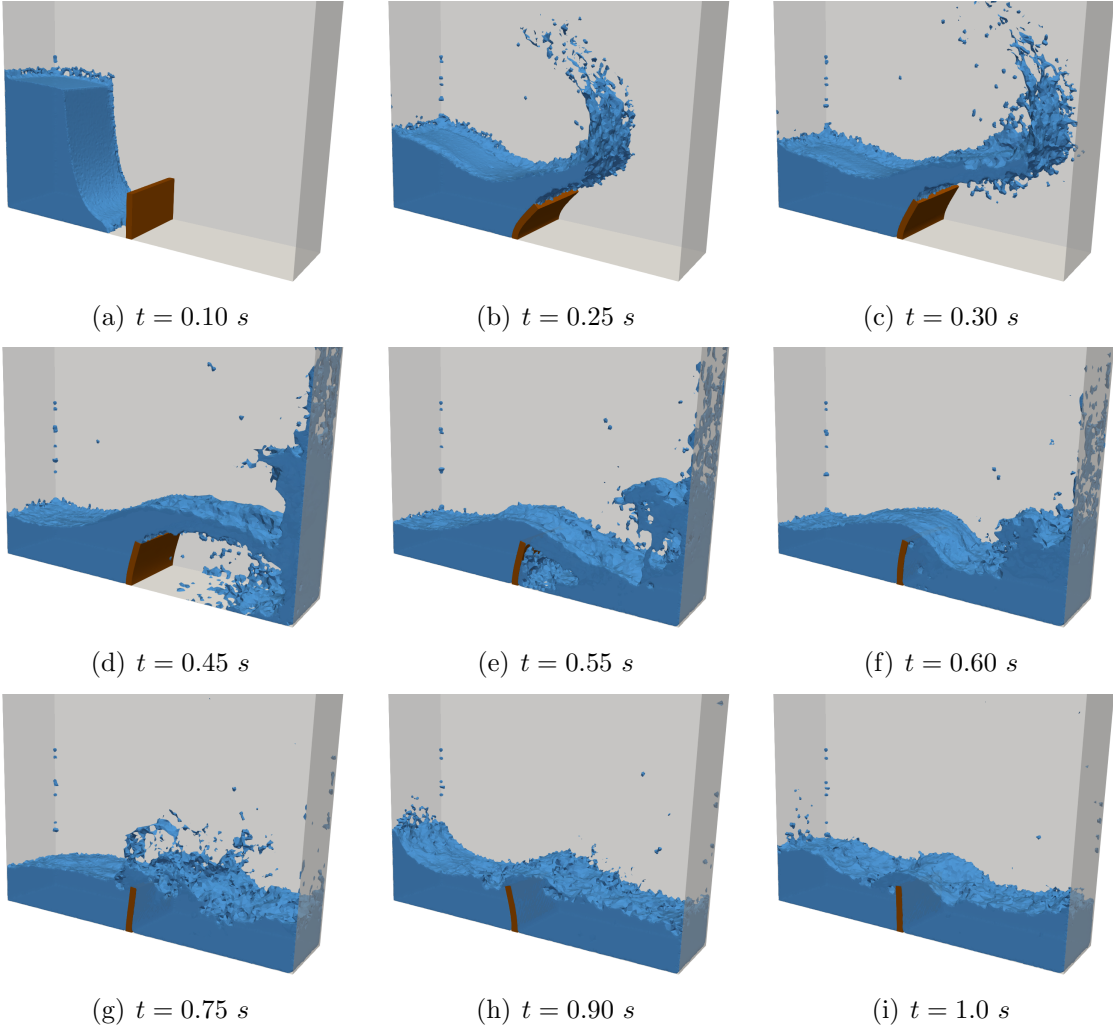


Figure 4: Dam Break with elastic obstacle. Snapshot of the simulation at different time steps

7 CONCLUSIONS

This work presents a fully Lagrangian explicit partitioned approach for FSI problems coupling the Particle Finite Element Method for the fluid subdomain and the commercial FEM software Abaqus/Explicit for the structural subdomain. The Lagrangian description of the Particle Finite Element Method for the fluid subdomain is particularly effective for free surface flows and large structural displacements. The use of a commercial software for the structure allows for using in the simulations of all its advanced functionalities, such as a wide library of finite elements and material constitutive laws, or contact interactions.

The coupling achieved by means of the Gravouil and Combescure algorithm, which ensures strong coupling as well as the possibility to use incompatible space-time discretizations in the two subdomains. The resulting interface coupling problem is given by a small system of fully decoupled equations for the case of conforming meshes, and only weakly coupled otherwise. In the 3D framework, the fast mesh generators of the PFEM approach may generate meshes including bad quality tetrahedra, that can lead to an unacceptable increase in the computational time when an explicit integration scheme is employed. Consequently, a novel efficient smoothing technique based on an elastic analogy has been presented. The presented numerical test confirms the accuracy of the coupled approach, as well as the effectiveness of the smoothing procedure.

REFERENCES

- [1] Dassault Systèmes SIMULIA. *Abaqus Documentation*. Providence, RI, USA, (2016).
- [2] Bernard-Champmartin, A. and De Vuyst, F. A low diffusive Lagrange-remap scheme for the simulation of violent air-water free-surface flows. *J of Comp Physics* (2014) **274**:19-49.
- [3] 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) **4**:357-369.
- [4] Edelsbrunner, H. and Mücke, E. Three-dimensional Alpha Shapes. *ACM Transactions on Graphics* (1994) **15**:255-261.
- [5] Edelsbrunner, H. and Shah, N.R. Incremental Topological Flipping Works for Regular Triangulations. *Algorithmica* (1996) **15**:223-241.
- [6] Field, D. A. Laplacian smoothing and Delaunay triangulation. *Int J Num Meth Biomed Engng* (1988) **4**:709-712.
- [7] Freitag, L. A. and Ollivier-Gooch, C. Tetrahedral mesh improvement using swapping and smoothing. *Int J Numer Meth Engng* (1997) **40**:3979-4002
- [8] Franci, A., Oñate, E., and Carbonell J. M. Unified Lagrangian formulation for solid and fluid mechanics and FSI problems. *Computer Meth in Appl Mech and Engng* (2016) **298**:520-547.

- [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] Idelsohn, S., Oñate, E. and Del Pin, F. The particle finite element method: A powerful tool to solve incompressible flows with free-surfaces and breaking waves *Int J Numer Meth Engng* (2004) **61**:964-989
- [11] Idelsohn, S.R., Marti, J., Limache A., and Oñate, E. Unified Lagrangian formulation for elastic solids and incompressible fluids: application to fluid-structure interaction problems via the PFEM. *Computer Meth in Appl Mech and Engng* (2008) **197**:1762-1776.
- [12] Idelsohn, S., 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.
- [13] Klingner, B. M. and Shewchuk, J. R. Multi-fluid flows with the Particle Finite Element Method. *Proc 16th Int Meshing Roundtable* (2007) 3-23.
- [14] Li, Z., Leduc, J., Nunez-Ramirez, J., Combescure, A. and Marongiu, J. C. A non-intrusive partitioned approach to couple smoothed particle hydrodynamics and finite element methods for transient fluid-structure interaction problems with large interface motion *Comp Mech* (2001) **55**:697-718.
- [15] Meduri, S., Cremonesi, M., Perego, U., Bettinotti, O., Kurkchubasche, A. and Oancea, V. A partitioned fully explicit Lagrangian finite element method for highly nonlinear fluidstructure interaction problems. *Int J Numer Meth Engng* (2018) **113**:43-64
- [16] Meduri, S., Cremonesi, M. and Perego, U. Elastic analogy mesh smoothing for three-dimensional explicit Particle Finite Element Method. *SUBMITTED*
- [17] Oakley, D.R. and Knight, N.F. Adaptive Dynamic Relaxation algorithm for non-linear hyperelastic structures Part II. Single-processor implementation. *Computer Meth in Appl Mech and Engng* (1995) **126**:91-109.
- [18] 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.
- [19] Vartziotis, D., Wipper, J. and Schwald, B. The geometric element transformation method for tetrahedral mesh smoothing. *Computer Meth in Appl Mech and Engng* (2005) **199**:169-182.
- [20] Walhorn, E., Hübner, B., Kölke, A. and Dinkler, D. Fluid-structure coupling within a monolithic model involving free surface flows. *Computers and Structures* (2005) **83**:2100-2111.

- [21] 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.