Fluid-structure interaction solved by a Lagrangian finite element method

Massimiliano Cremonesi[1], Attilio Frangi[1], Umberto Perego[1]
[1]*Department of Structural Engineering, Politecnico di Milano, Italy*
E-mail: cremonesi@stru.polimi.it, attilio.frangi@polimi.it, umberto.perego@polimi.it

Keywords: fluid-structure interaction, Lagrangian approach, particle finite element method

SUMMARY. In this work a Lagrangian finite element approach is developed for the solution of fluid-structure interaction problems. The fluid subproblem is solved with the Particle Finite Element Method, while the structural subproblem with a classical finite element approach. Particular attention has been paid to the identification of the interaction boundaries and to the remeshing strategy, to avoid excessive concentration or rarefaction of nodes in regions of the fluid mesh.

1 INTRODUCTION

The computational treatment of the free surface and of the interface between solid and fluid in fluid-structure interaction problems is always critical. The Arbitrary Lagrangian Eulerian method (ALE) [1] in which the movement of the fluid particles is independent of that of the mesh nodes, is often used coupled with methods to track the interfaces (e.g. volume of fluid [2] or level set [3]). A possibility to avoid the difficulties concerning the interfaces tracking is to adopt a Lagrangian approach for both fluid and structure. In the present work a fluid structure interaction algorithm is presented based on a staggered approach in which the fluid is treated in a Lagrangian framework using the Particle Finite Element Method [4] [5] and the structure using a classical finite element method.

An advantage of the Lagrangian approach for the fluid flow is that the convective terms in the momentum conservation disappear. The difficulty is however transferred to the necessity of frequent mesh regeneration due to excessive element distortion. A remedy which allows to avoid these distortions consists of systematic remeshing of the problem domain [6].

To avoid concentration and rarefaction of particles, due to the fluid motion, a remeshing strategy has been implemented. In particular, the possibility of removing particles too close to others and of adding particles in regions where the number of nodes is too low, is introudced.

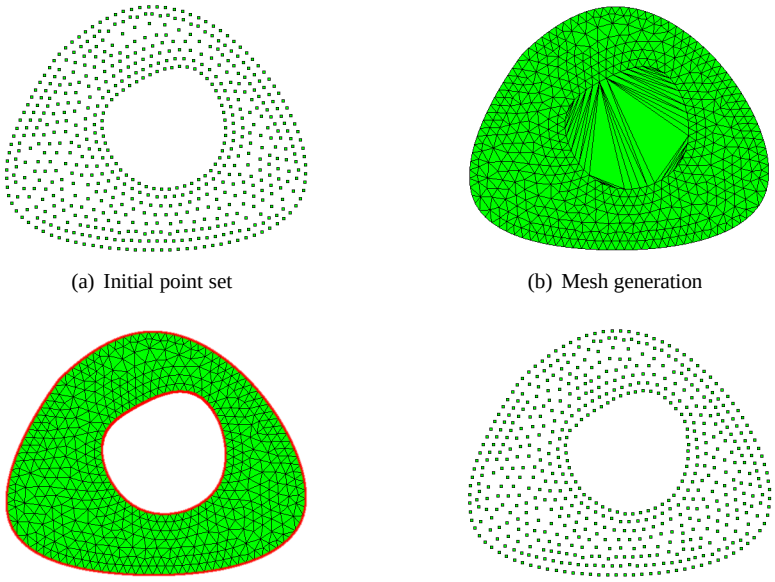
2 THE PARTICLE FINITE ELEMENT METHOD

The Particle Finite Element Method (PFEM) is a method for the solution of fluid flow problems characterized by breaking waves and free-surfaces [5]. The conceptual steps of the PFEM can be summarized as follows (see also the sketch in Figure 1):

1. discretize the fluid domain with a set of points (Figure 1(a));
2. connect the points with a Delaunay triangulation obtaining a triangular finite element mesh (Figure 1(b));
3. identify the external and internal boundaries (Figure 1(c));
4. solve the Lagrangian Navier-Stokes equations;
5. update points position using the computed velocity and pressure fields (Figure 1(d));

6. go back to step 2 and repeat for the next time step.

In the next sections the steps of this solution scheme will be explained in details.



(a) Initial point set (b) Mesh generation

(c) Boundary identification (d) Update of the particle position

Figure 1: Steps of Particle Finite Element Method

2.1 *Boundary identification*

In a Lagrangian framework the reference volume and the external boundaries are defined by the node positions. Considering that the nodes of the mesh move as a consequence of the fluid flow, every time the connectivity of the elements is regenerated, the nodes belonging to the boundary may change. Moreover with the Delaunay triangulation all the points of the set are enclosed in a convex figure (convex hull), which may not be conformal with the real external boundaries. Figure 2 clarifies this difficulty: Figure 2(a) shows a set of points and Figure 2(b) its Delaunay triangulation. It is clear that the Delaunay triangulation does not match the real external and internal boundaries. To overcome this difficulty the Delaunay triangulation can be coupled with the so called *alpha-shape method* [8]. Using a criterion based on the element distortion, the alpha-shape method allows to remove the unnecessary triangles from the Delaunay triangulation, finding the real shape of geometrical figure containing the nodes.

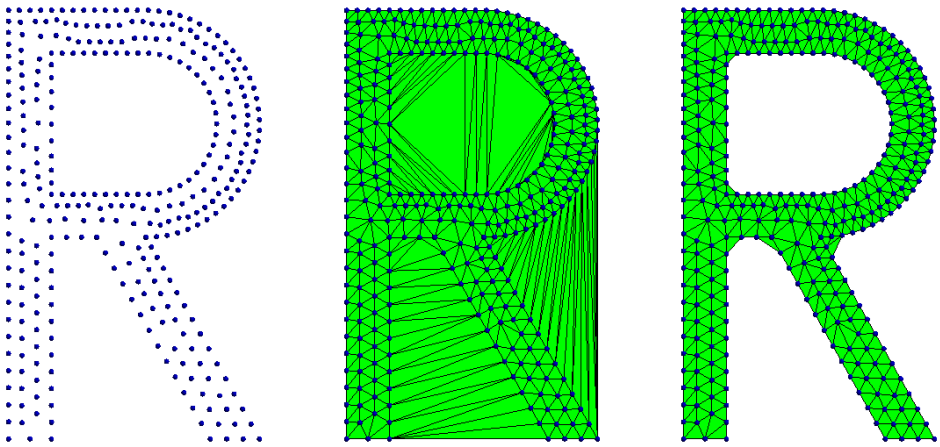
For every triangle e the minimal distance between two nodes h_e and the radius of the circumcircle R_e are introduced. The average value h of the h_e over all the elements of the mesh is also defined. The quantity:

$$\alpha_e = \frac{R_e}{h} \tag{1}$$

is an index of the element distortion and can be used to identify which triangles should be removed. In fact if

$$\alpha_e \geq \alpha \Rightarrow R_e \geq \alpha h \tag{2}$$

the triangle e is removed from the mesh. A parameter $\alpha \geq 1$ as been introduced. Increasing the value of the parameter α, less triangles are removed from the mesh, and for $\alpha \to \infty$ the Delaunay triangulation is recovered. Figure 2 shows an example of the effects of the Delaunay triangulation coupled with the alpha-shape criterion.



(a) (b) (c)

Figure 2: (a) Distrubution of points, (b) Delaunay triangulation, (c) Delaunay triangulation with α-shape.

The alpha-shape method can also be used to recover the fluid particles which separate from the rest of the domain. In fact, if a particle is connected to the domain only through excessively disorted triangles, the alpha-shape scheme removes these traingles and the particle is considered to be an isolated point. The motion of this isolated point is governed by the equation of motion of a concentrated mass subjected to its initial velocity and to the external forces. Similarly, if a single point comes close to the domain, the alpha-shape crietrion creates a triangle which incorporates the particle in the rest of the fluid domain.

2.2 *Governing equations and space discretization*

Let $\mathbf{X}$ be the position of particles in the reference configuration Ω_0 at time t, $\mathbf{U} = \mathbf{U}(\mathbf{X}, t)$ and $p = p(\mathbf{X}, t)$ the velocity and pressure fields, respectively. The motion of a Newtonian incompressible fluid is governed by the momentum conservation and the mass conservation which, in a Lagrangian framework, write:

$$\rho \frac{D\mathbf{U}}{Dt} = -\frac{1}{J}\text{Div}\left(Jp\mathbf{F}^{-T}\right) + \frac{1}{J}\mu\text{Div}\left[J\ (\text{Grad}\mathbf{U})\ \mathbf{F}^{-1}\mathbf{F}^{-T}\right] + \rho\mathbf{b} \quad \text{in } \Omega_0 \tag{3}$$

$$\text{Div}\left(J\mathbf{F}^{-1}\mathbf{U}\right) = 0 \quad \text{in } \Omega_0 \tag{4}$$

where ρ is the density, $\mathbf{F}$ is the deformation gradient, $J = \det\mathbf{F}$, $\mathbf{b}$ the body forces and μ the viscosity. In the Lagrangian approach, the non linearity is due to fact that the current configuration

differs from the original one by large displacements. This non-linearity appears in the equations through the deformation gradient $\mathbf{F}$.

To discretize in space the problem (3)-(4) an isoparametric finite element discretization is introduced. As explained in the next section, the same order of interpolation is used for both velocity and pressure unknows. The semi-discretized form of the equations (3)-(4) reads:

$$\mathbf{M}(\mathbf{x})\frac{D\mathbf{V}}{Dt} + \mathbf{K}(\mathbf{x})\mathbf{V} + \mathbf{D}^T(\mathbf{x})\mathbf{P} = \mathbf{B} \tag{5}$$

$$\mathbf{D}(\mathbf{x})\mathbf{V} = \mathbf{0} \tag{6}$$

where $\mathbf{M}$ is the mass matrix, $\mathbf{K}$ is the fluid stiffness matrix, $\mathbf{D}$ is the discretization of the divergence operator and $\mathbf{B}$ is the vector of external forces:

$$\begin{aligned}
M_{ab} &= \int_{\Omega_0} \rho_0 N_a N_b \mathbf{I} d\Omega_0 \\
D_{ab} &= \int_{\Omega_0} J N_a \text{Grad}(N_b)\mathbf{F}^{-1} d\Omega_0 \\
K_{ab} &= \int_{\Omega_0} \mu J \left(\text{Grad}(N_a)\mathbf{F}^{-1}\right) \otimes \left(\text{Grad}(N_b)\mathbf{F}^{-1}\right) d\Omega_0 \\
B_a &= \int_{\Omega_0} \rho_0 \mathbf{b} N_a \, d\,\Omega_0
\end{aligned}$$

where $\mathbf{I}$ is the identity matrix and N_a the nodal shape function. The matrix operators $\mathbf{K}$ and $\mathbf{D}$ clearly depend on the current configuration $\mathbf{x}$ through the deformation gradient $\mathbf{F}$.

2.3 *Pressure stabilization*

In the present approach every time the mesh is regenerated connecting the points with the Delauany triangulation, the element related informations are lost. The same occurs forinformations related tonodes not coinciding with triangle vertices. Consequently, to avoid convecting data from the old to the new mesh, only vertx node data can be stored, and only linear shape functions can be used in the space discretization of the problem (3)-(4). To avoid spurious oscillations due to the fact that the *inf-sup condition* is not satisfied [9], a stabilization must be introduced. In particular a *pressure-stabilizing/Petrov-Galerkin* (PSPG) stabilization is adopted [10]. Introducing a finite element space discretization as in the previous section, the semi-discrete stabilized equations read:

$$\mathbf{M}(\mathbf{x})\frac{D\mathbf{V}}{Dt} + \mathbf{K}(\mathbf{x})\mathbf{V} + \mathbf{D}^T(\mathbf{x})\mathbf{P} = \mathbf{B} \tag{7}$$

$$\mathbf{C}(\mathbf{x})\frac{D\mathbf{V}}{Dt} + \mathbf{D}(\mathbf{x})\mathbf{V} + \mathbf{L}(\mathbf{x})\mathbf{P} = \mathbf{H}(\mathbf{x}) \tag{8}$$

where the matrices $\mathbf{C}$, $\mathbf{L}$ and $\mathbf{H}$ are given by:

$$C_{ab} = \sum_{e=1}^{N_{el}} \int_{\Omega_0^e} \tau_{pspg}^e \text{Grad}(N_a) N_b d\Omega_0$$

$$L_{ab} = \sum_{e=1}^{N_{el}} \int_{\Omega_0^e} \frac{\tau_{pspg}^e}{\rho} \text{Grad}(N_a) \cdot \text{Grad}(N_b) \mathbf{F}^{-1} d\Omega_0$$

$$H_a = \sum_{e=1}^{N_{el}} \int_{\Omega_0^e} \tau_{pspg}^e \text{Grad}(N_a) \cdot \mathbf{b} d\Omega_0$$

and the stabilization parameter is defined as:

$$\tau_{pspg}^e = \frac{z_e}{2\|\mathbf{u}\|} \tag{9}$$

and z_e is the "element length" defined to be equal to the diameter of the circle which is area-equivalent to the element e.

2.4 *Time integration*

The time integration is performed with a classical backward Euler scheme. Introducing a partition of the time domain into N time steps of the same length Δt, and choosing as the reference configuration at time t^n, the discretized system reads:

$$\frac{1}{\Delta t}\mathbf{M}\left[\mathbf{U}^{n+1} - \mathbf{U}^n\right] + \mathbf{K}(\mathbf{U}^{n+1})\mathbf{U}^{n+1} + \mathbf{D}^T(\mathbf{U}^{n+1})\mathbf{P}^{n+1} = \mathbf{B} \tag{10}$$

$$\frac{1}{\Delta t}\mathbf{C}\left[\mathbf{U}^{n+1} - \mathbf{U}^n\right] + \mathbf{D}(\mathbf{U}^{n+1})\mathbf{U}^{n+1} + \mathbf{L}(\mathbf{U}^{n+1})\mathbf{P}^{n+1} = \mathbf{H} \tag{11}$$

where the matrices $\mathbf{K}$, $\mathbf{D}$ and $\mathbf{L}$ and the vector $\mathbf{H}$ depend nonlinearly on the vector $\mathbf{U}^{n+1}$.

2.5 *Adding and removing particles*

In the Lagrangian approach, the particles move as a cosequence of the fluid flow and it may happen that particles concetrate in same regions of the domanin and, on the contrary, in other regions the number of particles becomes too low to obtain an accurate solution. To overcome these difficulties, the possibility of adding and removing particles has been introduced in the proposed implementation based on two criteria: the first one to establish if a particle should be removed from a certain region and the second one to determine if new particles are necessary in another region.

The first criterion checks if the particles are too close with respect to each other. For every node of the mesh a circle, which has the node as center, is created. The radius γ of the circle is a parameter depending on the average dimensions of a mesh element. If other nodes of the mesh lay in the circle, the node on the center is removed from the mesh. Decreasing the value of γ less nodes are removed from the mesh.

The second criterion checks if in a region there are too few nodes. The area of every element of the mesh is compared with a reference value ω. If an element area is greater than ω a particle is added in the center of the triangle. Increasing the value of ω, less nodes are added to the mesh. Once a new particle is added, to solve equations (10)-(11), the value of the velocity at the previous step is required. This value is computed linearly interpolating the velocity of the nodes of the element in which the new node lays.

3 FLUID-STRUCTURE INTERACTION ALGORITHM

The fluid-structure interaction problem is solved with a staggered scheme. This algorithm consists of the independent solution of fluid and structure subproblems coupled via transmission conditions. The main advantage of this approach is that existing flow and structure solvers can be reused to perform the computations. In this work the Lagrangian Particle Finite Element Method is used to solve the fluid part and a classical finite element method is used for the solid part. The Dirichlet-Neumann algorithm is used to compute the coupling terms. A Dirichlet boundary condition (continuity of velocities) is imposed at the interface for the fluid subproblem, while a Neumann boundary condition (continuity of stresses) is imposed for the structure subproblem. The Dirichlet-Neumann algorithm iterates over these two subproblems until convergence is reached.

One of the main difficulties in the dynamic fluid-structure interaction is the determination of the current interaction surfaces. This is achieved, in the proposed formulation, by exploiting the features of the Lagrangian approach based on continuous remeshing [7]. A set of fictitious fluid particles is superposed to the nodes of the solid domain which can come in contact with the fluid domain. When the Delaunay triangulation is performed, the alpha-shape criterion selects those parts of the interface where the fluid particles are actually in contact with the structure. Once the interfaces are identified the Dirichlet-Neumann algorithm is applied to compute the coupled solution. Figure 3 shows the superposition of fictitious particles and the generation of two distinct domains (Figure (a), (b) and (c)) and of a coupled domain (figure (d) and (e)).

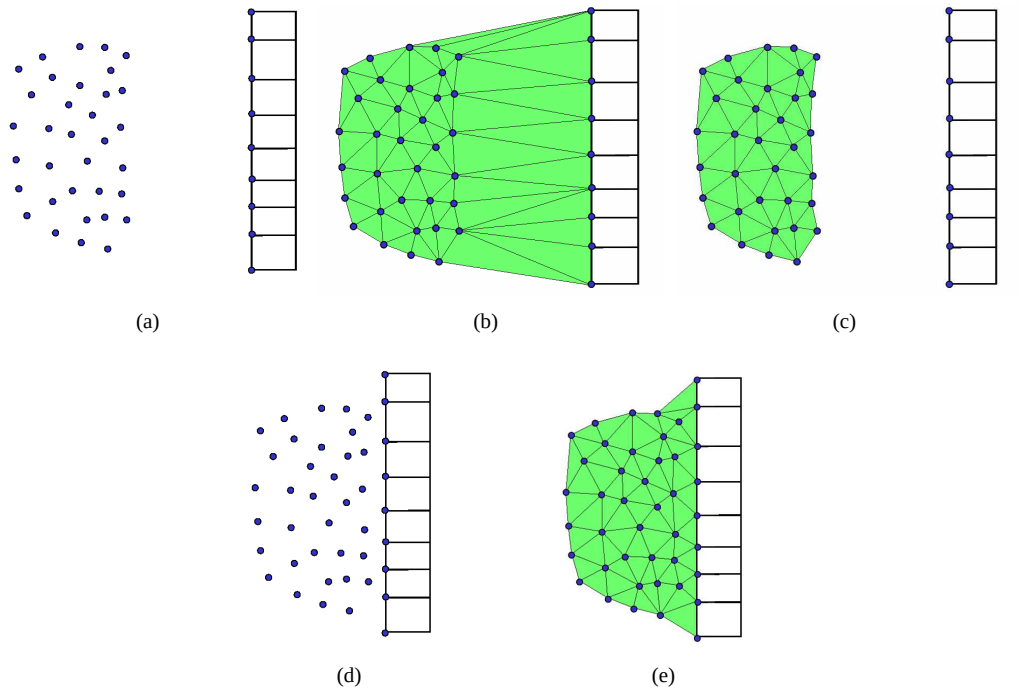


Figure 3: Sketch of fluid structure interaction: (a) superposition of the fictitious fluid particles. (b) Delaunay triangulation; (c) Delaunay triangulation with α-shape correction; (d) same as (a) but with the two domains in contact; (e) same as (c) but with the two domains in contact.

It is known that the Dirichlet-Neumann algorithm has difficulties in the convergence when the densities of the fluid of the solid are comparable [11]. In all the examples considered in the next section the density of the solid is significantly greater than the density of the fluid, avoiding convergence problem.

4 NUMERICAL EXAMPLES

4.1 *Dam collapse*

In the first example, the interaction between a fluid and a rigid container is considered. The problem consists of a column of water initially located on the left part of a tank sustained by a removable wall. At time t = 0, the removable wall is suddenly removed and, under the effect of gravity, the water flows until it collides with the right wall of the tank. Figure 4 shows same snapshots of the problem. In particular Figures 4(a), 4(c) and 4(e) show the snapshots of the analysis with the standard remeshing, while Figures 4(b), 4(d) and 4(f) show the snapshots at the same time steps using the remeshing strategy described in section 2.5. It can be clearly observed how the re-positioning of some particles according to the simple proposed scheme greatly improves the quality of the finite element mesh.

4.2 *Fluid flow through an elastic valve*

The proposed particle finite element method is particularly suited to model fluid-structure interactions involving large fluid motions with large structure displacements. To give a qualitative example of the potential of the method, the following idealized problem has been modeled.

A fluid flow hitting an elastic valve is considered. Under the action of the gravity force the fluid drops down from a funnel-shaped rigid container into another rigid container. When all the fluid particles are fallen down, the rigid bottom is suddenly removed, so that the fluid, due to its weight pushes on the elastic valve which deforms and let the fluid pass through the valve.

In this example, all the capabilities of the implemented code are required: a flow with free surface and fluid-structure interaction in which the solid undergoes large displacements are considered. Figure 5 shows snapshots of the problem configuration at different time steps, in which the dots represent the nodes of the fluid mesh (here identified as fluid particles) while the quadrilateral finite elements represent the discretization of the structure.

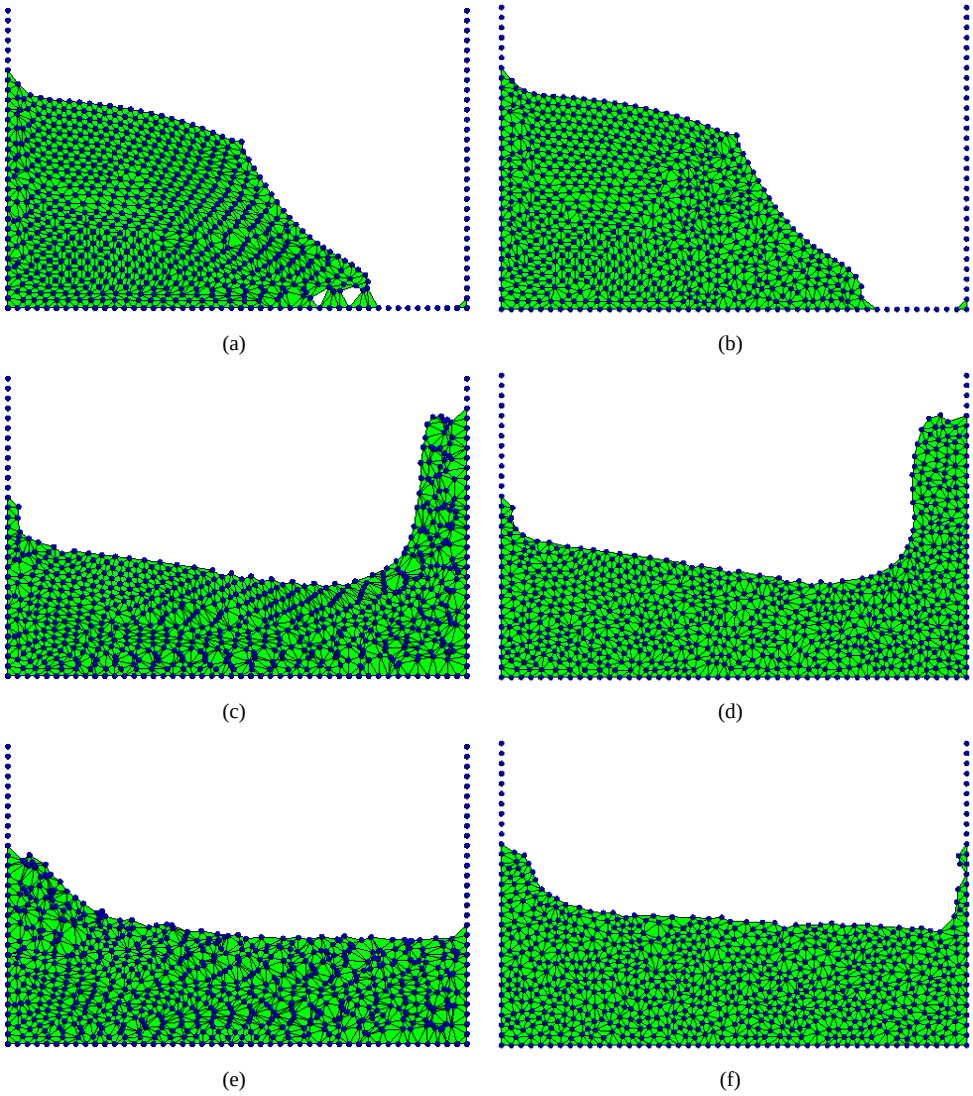


Figure 4: Dam collapse: snapshots at different time steps.

References

[1] Donea J. “Arbitrary Lagrangian-Eulerian finite element method for transient fluid structure interactions”, *Comput. Methods Appl. Mech. Engrg.* , **33** (1982) 689-723

[2] Hirt C. W., Nichols B. D., “ Volume of fluid (VOF) method for the dynamics of free boundaries”, *Journal of Computational Physics*, **39** (1981) 201-225

[3] Osher, S. and Fedkiw, R.P. “Level set methods and dynamic implicit surfaces”, *Springer-Verlag* (2003)

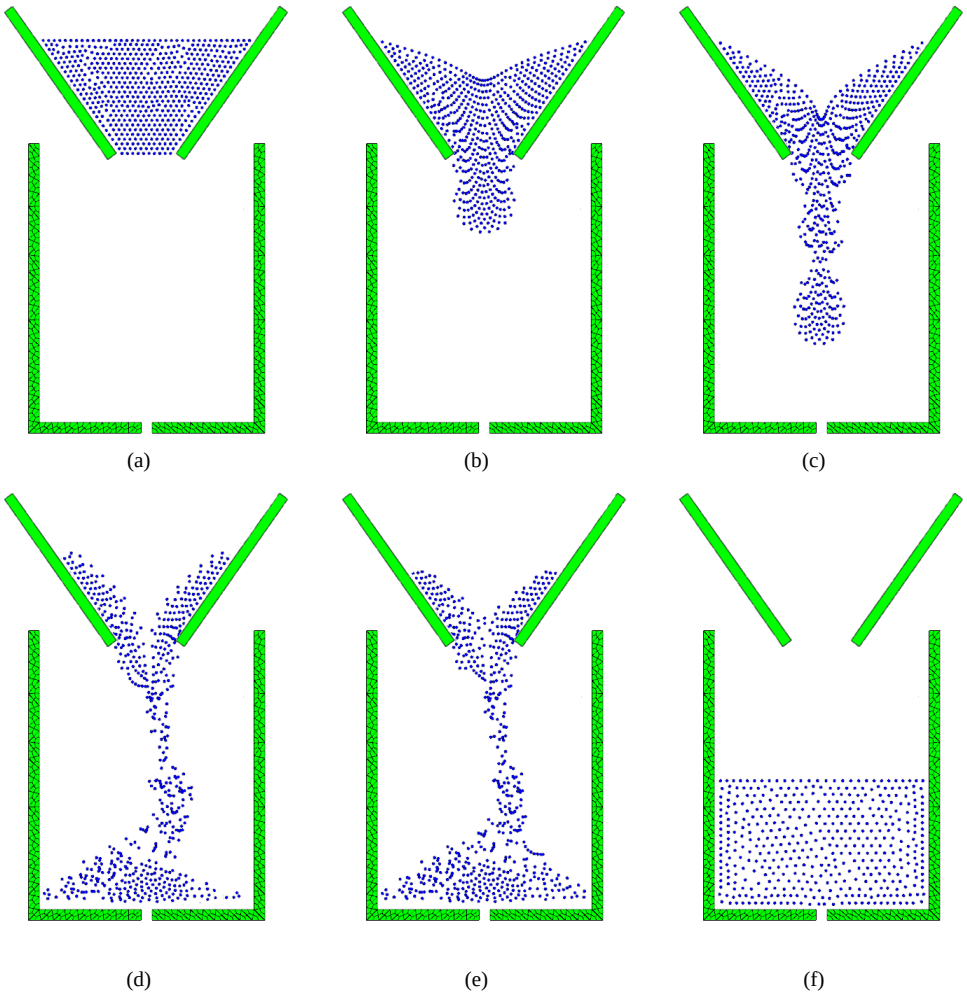


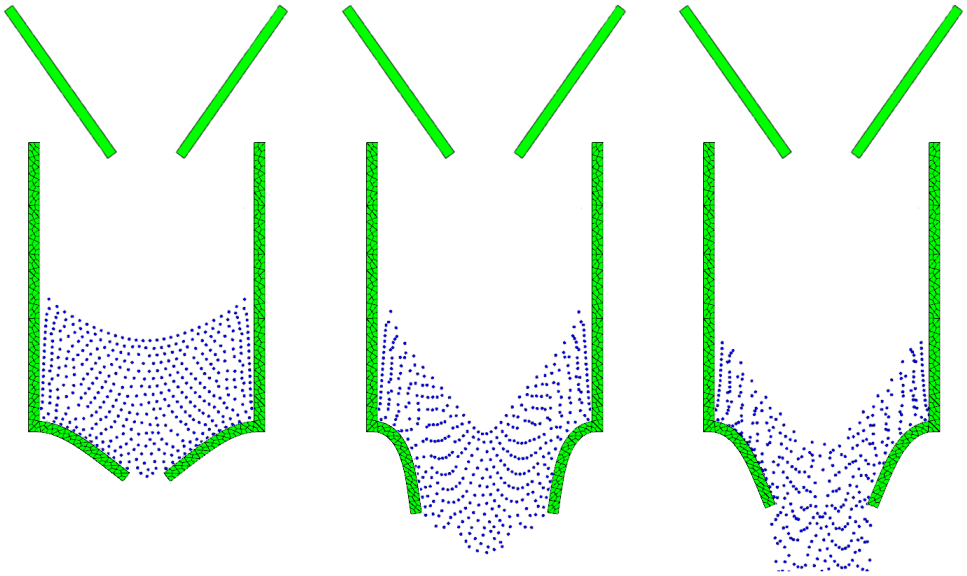
Figure 5: Fluid flow trough an elastic valve: snapshots at different time steps.

[4] Idelsohn S.R., Oñate E., 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.*, **61** (2004) 964-989

[5] Oñate E., Idelsohn S.R., Del Pin F., Aubry R., "The particle finite element method. An overview", *Int. J. Computat. Methods*, **1(2)** (2004) 267-307

[6] Radovitzky R., Ortiz M., "Lagrangian finite element analysis of Newtonian fluid flows", *Int. J. Numer. Meth. Engng.*, **43** (1998) 607-619

[7] Cremonesi M., Frangi A., Perego U, "A Lagrangian Finite Element Approach for the analysis of fluid-structure interaction problems", to appear



(a) (b) (c)

Figure 6: Fluid flow trough an elastic valve: snapshots at different time steps.

[8] Edelsbrunner H., Mucke E. P., "Three dimensional alpha shapes", *ACM Transactions on Graphics* , **13** (1994) 43-72

[9] Brezzi F., Fortin M., "Mixed and Hybrid Finite Element Method", *Springer-Verlag* (1991)

[10] Tezduyar T. E., Mittal S., Ray S. E., Shih R., "Incompressible flow computations with stabilized bilinear and linear equal-order-interpolation velocity-pressure elements", *Comput. Methods Appl. Mech. Engrg.*, **95(2)** (1992) 221-242

[11] Causin P., Gerbeau J.F., Nobile, F. "Added-mass effect in the design of partitioned algorithms for fluid-structure problems", *Comput. Methods Appl. Mech. Engrg.* , **194** (2005)

[12] Aubry R., Idelsohn S.R., Oate E., "Particle finite element method in fluid-mechanics including thermal convection-diffusion", *Computers & Structures* , **83** (2005) 1459-147

[13] Larese A., Rossi R., Oñate E., Idelsohn S.R., "Validation of the particle finite element method (PFEM) for simulation of free surface flows", *Engineering Computations*, **25** (2008) 385-425

[14] Oñate E., Idelsohn S. R., Celigueta M. A., Rossi, R., "Advances in the particle finite element method for the analysis of fluid-multibody interaction and bed erosion in free surface flows", *Comput. Methods Appl. Mech. Engrg.* , **197** (2008) 1777–1800