



**HAL**  
open science

# Stabilized Finite Element Methods vs LES modelling for fluid-structure interaction with anisotropic adaptive meshing

Elie Hachem, Stephanie Feghali, Thierry Coupez

► **To cite this version:**

Elie Hachem, Stephanie Feghali, Thierry Coupez. Stabilized Finite Element Methods vs LES modelling for fluid-structure interaction with anisotropic adaptive meshing. 10e colloque national en calcul des structures, May 2011, Giens, France. 8p. ; Clé USB. hal-00592696

**HAL Id: hal-00592696**

**<https://hal.science/hal-00592696>**

Submitted on 3 May 2011

**HAL** is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire **HAL**, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.

# Stabilized Finite Element Methods vs LES modelling for fluid-structure interaction with anisotropic adaptive meshing

E. Hachem<sup>1</sup>, S. Feghali<sup>2</sup>, T. Coupez<sup>3</sup>

*Center for Material Forming (CEMEF), MINES-ParisTech, UMR CNRS 7635, Sophia-Antipolis, France, <sup>1</sup>Elie.Hachem@mines-paristech.fr;  
<sup>2</sup>Stephanie.El\_Feghali@mines-paristech.fr <sup>3</sup>Thierry.Coupez@mines-paristech.fr*

---

**Résumé** — This paper presents a stabilised finite element method for the solution of incompressible multiphase flow problems in three dimensions using an immersed volume method with anisotropic adaptive meshing. A recently developed stabilised finite element solver which draws upon features of solving general fluid-structure interactions is presented. The proposed method is developed in the context of the monolithic formulation. Such strategy gives rise to an extra stress tensor in the Navier-Stokes equations coming from the presence of the structure in the fluid. The distinctive feature of the Variational MultiScale approach is not only the decomposition for both the velocity and the pressure fields into coarse/resolved scales and fine/unresolved scales but also the possible efficient enrichment of the extra constraint. This choice of decomposition is shown to be favorable for simulating multiphase flows at high Reynolds number. We assess the behaviour and accuracy of the proposed formulation coupled to the levelset method approximation in the simulation of 2D and 3D time-dependent numerical examples such as : vortex shedding behind an obstacle, conjugate heat transfer inside industrial furnaces and the rigid bodies motion in incompressible flows.

**Mots clés** — Immersed Stress Method, Stabilized Finite Elements, Fluid-Structure Interaction, Anisotropic Mesh Adaptation, Monolithic Approach.

---

## 1 Introduction

Fluid Structure Interaction (FSI) is of great relevance in many fields of engineering as well as in the applied sciences and material forming with applications ranging from bioengineering to aerodynamics and from civil engineering to automotive. Often, when interaction effects are essential this comes along with large/small structural deformations and/or with turbulent flows. However, many available approaches may lack robustness especially in such severe situations. The components in all engineering fields are continuously pushed towards higher performance by seeking new developments that must be able to deal with different situations and regimes. In particular it must be able to treat encountered problems ranging from the mesh adaptation issues to the coupling engines between different codes, and from small/large deformations to low/high Reynolds numbers flows.

Most of the commercial software packages solve FSI problems using an Arbitrary Lagrangian Eulerian (ALE) formulation [1, 2, 3]. The solid domain is treated with a Lagrangian formulation. The nodes belonging to the interface between the solid and the fluid are moved with the solid. The displacement of the nodes in the fluid domain do not depend on the fluid motion, but only ensures the continuity between the fluid and the solid domain, and a good mesh quality. ALE methods are robust and accurate, and do not need any extra degrees of freedom. However, important problems arise if the deformations, displacements and rotations of the solid becomes very important [4, 5, 6].

A higher popularity has been gained recently by partitioned approaches which allow the use specific solver for each domain. The difficulty remains in transferring the information between the codes. The coupling between the two phases can be enforced using different schemes : weakly or strongly coupled version. The former approach manages with just one solution of either field per time step but consequently lack accurate fulfilment of the coupling conditions. The latter requires sub-iterations. The predominant approach consists in solving the problem iteratively, using fixed-point schemes [7] or Newton Krylov methods [8, 9, 10, 11]. Actually, the fixed-point methods with dynamic relaxation seem to be the most interesting variant [12]. This approach allows the use of fluid and solid solvers for each of the

two phases. It is accurate and quite efficient but present an inherent instability depending on the ratio of the densities and the geometry of the domain [13]. As a result, the numerical cost increase drastically and coupling algorithms may not converge. For 3D problems, such difficulties become even more severe.

Monolithic approaches have been proposed to overcome these drawbacks. The whole domain (composed by fluid and solid phases) is considered as a single one, meshed by a single grid, and solved with an Eulerian framework. The continuity at the interface is then obtained naturally and there is no need to enforce it, as it was the case in partitioned methods. If the multi-mesh approaches permit the use of classical fluid and solid solvers, monolithic approaches impose the use of an appropriated unique constitutive equation describing both the fluid and the solid domain. Interface tracking, between the two different domains, can be completed by Immersed Boundary (IB) methods [14] where the interface is convected on a Lagrangian way. Other methods such as the fictitious domain [14, 15] treat the coupling between the domains by applying a constraints across the rigid body using a Lagrange multiplier.

Here in this work, a new monolithic method is developed : the Immersed Stress Method. This method can be seen as an extension of the Immersed Volume Method (IVM) [16] to treat real fluid-structure interactions. The motivation of pursuing such general approach comes from the desire of not solving two equations, e.g. one in the solid and another in the fluid, where in some cases ; one may still need to provide the boundary conditions between the two domains. Recall also that the complexity to ensure such conditions is amplified when simulating turbulent fluid structure interactions. When dealing with a large diversity of shapes, dimensions and physical properties of structures, such simulations become rapidly very costly, time consuming and limited. A complete description and details about the immersed volume method but used for a different context (developed previously and applied to thermal couplings) is given in [17, 16].

Therefore, we retain the use of a monolithic formulation for fluid/solid and coupling it to some additional features for accurate resolution, in particular at the interface. The monolithic approach in here is made of a unique mesh in which the different domains are taking place by the level set function. Consequently, different structures are immersed in a larger domain of different material properties so that boundary conditions at the interface can be replaced naturally.

The second important ingredient of the approach is the use of anisotropic mesh adaptation [18, 19, 20, 16] at the interface between two different materials. The idea is to apply a fast mesh generation algorithm that allows the creation of meshes with extremely anisotropic elements stretched along the interface, which is an important requirement for FSI problems having internal/boundary layers. It is successfully applied for fixed and some moving objects [17, 16].

The last and most important ingredient focuses more on the finite element solver : on modeling the interaction between the fluid (laminar or turbulent) and the structure in question (rigid, elastic, viscoelastic, etc). For FSI simulations of elastic/rigid body immersed for instance in an incompressible fluid, the global behavior is described by the classical Navier-Stokes equations, with an extra stress tensor [21]. For instance, we simulate a rigid solid using the Navier-Stokes solver under constraints to impose the nullity of the deformations. This can be done by simply penalizing the strain rate using a very important viscosity in the solid, which can sometimes be sufficient [22, 17]. It is also possible to enforce directly the nullity of the strain by using an Augmented Lagrange Multipliers method [23, 24, 25], solved by an iterative Uzawa algorithm. The problem is then solved by adding an extra-stress tensor coming from the presence of the structure in the fluid. Linear or harmonic mixture laws of the mechanical properties characterizing each domain are then applied at the interface.

## 2 Finite Element Formulation

### 2.1 Governing equations

The governing equations are considered to be three-dimensional and unsteady. Using a monolithic approach, a unique constitutive equation will be solved on the whole domain, with a variation of the parameters depending on the phase that should be modelled. Recall that the the concept of the Immersed Stress Method (ISM) is based on solving the single set of equations by differentiating the subdomains and refining the mesh at this interface using the level set method. The ISM allows the immersion of any structure using the level-set function, mixes the physical properties ( $\rho$  and  $\eta$ ) and finally applies the

anisotropic mesh adaptation at the vicinity of the interface.

The strong form for the whole domain reads then :

$$\left\{ \begin{array}{l} \rho(\partial_t \mathbf{v} + \mathbf{v} \cdot \nabla \mathbf{v}) - \nabla \cdot (2\eta_f \boldsymbol{\varepsilon}(\mathbf{v}) + \boldsymbol{\tau}_s - p \mathbf{I}_d) = 0 \\ \nabla \cdot \mathbf{v} = 0 \\ \boldsymbol{\varepsilon}(\mathbf{u}) - \frac{3}{2E} \boldsymbol{\tau}_s = 0 \\ \partial_t \mathbf{u} + \mathbf{v} \cdot \nabla \mathbf{u} = \mathbf{v} \\ + \text{Boundary conditions} \end{array} \right. \quad (1)$$

where  $\boldsymbol{\tau}_s$  is the extra stress tensor reflecting the presence of the immersed structure (elastic/rigid...) in the incompressible fluid and  $E$  is the Young modulus. Note that depending on the value of  $E$ , the treated structure will inherit the appropriate law (rigid, elastic, ...).

## 2.2 Stabilized finite element method

Based on a mesh  $\mathcal{K}_b$  of  $\Omega$  into set of  $N_{el}$  elements  $K$ , the functional spaces for the velocity, the pressure and the stress are approached by the finite dimensional spaces spanned by  $V_h$ ,  $P_h$  and  $\mathcal{T}_h$ . The variational multiscale method is used to stabilize the Galerkin formulation and allows the use of equal order continuous interpolations for the velocity and the pressure unknowns (see [26] for details). A piecewise constant interpolation for stresses can be used. It consists in here of a decomposition for both the velocity and the pressure fields into coarse/resolved scales and fine/unresolved scales. The distinctive feature of the proposed approach resides in the efficient enrichment of the extra constraint. We first solve and then we substitute the fine-scale solution into the large-scale problem providing additional terms, tuned by a local time-dependent stabilizing parameter, that enhance the stability and accuracy of the standard Galerkin formulation for the transient Navier-Stokes equations. Such approach can deal with laminar/turbulent FSI problems and can handle large/small deformations. Additionally, when higher accuracy is needed, by applying a robust and fast mesh adaptation it provides a much more computational efficiency than coupling solvers.

The enrichment of the functional spaces for the velocity, pressure and stress solutions is performed as follows :  $\mathcal{V}_h \oplus \mathcal{V}'$ ,  $P_h \oplus P'$  and  $\mathcal{T}_h \oplus \mathcal{T}'$ . To this end,  $\mathbf{v}$ ,  $p$ ,  $\boldsymbol{\tau}$  will be approximated as :

$$\mathbf{v} = \mathbf{v}_h + \mathbf{v}' \in \mathcal{V}_h \oplus \mathcal{V}', \quad \mathbf{p} = \mathbf{p}_h + \mathbf{p}' \in \mathcal{P}_h \oplus \mathcal{P}', \quad \boldsymbol{\tau} = \boldsymbol{\tau}_h + \boldsymbol{\tau}' \in \mathcal{T}_h \oplus \mathcal{T}' \quad (2)$$

### 2.2.1 Numerical scheme

Three equation with three primary variables requires larger computational cost. To circumvent this issue an augmented Lagrangian method and Uzawa's algorithm would be used to solve the system without increasing the size of the linear system. In the same iteration, the problem of non linearity, the time integration and the computation of the Lagrange multiplier would be solved. An implicit time scheme with a Newton method for the non-linear term is used.

## 3 LevelSet method

The interface between the phases is resolved using a convected level set approach developed in [27]. This approach enables first to restrict convection resolution to the neighbourhood of the interface and second to replace the reinitialisation steps by an advective reinitialisation. This enables an efficient resolution and accurate computations of flows even with large density and viscosity differences. The level set function is discretized using a stabilized upwind Petrov-Galerkin method and can be coupled to a direct anisotropic mesh adaptation process enhancing the interface representation.

### 3.1 Anisotropic mesh adaptation

Accurate calculation of the velocities, strains and stresses along the fluid-solid interface is critical for a correct modelling of industrial applications. The difficulty arises due to the discontinuity of the

properties of the material across the interface. If this latter is not aligned with the element edges, it may intersect the element arbitrarily such that the accuracy of the finite element approach can be compromised. In order to circumvent this issue, the level-set process is thus coupled to an anisotropic mesh adaptation as described in [18]. The idea of this method is to pre-adapt the mesh at the interface. The mesh becomes locally refined, elements are stretched, which enables to sharply define the interface and to save a great number of elements compared to classical isotropic refinement. This anisotropic adaptation is performed by constructing a metric map that allows the mesh size to be imposed in the direction of the distance function gradient. Let us briefly describe the main principles of this technique. First of all, one has to resort to a so-called metric which is a symmetric positive defined tensor representing a local base that modify the distance computation, such that :

$$\|\mathbf{x}\|_{\mathbb{M}} = \sqrt{\mathbf{x}^T \cdot \mathbb{M} \cdot \mathbf{x}}, \quad \langle \mathbf{x}, \mathbf{y} \rangle_{\mathbb{M}} = \mathbf{x}^T \cdot \mathbb{M} \cdot \mathbf{y}. \quad (3)$$

The metric  $\mathbb{M}$  can be regarded as a tensor whose eigenvalues are related to the mesh sizes, and whose eigenvectors define the directions for which these sizes are applied. For instance, using the identity tensor, one recovers the usual distances and directions of the Euclidean space. In our case the direction of mesh refinement is given by the unit normal to the interface which corresponds to the gradient of the level-set function :  $\mathbf{x} = \nabla\alpha/|\nabla\alpha|$ . A default mesh size, or background mesh size,  $h_d$  is imposed far from the interface and it is reduced as the interface comes closer. A likely choice for the mesh size evolution is the following :

$$h = \begin{cases} h_d & \text{if } |\alpha(\mathbf{x})| > e/2 \\ \frac{2h_d(m-1)}{m e} |\alpha(\mathbf{x})| + \frac{h_d}{m} & \text{if } |\alpha(\mathbf{x})| \leq e/2 \end{cases} \quad (4)$$

Eventually, at the interface, the mesh size is reduced by a factor  $m$  with respect to the default value  $h_d$ . Then this size increases until equalling  $h_d$  for a distance that corresponds to the half of a given thickness  $e$ . The unit normal to the interface  $\mathbf{x}$  and the mesh size  $h$  defined above, lead to the following metric :

$$\mathbb{M} = C(\mathbf{x} \otimes \mathbf{x}) + \frac{1}{h_d} \mathbb{I} \quad \text{with} \quad C = \begin{cases} 0 & \text{if } |\alpha(\mathbf{x})| \geq e/2 \\ \frac{1}{h^2} - \frac{1}{h_d^2} & \text{if } |\alpha(\mathbf{x})| < e/2 \end{cases} \quad (5)$$

where  $\mathbb{I}$  is the identity tensor. This metric returns to isotropic far from the interface (with a mesh size equal to  $h_d$  for all directions) and to anisotropic near the interface (with a mesh size equal to  $h_i$  in the direction  $\mathbf{x}$  and equal to  $h_d$  in the others). This method can be assisted by a posteriori anisotropic error estimator, the search of the optimal mesh (metric) that minimizes the error estimator. As a result, an optimal metric as a minimum of an error indicator function and for a given number of elements is obtained. In practice, the mesh is generated in several steps using the MTC mesher and remesher developed by [19]. The proposed mesh generation algorithm works well for 2D or 3D complex shapes. It allows the creation of meshes with extremely anisotropic elements stretched along the interface. The mesh size is then only refined in the direction of the high physical and mechanical properties gradients. This allow both conserving a high precision in the calculus and in the geometry description, in spite of an important decrease of the total number of degrees of freedom. The grid is furthermore only modified in the vicinity of the interface which keeps the computational work devoted to the grid generation low. Note also that the proposed method can easily handle arbitrary complex geometries. As shown in figure 1 which presents a close-up on the interface zone at the end of the anisotropic adaptation process, the mesh has been gradually refined when approaching the interface. Consequently, only additional nodes are locally added in this region, whereas the rest of domain keeps the same background size.

Another method to construct metrics, based on an anisotropic a posteriori error estimator, can also be used with CimLib. This error estimator uses the eigenvalues and eigenvectors of the recovered Hessian matrix for a given function and a fixed number of nodes to construct the metric field [28, 18]. Figure 1 presents a dynamic anisotropic mesh adaptation behind a solid obstacle. As shown, the proposed mesh adaptation approach is capable of handling at the same time a fine mesh around the solid body and the inside the vortices.

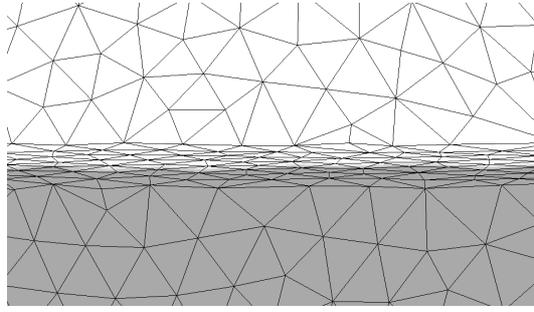


FIGURE 1 – Zoom on the fluid-solid interface after anisotropic adaptation

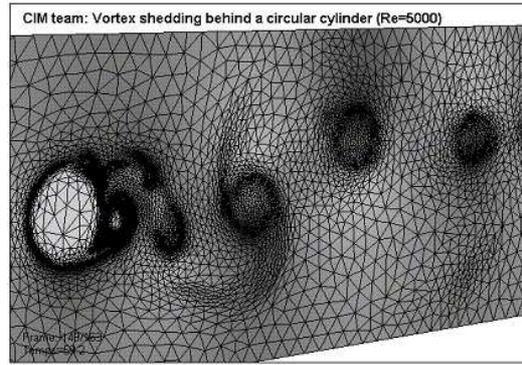


FIGURE 2 – Dynamic anisotropic mesh adaptation for vortex shedding behind a rigid obstacle

## 4 Numerical simulations

In this section, we present relatively simple 2D and 3D test cases in order to validate the proposed formulation and to check the accuracy and the efficiency of the immersed stress method. All the numerical simulations were carried out by using the C++ CimLib finite element library .

In figure 3, four solids objects with different densities are falling due to gravity in an air-filled channel. The objective of this test, referred as the Tetris benchmark, is to show the capability of the dynamic mesh adaptation as well as the method to handle high discontinuities of the solids and fluid physical properties. As shown in figure 4, the convection dominated flows of the surrounding air, the four rigid solids movement as well as the representation of the objects are all well taken into account using one a single domain with one set of equations.

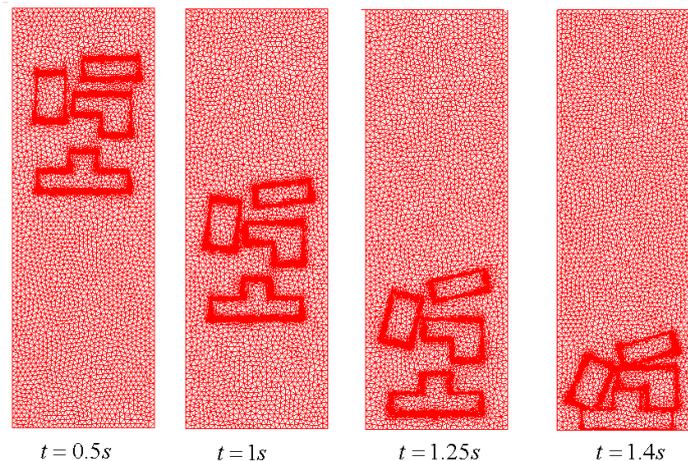


FIGURE 3 – The Immersed Stress Method with dynamic anisotropic mesh adaptation

Figure 5 presents the parallel numerical simulation of turbulent heat transfer inside an industrial furnace using the proposed monolithic fluid-structure approach with fixed anisotropic mesh adaptation. The

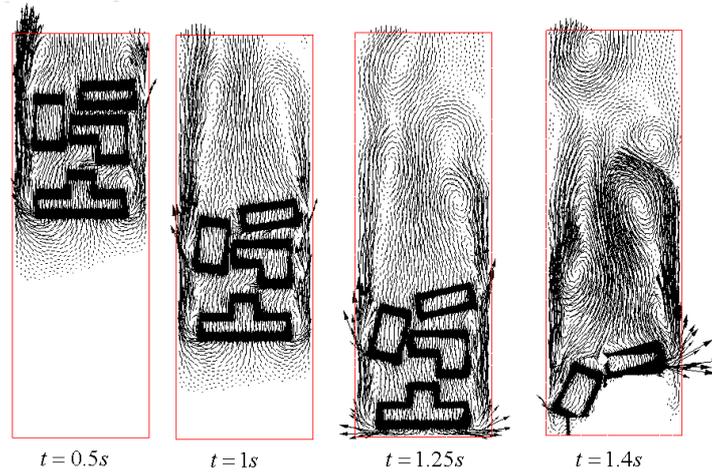


FIGURE 4 – Immersion of four rigid bodies : velocity vectors at different time instants

mesh generation algorithm allows the creation of meshes with extremely anisotropic elements stretched along the interface, which is an important requirement for FSI problems having internal/boundary layers. The final obtained mesh reflects the capability of the method to render a well respected geometry in terms of curvature, angles and complexity. Contrary to others techniques, this promising method can provide an alternative to body-fitted mesh for very complex geometry.

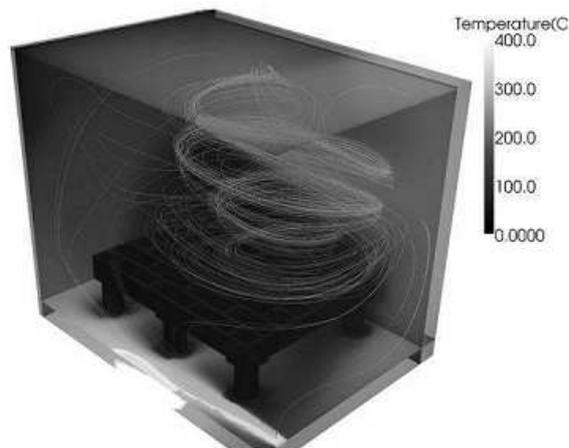


FIGURE 5 – Turbulent heat transfer inside an industrial furnace

## 5 Conclusion

In this paper we have presented a stabilized three-field velocity-pressure-stress, referred as Immersed Stress Method, designed for the computation of rigid bodies in an incompressible Navier-Stokes flow. The presence of the solid is taken into account as an extra stress in the Navier-Stokes equation. The sharp discontinuity of the material properties was captured by an anisotropic refined solid-fluid interface. The robustness of the method to compute the flow and heat transfer with large materials properties differences is demonstrated using stabilized finite element formulations. The approach is applied to the numerical simulation of 2D and 3D test cases. Finally, the capability of the model to simulate the fluid-rigid body interaction was demonstrated.

## Références

- [1] C.W. Hirt, A. A. Amsden, J. L. Cook, *An Arbitrary Lagrangian-Eulerian computing method for all speeds*, Journal of Computational Physics, **14**, 227-253, 1974.
- [2] Hughes, T. J. R. and Liu, W. K. and Zimmerman, T., *Lagrangian-Eulerian finite element formulation for incompressible viscous flow*, **29**, 329-349, 1981.
- [3] J. Donea, S. Giuliani, J. P. Halleux, *An Arbitrary Lagrangian-Eulerian finite element method for transient dynamic fluid-structure interactions*, Computer Methods in Applied Mechanics and Engineering, **33**, 689-723, 1982.
- [4] D. Benson, *An efficient, accurate, simple ALE method for nonlinear finite element programs*, Computer Methods in Applied Mechanics and Engineering, **72**, 305-350, 1989.
- [5] M. Souli, A. Ouahsine, L. Lewin, *ALE formulation for fluid structure interaction problems*, Computer Methods in Applied Mechanics and Engineering, **190**, 659-675, 2000.
- [6] R. van Loon, P.D. Anderson, F. N. van de Vosse, S.J. Sherwin, *Comparison of Various fluid-structure interaction methods for deformable bodies*, Computers and Structures, **85**, 833-843, 2007.
- [7] P. Le Tallec, J. Mouro, *Fluid structure interaction with large structural displacements*, Computer Methods in Applied Mechanics and Engineering, **190**, 3039-3067, 2001.
- [8] M. A. Fernández, M. Moubachir, *A Newton method using exact jacobians for solving fluid-structure coupling*, Computers and Structure, **83**, 127-142, 2005.
- [9] J-F. Gerbeau, M. Vidrascu, *A quasi-Newton algorithm based on a reduced model for fluid structure interaction problems in blood flow*, Mathematical Modelling and Numerical Analysis, **37**, 631-647, 2003.
- [10] C. Michler, E.H. van Brummelen, R. de Borst, *An interface NewtonKrylov solver for fluidstructure interaction*, International Journal for Numerical Methods in Fluids, **47**, 1189-1195, 2005.
- [11] J-F. Gerbeau, M. Vidrascu, P. Frey, *Fluid structure interaction in blood flows on geometries coming from medical imaging*, Computers and Structure, **83**, 155-165, 2005.
- [12] U. Küttler, A. W. Wall, *Fixed-point fluidstructure interaction solvers with dynamic relaxation*, Computational Mechanics, **43**, 61-72, 2008.
- [13] P. Caussin, J.-F. Gerbeau, F. Nobile, *Added-mass effect in the design of partitioned algorithms for fluid-structure problems*, Computer Methods in Applied Mechanics and Engineering, **197**, 4506-4527, 2005.
- [14] C. S. Peskin, *The immersed boundary method*, Acta Numerica, **11**, 479-517, 2002.
- [15] R. Glowinski, T. W. Pan, T.-I. Helsen, D.D. Joseph, *A distributed lagrange multiplier/fictitious domain methods for particulate flows*, International Journal of Multiphase flow, **25**, 755-794, 1999.
- [16] E. Hachem, T. Kloczko, H. Dignonnet, T. Coupez, *Stabilized finite element solution to handle complex heat and fluid flows in industrial furnace using the immersed volume method*, International Journal for Numerical Methods in Fluids, online, <http://dx.doi.org/10.1002/fld.2498>, 2010.
- [17] E. Hachem, *Stabilized Finite Element Method for Heat Transfer and Turbulent Flows inside Industrial Furnaces*, Ecole Nationale Supérieure des Mines de Paris, 2009.
- [18] T. Coupez, *Metric construction by length distribution tensor and edge based error for anisotropic adaptive meshing*, submitted to Journal of Computational Physics, DOI :10.1016/j.jcp.2010.11.041., 2010.
- [19] T. Coupez, *Génération de maillage et adaptation de maillage par optimisation locale*, Revue européenne des éléments finis, **9**, 403-423, 2000.
- [20] C. Gruau, T. Coupez, *3D tetrahedral, unstructured and anisotropic mesh generation with adaptation to natural and multidomain metric*, Computer Methods in Applied Mechanics and Engineering, **194**, 4951-4976, 2005.
- [21] S. Feghali, E. Hachem and T. Coupez, *Monolithic stabilized finite element method for rigid body motions in the incompressible Navier-Stokes flow*, submitted to Er. Journ. of Comp. Mech., **19**, 547-573, 2010.
- [22] J. B. Ritz, J. P. Caltagirone, *A Numerical Continuous Model for the Hydrodynamics of Fluid Particle Systems*, International Journal for Numerical Methods in Fluids, **30**, 1067-1090, 1999.
- [23] R. Glowinski, T.W. Pan, T. I. Helsen, D.D. Joseph, J. Periaux, *A fictitious domain approach to the direct numerical simulation of incompressible viscous flow past moving rigid bodies : application to particulate flow*, Journal of Computational Physics, **169**, 363-426, 2001.
- [24] M. Coquerelle, G.-H. Cottet, *A vortex level set method for the two-way coupling of an incompressible fluid with colliding rigid bodies*, Journal of Computational Physics, **227**, 9121-9137, 2008.
- [25] J. Janela, A. Lefebvre, B. Maury, *A penalty method for the simulation of fluid-rigid body interaction*, ESAIM Proceedings, **14**, 115-123, 2005.

- [26] E. Hachem, B. Rivaux, T. Kloczko, H. Dignonnet, T. Coupez, *Stabilized finite element method for incompressible flows with high Reynolds number*, Journal of Computational Physics, **229**, 8643-8665, 2010.
- [27] L. Ville, L. Silva, T. Coupez, *Convected level set method for the numerical simulation of fluid buckling*, online in Int. J. Numer. Meth. Fluids, <http://dx.doi.org/10.1002/fld.2259>, 2010.
- [28] Y. Mesri, H. Dignonnet, T. Coupez, *Advanced parallel computing in material forming with CIMLIB*, European Journal of Computational Mechanics, **18**, 669-694, 2009.