Available online at www.sciencedirect.com
SCIENCE DIRECT®
C. R. Mecanique 331 (2003) 525–530
COMPTES RENDUS MECANIQUE

Modified fixed point algorithm in fluid–structure interaction

Simone Deparis [a,*], Miguel Ángel Fernández [a], Luca Formaggia [b], Fabio Nobile [c]

[a] *École polytechnique fédérale de Lausanne, IACS, 1015 Lausanne, Switzerland*
[b] *Politecnico di Milano, MOX, Piazza L. da Vinci 32, 20133 Milano, Italy*
[c] *CES, The University of Texas at Austin, Austin, TX 78712, USA*

Received 6 November 2002; accepted 27 May 2003

Presented by Olivier Pironneau

Abstract

In this work, we address the numerical solution of some non-linear problems arising in the time discretization of fluid–structure interaction problems with fully implicit schemes. At each time step, we have to solve a highly non-linear coupled system, since the fluid domain depends on the unknown displacement of the structure. We propose a modified fixed-point algorithm which combines the Block-Gauss–Seidel iterations with a transpiration formulation. Numerical experiments show the great improvement in computing time with respect to the standard method. ***To cite this article: S. Deparis et al., C. R. Mecanique 331 (2003).***

Résumé

Accélération avec des conditions de transpiration d'un algorithme de point fixe en interaction fluide–structure. Dans cette Note on s'intéresse à la résolution numérique des problèmes non-linéaires issus de la discrétisation en temps, avec des schémas implicites, de problèmes d'interaction fluide–structure. A chaque pas de temps, on doit résoudre un système couplé fortement non-linéaire, car le domaine fluide dépend de l'inconnue associée au déplacement de la structure. Nous proposons un algorithme de point fixe modifié qui combine des itérations Block-Gauss–Seidel avec une formulation de type transpiration. Des expériences numériques mettent en évidence le gain en temps de calcul par rapport à la méthode standard. ***Pour citer cet article : S. Deparis et al., C. R. Mecanique 331 (2003).***

Keywords: Computational fluid mechanics; Fluid–structure interaction; Block-Gauss–Seidel iterations; Transpiration; Highly coupled non-linear problems; Weak and strong coupling algorithms; Partitioned procedures

Mots-clés : Mécanique des fluides numérique ; Interaction fluide–structure ; Itérations de Gauss–Seidel ; Transpiration ; Problémes couplès hautements non-linéaires ; Algorithmes de couplage faible et fort ; Méthode à partion

* Corresponding author.
E-mail addresses: simone.deparis@epfl.ch (S. Deparis), miguel.fernandez@epfl.ch (M.Á. Fernández), luca.formaggia@mate.polimi.it (L. Formaggia), nobile@ices.utexas.edu (F. Nobile).

1631-0721/$ – see front matter
doi:10.1016/S1631-0721(03)00119-0

Version française abrégée

Dans ce travail on s'intéresse à la résolution numérique des problèmes non-linéaires issus de la discrétisation en temps, avec des schémas implicites, de problèmes d'interaction fluide–structure. Des expériences numériques montrent que ces schémas sont les seuls qui assurent stabilité quand la densité du fluide est comparable à celle de la structure (voir [8]). Ainsi, à chaque pas de temps, on doit résoudre un système couplé fortement non-linéaire. Nous proposons un algorithme de point fixe modifié qui combine des itérations Block-Gauss–Seidel (BGS) (consulter [2]) avec une formulation en transpiration (voir [4,5,7]). Des expériences numériques mettent en évidence le gain en temps de calcul par rapport à la méthode BGS standard.

Le système sous étude occupe, à l'instant $t \geqslant 0$, un domaine mobile $\Omega(t) \subset \mathbb{R}^3$. Ce domaine comporte une structure déformable $\Omega^{\mathrm{s}}(t)$ (paroi artérielle, pipe-line,...) et un fluide en mouvement (sang, pétrole,...) sur le complémentaire $\Omega^{\mathrm{f}}(t)$ de $\Omega^{\mathrm{s}}(t)$ dans $\Omega(t)$. Le fluide est supposé newtonien et incompressible, son état est décrit par sa vitesse et pression (u, p). L'évolution du solide (en grands déplacements) est décrite par son mouvement et son tenseur de contraintes (x^{s}, S). L'évolution du domaine $\Omega(t)$ est représentée par une carte ALE (Arbitrairement Lagrangienne–Eulérienne) x, qui nous permet d'écrire les équations fluide en formalisme ALE. Ainsi, on obtient la formulation forte couplée fluide–structure (1).

On suppose que les Éqs. (1) ont été discrétisées en espace et on considère un schéma implicite pour la discrétisation en temps. Par conséquent, à chaque pas de temps, on doit résoudre un système couplé non-linéaire ou, de manière équivalente, le problème de point-fixe (2). Typiquement la résolution du problème (2) est opérée par des itérations BGS (3). Cependant, il est bien connu que cette stratégie est très coûteuse en temps de calcul. En effet, à la lente convergence de l'algorithme se rajoute le coût de la mise à jour du maillage et des matrices du fluide à chaque itération BGS.

Si l'interface fluide–structure exhibe des « petites » variations (par rapport à une tolérance $\mathrm{TOL}_{\mathrm{trans}}$ donnée) entre les étapes k et $k+1$ de (3), le mouvement de l'interface peut être pris en compte, dans le fluide, par le moyen de conditions de type transpiration (voir [4,5,7]). Classiquement, ces conditions on été développées pour résoudre, à faible coût (domaine fluide fixe), des problèmes d'interaction fluide–structure avec des mouvements de l'interface de petite amplitude. Dans cette note nous proposons un algorithme de point-fixe modifié qui combine les itérations BGS (2) classiques avec un formulation de type transpiration. La Fig. 2 montre les différentes étapes de cet algorithme. Les tolérances $\mathrm{TOL}^{\mathrm{in}}_{\mathrm{trans}}$ et $\mathrm{TOL}^{\mathrm{out}}_{\mathrm{trans}}$ définissent les intervalles d'amplitudes relatives du déplacement de l'interface dans lequels la transpiration est utilisée. On remarque que le test de convergence principal de l'algorithme est toujours effectué après deux itérations consécutives de la méthode BGS standard.

Dans la Fig. 3 on a représenté le nombre de sous-itérations pour chaque pas de temps effectuées avec la méthode standard BGS et BGS avec transpiration. Dans la méthode proposée le nombre d'itérations BGS est fortement réduit et le nombre total d'itérations est similaire à celui de la méthode standard. Le gain en temps de calcul est de 48% sur 240 pas de temps. Pour une analyse complète de cette approche voir [3].

1. Introduction

In this work, we focus on the numerical solution of non-linear systems of equations arising in the time discretization of fluid–structure interaction problems where the fluid is incompressible and the structure esperiences relatively large displacements. Numerical evidences and analyses carried out on simplified problems show that implicit schemes are required to ensure stable numerical solutions for problems where the fluid and structural densities are comparable, as it happens, for instance, in haemodynamics applications (see for instance [6,8]). Consequently, we have to solve at each time step a highly non-linear coupled system where the fluid domain depends on the structural motion.

To this purpose, we here propose a modified fixed-point algorithm which combines Block-Gauss–Seidel (BGS) iterations (see [2]) with a transpiration formulation (see [4,5,7]). The underlying idea of our approach relies on

the observation that whenever the interface deformation associated to standard (BGS) iterations is moderate it can be treated through transpiration techniques, which do not require updating the fluid computational mesh and system matrices. In fact, they just involve suitable modifications of the interface boundary conditions. Numerical experiments will point out the excellent improvement in computing time obtained by the proposed algorithm. A more complete analysis of this approach will be reported in a forthcoming paper [3].

2. Mechanical problem

The system under study occupies a moving domain $\Omega(t)$ in its actual configuration. It is made of a deformable structure $\Omega^{\mathrm{s}}(t)$ (for instance the wall of a blood vessel or of a pipe-line,...) surrounding a fluid under motion (blood, oil,...) in the complement $\Omega^{\mathrm{f}}(t)$ of $\Omega^{\mathrm{s}}(t)$ in $\Omega(t)$. We aim to find the time evolution of the configuration $\Omega(t)$, as well as the velocity and Cauchy stress tensor within the fluid and the structure.

To this aim, we consider the basic conservation and constitutive laws, yet, because of the possibly large structural displacements, the time variation of the actual configuration can not be neglected. More precisely, we describe the evolution of $\Omega(t)$ by means of an Arbitrary Lagrangian Eulerian (ALE) mapping $x : \Omega_0 \times \mathbb{R}^+ \to \mathbb{R}^3$, which maps any point x_0 of a fixed (reference) configuration Ω_0 to its image $x(x_0, t)$ in the actual configuration $\Omega(t)$. On the structure ($x^{\mathrm{s}} = x|_{\Omega_0^{\mathrm{s}}}$), the point $x^{\mathrm{s}}(x_0, t)$ corresponds to the position, at time t, of the material point x_0, which implies that the configuration (or grid) velocity $w = \dot{x}$ in Ω_0^{s} is equal to the actual structural velocity. In this way, we recover the usual Lagrangian description of the structure motion. Conversely, in the fluid domain, the mapping $x^{\mathrm{f}} = x|_{\Omega_0^{\mathrm{f}}}$ can be any reasonable extension of the material interface deformation: $x^{\mathrm{f}} = \mathrm{Ext}(x^{\mathrm{s}}|_{\Gamma_0^{\mathrm{w}}})$, $\dot{x}^{\mathrm{f}}|_{\Gamma_0^{\mathrm{w}}} = \dot{x}^{\mathrm{s}}|_{\Gamma_0^{\mathrm{w}}}$. The fluid will be assumed to be Newtonian, viscous, homogeneous and incompressible with density ρ and kinetic viscosity μ. Its behavior is described by its velocity u and pressure p. The elastic solid under large displacements is described by its velocity $w^{\mathrm{s}} = \dot{x}^{\mathrm{s}}$ and stress tensor S (second Piola–Kirchoff tensor). The field S is related to x^s trough an appropriate constitutive law. The coupling between the solid and the fluid is realized through standard boundary conditions at the fluid–structure interface Γ_0^{w}, namely, the kinematic condition, expressing the continuity of the velocity, and the continuity of the stress. That is, $u = \dot{x^s}$ and $FSn_0 = J\sigma(u, p)F^{-\mathrm{T}}n_0$, with $F = \nabla_0 x$, $J = \det F$, $\sigma(u, p) = -p\,\mathrm{I} + \mu[\nabla u + (\nabla u)^{\mathrm{T}}]$. Here n_0 stands for the unit normal vector on Γ_0^{w} pointing inside Ω_0^{s}. The ALE map x allows us to recast the fluid equations into the reference domain Ω_0^{f}. Thus, the strong coupled problem, with an ALE formulation for the fluid, is given by:

$$\begin{aligned} \left.\frac{\partial J\rho u}{\partial t}\right|_{x_0} + \mathrm{div}_0\big\{J\big[\rho u \otimes (u - w) - \sigma(u, p)\big]F^{-\mathrm{T}}\big\} = 0, \quad & \text{in } \Omega_0^{\mathrm{f}} \\ \mathrm{div}\, u = 0, \quad & \text{in } \Omega^{\mathrm{f}}(t) \\ \rho_0^{\mathrm{s}}\ddot{x}^{\mathrm{s}} - \mathrm{div}_0(FS) = f, \quad & \text{in } \Omega_0^{\mathrm{s}} \end{aligned} \tag{1}$$

provided with the above interface conditions. Specific choices of the boundary conditions to be imposed on $\partial\Omega_0^{\mathrm{s}} \setminus \Gamma_0^{\mathrm{w}}$ and on the inlet and outlet boundaries $\Gamma^{\mathrm{in}}(t)$ and $\Gamma^{\mathrm{out}}(t)$ will depend on the specific problem under consideration.

3. Discrete problem and algorithm

We assume that Eqs. (1) have been appropriately discretized in space, for instance, by a FEM formulation, and we consider fully implicit schemes for the time discretization. At each time step, we have to solve a highly non-

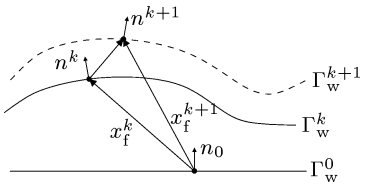

Fig. 1. Taylor expansion of the velocity.

linear system, where the fluid domain and the structural displacement are tightly coupled. We formally write the non-linear system as a fixed point problem,

$$\begin{aligned}
&(x^f, w^f) = \mathcal{M}\big((x^f, w^f), (x^s, w^s)\big) && \text{(fluid mesh update)}\\
&(u, p) = \mathcal{F}\big((u, p), (x^f, w^f)\big) && \text{(fluid subproblem)}\\
&(x^s, w^s) = \mathcal{S}\big((x^s, w^s), (u, p)\big) && \text{(solid subproblem)}
\end{aligned} \tag{2}$$

where (u, p), (x_s, w_s) and (x_f, w_f) stand, respectively, for the discretized fluid velocity and pressure, the structural motion and velocity, and the fluid domain motion and velocity at a given time step.

Standard strategies to solve (2) are Block-Jacobi or Block-Gauss–Seidel iterations, see [2]. For instance, the Block-Gauss–Seidel method consists in sub-iterating between the three subproblems. For $k \geqslant 0$ we set

$$\begin{aligned}
&\big(x_f^{k+1}, w_f^{k+1}\big) = \mathcal{M}\big(\big(x_f^k, w_f^k\big), \big(x_s^k, w_s^k\big)\big) && \text{(update the fluid mesh)}\\
&(u^{k+1}, p^{k+1}) = \mathcal{F}\big((u^k, p^k), \big(x_f^{k+1}, w_f^{k+1}\big)\big) && \text{(solve the fluid subproblem)}\\
&\big(x_s^{k+1}, w_s^{k+1}\big) = \mathcal{S}\big(\big(x_s^k, w_s^k\big), (u^{k+1}, p^{k+1})\big) && \text{(solve the solid subproblem)}
\end{aligned} \tag{3}$$

with u^0, p^0, x_s^0, w_s^0, x_f^0 and w_f^0 given from the preceding time step. We stop the iterations when the difference between two successive interface displacement falls below a fixed tolerance. Eventually, we can add a relaxation step on the structure displacement, see [6,8].

It is well known that this strategy is very CPU time consuming. Indeed, we must add, to the often slow convergence of the algorithm, the cost of updating the fluid mesh and the corresponding fluid matrices at each iteration. In this paper, we propose a method to improve the performance of the standard BGS iteration (3).

Whenever the fluid–structure interface exhibits a 'small' variation (with respect to a specified tolerance $\mathrm{TOL}_{\mathrm{trans}}$) between steps k and $k+1$ of the BGS method (3), the interface motion can be taken into account for the fluid problem through transpiration boundary conditions, without the need of updating the mesh and, consequently, the system matrices. Indeed, transpiration techniques have been developed in the past (see for instance [4,5,7]) in order to solve at low cost fluid–structure interaction problems featuring moderate deformation.

A transpiration interface condition on Γ_w^k can be derived, in a heuristic way (see [7] and refer to [4,5] for a more rigorous justification), from a truncated Taylor expansion of the fluid velocity in the vicinity of the reference fluid–structure interface Γ_w^k, see Fig. 1. Thus, the kinematic coupling condition is replaced by the following transpiration condition $u^{k+1} = w_f^{k+1} - \nabla u^{k+1}(x_f^{k+1} - x_f^k)$, on Γ_w^k. The implicit dependence on u^{k+1} can be made explicit by modifying the relation into $u^{k+1} = w_f^{k+1} - \nabla u^k(x_f^{k+1} - x_f^k)$. This latter condition can now be used to solve the fluid subproblem in the fixed fluid domain $\Omega_f^{\mathrm{trans}} = \Omega_f^k$. In the same way, the fluid load at the moving interface can be recovered from a similar Taylor expansion.

By exploiting the previous considerations we have derived the modified BGS algorithm reported in Fig. 2. The boxes on the right of Fig. 2 represents the transpiration loop. Here, instead of updating fluid mesh and matrices, we just enforce the transpiration velocity w_f^{k+1} at the interface. Tolerances $\mathrm{TOL}_{\mathrm{trans}}^{\mathrm{in}}$ and $\mathrm{TOL}_{\mathrm{trans}}^{\mathrm{out}}$ define the range of relative interface displacements where the transpiration formulation will be used. The convergence test of the whole algorithm is always done after two standard BGS iterations (*2x* in the figure), in order to ensure the convergence of the original coupled problem. This also implies that the algorithm terminates with standard BGS iterations and with an updated mesh.

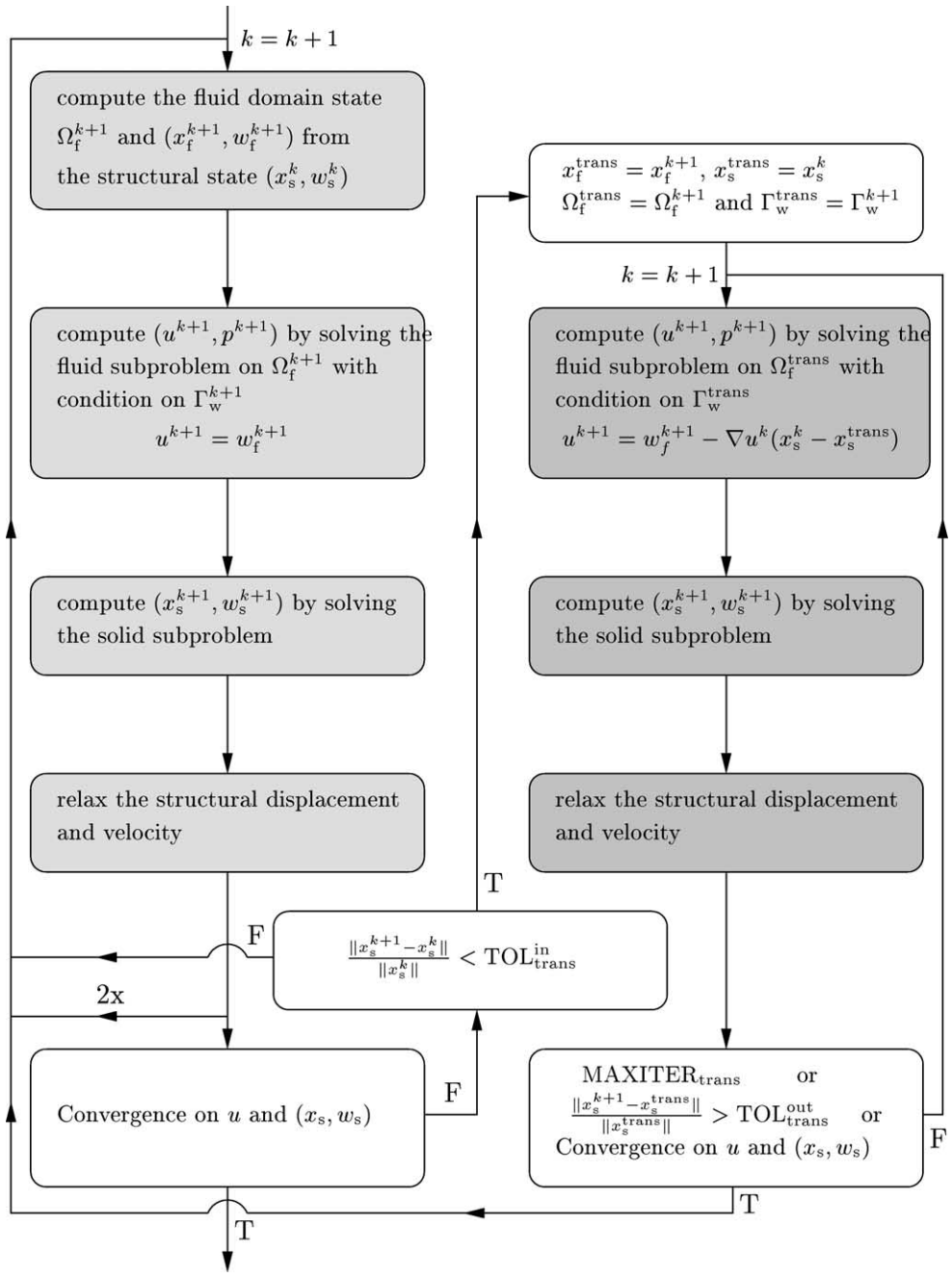

Fig. 2. Diagram of the proposed algorithm. On the left the BGS part, on the right the transpiration steps.

4. Numerical experiment

We have applied the above algorithm to a fluid–structure problem arising in the modeling of blood flow on large arteries. Namely, a thin elastic tube conveying an incompressible viscous fluid. In order to simplify the problem we

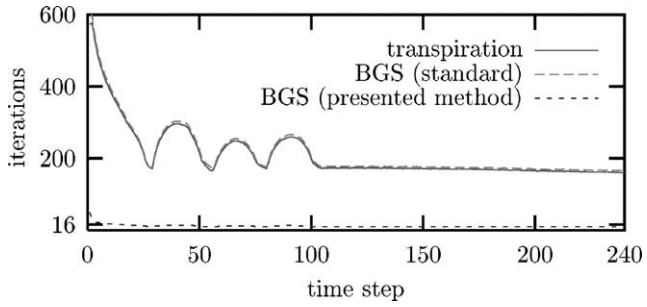

Fig. 3. Iterations history.

considered the axisymmetric incompressible Navier–Stokes equations (see [1]) combined with a generalized string model (see [8]) for the structure.

The initial domain is a cylinder of radius $R = 0.5\,\text{cm}$ and length $L = 12\,\text{cm}$. The cylinder wall may deform only along the radial direction. The fluid and the structure are initially at rest. On the outlet we impose $\sigma(u, p)n = 0$ and on the inlet two 'pressure waves' of a period of 5 ms, i.e., $\frac{P_{\text{in}}^{(j)}}{2}[1 - \cos(\frac{2\pi t}{5})]$, with $P_{\text{in}}^{(1)} = 4 \times 10^4$ dynes/cm^2 and $P_{\text{in}}^{(2)} = \frac{1}{2}P_{\text{in}}^{(1)}$. The fluid density is $\rho = 1$ gr/cm^3 and the viscosity $\mu = 0.035$ poise. For the structure we have taken: density $\rho_{\text{w}} = 1.1$ gr/cm^3, thickness $h = 0.1$ cm, Young modulus $E = 0.75 \times 10^6$ dynes/cm^2 and Poisson coefficient $\nu = 0.5$. We have adopted axisymmetric $\mathbb{P}_1$ iso $\mathbb{P}_2/\mathbb{P}_1$ finite elements for the fluid and $\mathbb{P}_1$ for the structure. For the time discretization we have used a mid-point scheme for the structure and implicit Euler for the fluid equations (see [8]), with a time step of $\Delta t = 0.1\,\text{ms}$. We have used the following values for the tolerances in the proposed numerical scheme $\text{TOL}_{\text{trans}}^{\text{in}} = 0.03$, $\text{TOL}_{\text{trans}}^{\text{out}} = 0.12$ and $\text{MAXITER}_{\text{trans}} = 50$.

In Fig. 3 we report the number of sub-iterations per time step required by the standard BGS method compared with the one obtained using the modified BGS scheme with transpiration. The number of BGS iterations is strongly reduced in the transpiration version. Let us notice that at each time step, the number of outer iterations is almost equal in the two schemes. However, the computing time is greatly reduced: a gain of 48% over 240 time steps.

References

[1] Z. Belhachmi, C. Bernardi, S. Deparis, Weighted Clément operator and application to the finite element discretisation of the axisymmetric Stokes problem, In preparation.
[2] R. Codina, M. Cervera, Block iterative algorithms for non-linear coupled problems, in: Advanced Computational Methods in Structural Mechanics, CIMNE, Barcelona, 1996.
[3] S. Deparis, M.A. Fernández, L. Formaggia, Acceleration of a fixed point algorithm for fluid–structure interaction using transpiration conditions, Accepted for printing in M2AN.
[4] T. Fanion, M.A. Fernández, P. Le Tallec, Deriving adequate formulations for fluide structure interaction problems: from ALE to transpiration, Rév. Européenne Élém. Finis 9 (6–7) (2000) 681–708.
[5] M.A. Fernández, Modèles simplifiés d'interaction fluide–structure, PhD thesis, University of Paris IX, France, 2001.
[6] P. Le Tallec, J. Mouro, Fluid structure interaction with large structural displacements, Comput. Methods Appl. Mech. Engrg. 190 (24–25) (2001) 3039–3067.
[7] G. Medic, Étude mathématique des modèles aux tensions de Reynolds et simulation numérique d'écoulements turbulents sur parois fixes et mobiles, PhD thesis, University of Paris VI, France, 1999.
[8] F. Nobile, Numerical approximation of fluid–structure interaction problems with application to haemodynamics, PhD thesis n. 2458, École Polytechnique Fédérale de Lausanne, Switzerland, 2001.