# Efficient Solution of Fluid-Structure Interaction Problems in Computational Hemodynamics

Alfio Quarteroni*,†, Paolo Crosetto* and Simone Deparis*

*MATHICSE - Chair of Modelling and Scientific Computing (CMCS), EPFL, CH-1015 Lausanne, Switzerland
†MOX-Dipartimento di Matematica "F. Brioschi", Politecnico di Milano, I-20133 Milano, Italy

**Keywords:** Hemodynamics, Fluid-Structure Interaction (FSI)
**PACS:** 46,47

## INTRODUCTION

Cardiovascular diseases are the first cause of death in industrialized countries. It is now widely established that blood fluid dynamics and its interaction with the vascular wall play an essential role in the development and evolution of vascular pathologies. Mathematical modeling and numerical simulation are precious tools which may help to understand biomechanical phenomena in both physiological and pathological states and can lead to the development of new therapies or to assist physicians in clinical decision making and treatment.

For the physiological state, a simulation of blood flow and its interaction with the vascular wall is performed using an aorta model. Then, to understand the complex flow patterns in pathological states, the simulation of a cerebral aneurysm is executed.

## DEFINITION OF THE FLUID STRUCTURE INTERACTION (FSI) PROBLEM

The interaction between the blood flow and the vessel wall is complex. To correctly study the flow in the arteries, at time $t$ we have to simulate a coupled problem that can be formally defined and sub-structured in a *fluid*, *structure problem*, and *geometry* problems. The *fluid problem*

$$\mathsf{F}(\mathbf{u}_f, \mathbf{d}_s, \mathbf{d}_f) = 0 \tag{1}$$

represents the fluid momentum and continuity equations in an Arbitrary Lagrangian-Eulerian (ALE) frame of reference

$$\begin{cases} \rho_f \dfrac{\partial \mathbf{u}}{\partial t} + (\rho_f (\mathbf{u} - \mathbf{w}) \cdot \nabla)\mathbf{u} - \nabla \cdot \sigma_f = 0 & \text{in } \Omega_f(t) \\ \nabla \cdot \mathbf{u} = 0 & \text{in } \Omega_f(t), \end{cases} \tag{2}$$

with interface boundary conditions

$$\mathbf{u} \circ \widetilde{\mathscr{A}_t} = \frac{d\mathbf{d}_s}{dt} \text{ on } \Gamma. \tag{3}$$

The unknowns are the fluid velocity and pressure, grouped in the vector variable $\mathbf{u}_f = (\mathbf{u}, p)$. The Cauchy stress tensor is noted $\sigma_f = -pI + \mu_f \frac{(\nabla \mathbf{u} + (\nabla \mathbf{u})^T)}{2}$, the density $\rho_f$ and the viscosity $\mu_f$. The ALE mapping $\widetilde{\mathscr{A}_t}$ ( e.g. [3, Chap 3]) is defined as

$$\begin{aligned} \widetilde{\mathscr{A}_t} : \hat{\Omega}_f &\rightarrow \Omega_f(t) \\ \hat{x} &\mapsto \widetilde{\mathscr{A}_t}(\hat{x}) = \hat{x} + \mathbf{d}_f(\hat{x}). \end{aligned}$$

The fluid problem is coupled with both the geometry problem, through the dependence on the fluid domain displacement $\mathbf{d}_f$ and on the associated ALE velocity $\mathbf{w}$

$$\mathbf{w} = \left.\frac{\partial \widetilde{\mathscr{A}_t}}{\partial t}\right|_{\mathbf{u}_f xo} = \left.\frac{\partial \mathbf{d}_f}{\partial t}\right|_{\mathbf{u}_f xo}, \tag{4}$$

CP1281, *ICNAAM, Numerical Analysis and Applied Mathematics, International Conference 2010, Vol.I*
edited by T. E. Simos, G. Psihoyios, and Ch. Tsitouras

and to the structure dynamics, through the boundary condition

$$\mathbf{d}_f = \mathbf{d}_s \text{ on } \Gamma. \tag{5}$$

The *structure or solid problem*

$$\mathsf{S}(\mathbf{u}_f, \mathbf{d}_s, \mathbf{d}_f) = 0 \tag{6}$$

represents the solid equation in a Lagrangian frame of reference

$$\rho_s \frac{\partial^2 \mathbf{d}_s}{\partial t^2} - \nabla \cdot \hat{\sigma}_s = 0 \text{ in } \Omega_s, \tag{7}$$

with interface boundary condition given by

$$\hat{\sigma}_s \hat{n} \quad = J_s \sigma_f F_s^{-T} \hat{n} \quad \text{ on } \Gamma \tag{8}$$

and where $\hat{\sigma}_s = \lambda \operatorname{tr}(\varepsilon) + 2\mu_s \varepsilon$, with $\varepsilon = \frac{(\nabla \mathbf{d}_s + (\nabla \mathbf{d}_s)^T)}{2}$, is the Piola stress tensor, $\lambda$ and $\mu_s$ are the Lamé coefficients, and $\Omega_s$ is the solid domain in the reference configuration. The unknown is the solid displacement $\mathbf{d}_s$. The problem is coupled with the fluid quantities $\mathbf{u}_f$ through the boundary condition (8) on $\Gamma$.

The *geometry problem*

$$\mathsf{G}(\mathbf{d}_s, \mathbf{d}_f) = 0 \tag{9}$$

represents the harmonic extension problem

$$\begin{cases} -\Delta \mathbf{d}_f &= 0 \qquad \text{in } \hat{\Omega}_f \\ \quad \mathbf{d}_f &= \mathbf{d}_s|_\Gamma \qquad \text{on } \Gamma, \end{cases} \tag{10}$$

with unknown $\mathbf{d}_f$. The latter is coupled with $\mathbf{d}_s$ through the boundary condition on $\Gamma$.

## Coupled System

To summarize, the coupled system can be written as

$$\mathsf{F}(\mathbf{u}_f, \mathbf{d}_s, \mathbf{d}_f) = 0, \tag{11}$$
$$\mathsf{G}(\mathbf{d}_s, \mathbf{d}_f) = 0, \tag{12}$$
$$\mathsf{S}(\mathbf{u}_f, \mathbf{d}_s, \mathbf{d}_f) = 0. \tag{13}$$

## Time Discretization

The coupled problem is time dependent and non-linear. Depending on the time-discretization, the non-linearities may be simplified or may even disappear. In this note we give only a short overview of possible methodologies that can be found for example in [2].

We consider a linear constitutive relation for structure equation; the extension to the non-linear case is possible and requires few modifications to the terminology. Here the indices $n$ and $n+1$ represent quantities at times $t^n$ and $t^{n+1}$.

Examples of possible time discretization schemes are

- Fully implicit: all the terms of the equations are considered implicitly, in particular the convective term of the fluid momentum equation reads $(\mathbf{u}^{n+1} - \mathbf{w}^{n+1})\nabla \mathbf{u}^{n+1}$.
- Convective explicit. The convective term in the fluid momentum equation is considered explicitly. It reads $(\mathbf{u}^n - \mathbf{w}^n)\nabla \mathbf{u}^{n+1}$.
- Geometry-convective explicit (GCE). The convective term is still considered explicitly and the domain $\Omega_f$ is extrapolated from the previous time step.

The fully implicit and convective explicit time discretization are quite similar and both give rise to non-linear problems. Instead with the GCE time discretization the problem at each time step is linear. We are going to explain this approach more in details.

## GEOMETRY-CONVECTIVE EXPLICIT PROCEDURES FOR FSI PROBLEMS

When considering a GCE discretization, the fluid velocity $\mathbf{u}^\star$ and the fluid domain $\Omega_{f_\star}$ are extrapolated from the previous time step. The geometry problem $\mathsf{G}(\mathbf{d}_s^n, \mathbf{d}_f^{n+1}) = 0$, which defines the fluid computational domain at time $t^{n+1}$ is first solved. Then use the computed geometry to solve the linear coupled fluid-structure problem

$$\mathsf{F}(\mathbf{u}_f^{n+1}, \mathbf{d}_s^{n+1}, \mathbf{d}_f^{n+1}) = 0, \tag{14}$$

$$\mathsf{S}(\mathbf{u}_f^{n+1}, \mathbf{d}_s^{n+1}, \mathbf{d}_f^{n+1}) = 0. \tag{15}$$

The weak form of this system discretized in space leads to the following linear system

$$\begin{pmatrix} C_{ff} & C_{f\Gamma} & 0 & 0 \\ 0 & \Delta_1 & 0 & -\Delta_2 \\ 0 & 0 & N_s & N_{s\Gamma} \\ C_{\Gamma f} & C_{\Gamma\Gamma} & N_{s\Gamma} & N_{\Gamma\Gamma} \end{pmatrix} \begin{pmatrix} \mathbf{u}_f{}^{n+1} \\ \mathbf{u}_\Gamma^{n+1} \\ \mathbf{d}_{f_s}{}^{n+1} \\ \mathbf{d}_{\Gamma s}^{n+1} \end{pmatrix} = \begin{pmatrix} \mathbf{f}_f^{n+1} \\ -\Delta_2 \mathbf{d}_{\Gamma s}^n \\ \mathbf{f}_s^{n+1} \\ \mathbf{f}_{\Gamma s}^{n+1} + \mathbf{f}_{\Gamma f}^{n+1} \end{pmatrix} \tag{16}$$

where $\Delta_1 = I$, $\Delta_2 = I/\delta t$, $\mathbf{u}_f$ is the discrete solution of the Navier-Stokes equations, while $\mathbf{d}_{f_s}$ is the discretized structure displacement. The velocity and the displacement on the interface are noted $\mathbf{u}_\Gamma$ and $\mathbf{d}_\Gamma$.

## Dirichlet-Neumann Iterative Method

The geometry problem $\mathsf{G}(\mathbf{d}_s^n, \mathbf{d}_f^{n+1}) = 0$ is first solved and initial guesses for $\mathbf{u}_f^{n+1,0}$ and $\mathbf{d}_s^{n+1,0}$ are extrapolated. The Dirichlet-Neumann iterative method is defined by solving at each sub-step $k$ the following problem:

$$\mathsf{F}(\mathbf{u}_f^{n+1,k+1}, \mathbf{d}_s^{n+1,k}, \mathbf{d}_f^{n+1}) = 0 \tag{17}$$

$$\mathsf{S}(\mathbf{u}_f^{n+1}, \mathbf{d}_s^{n+1,k+1}, \mathbf{d}_f^{n+1}) = 0 \tag{18}$$

until convergence.

The time and space discretized system reads

$$\left(\begin{array}{cc|cc} C_{ff} & C_{f\Gamma} & 0 & 0 \\ 0 & \Delta_1 & 0 & 0 \\ \hline 0 & 0 & N_s & N_{s\Gamma} \\ C_{\Gamma f} & C_{\Gamma\Gamma} & N_{s\Gamma} & N_{\Gamma\Gamma} \end{array}\right) \begin{pmatrix} \mathbf{u}_f{}^{k+1} \\ \mathbf{u}_{f_\Gamma}{}^{k+1} \\ \mathbf{d}_{f_s}{}^{k+1} \\ \mathbf{d}_{f_\Gamma}{}^{k+1} \end{pmatrix} = \begin{pmatrix} \mathbf{f}_f^{n+1} \\ -\Delta_2 \mathbf{d}_{f_\Gamma}{}^n + \Delta_2 \mathbf{d}_{f_\Gamma}{}^k \\ \mathbf{f}_s^{n+1} \\ \mathbf{f}_{\Gamma s}^{n+1} + \mathbf{f}_{\Gamma f}^{n+1} \end{pmatrix}. \tag{19}$$

We can notice that this expression represents the block Gauss-Seidel scheme applied to the linear system (16), i.e., the Richardson algorithm with acceleration parameter $\omega = 1$ preconditioned with the matrix in (19). In fact by writing equation (16) as $A\mathbf{x} = \mathbf{b}$ and by calling $A$ and $P_{DN}$ the matrices in (16) and (19) we can rewrite the system as

$$P_{DN}\mathbf{x}^{k+1} = (P_{DN} - A)\mathbf{x}^k + \mathbf{b}$$

which is the Richardson method

$$\mathbf{x}^{k+1} = \mathbf{x}^k + P_{DN}^{-1}(-A\mathbf{x} + \mathbf{b}).$$

Otherwise if we choose to solve the system (16) using GMRES [1], preconditioned with $P_{DN}$ (which we call *Dirichlet-Neumann preconditioner*), we obtain a different and more efficient algorithm.

## APPLICATIONS

We end this presentation by addressing two application using the method previously introduced.

## Aorta

The aorta is the largest artery in the human body. Its function is to distribute the blood to the systemic circulation. Here we present a simulation of the blood flow from the aortic valve to the descending aorta. Interactions between the blood flow and the arterial wall are taken into account.

We imposed at the inlet a typical physiological pressure for a human aorta (Figure 1)

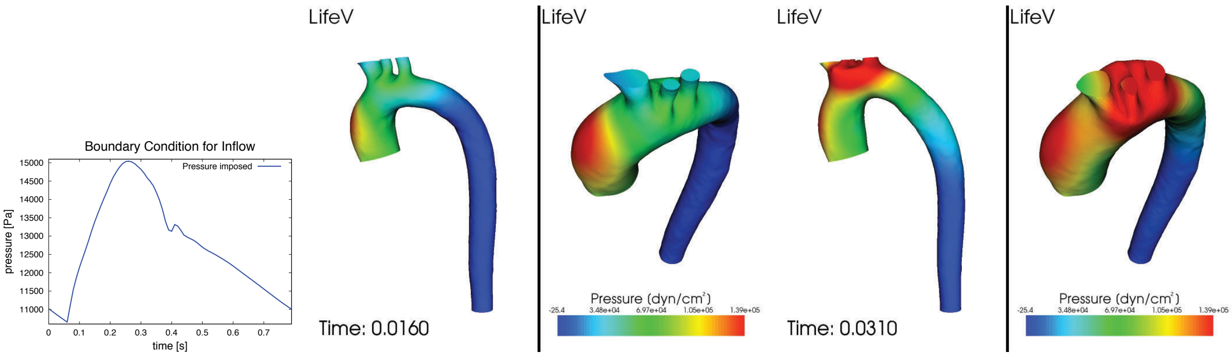


**FIGURE 1.** Left: Mean pressure at aortic valve. Center and Right: Pressure at times 0.016 and 0.031 s

## Cerebral aneurysm

An aneurysm is a local dilatation of an artery, of a vein or of the heart. It develops and grows over many years and the final stage is the rupture of the aneurysm that leads to hemorrhage and in the worst cases to death. Once again studying the blood fluid dynamics is mandatory to understand the relation between the geometry of the vessel and hemodynamic parameters. The final goal of these simulations is to allow the understanding of the flow inside an aneurysm and to predict its evolution based on patient specific data. Figure 2 represents the pressure, velocity, and displacement of the aneurism at time 0.121 s.

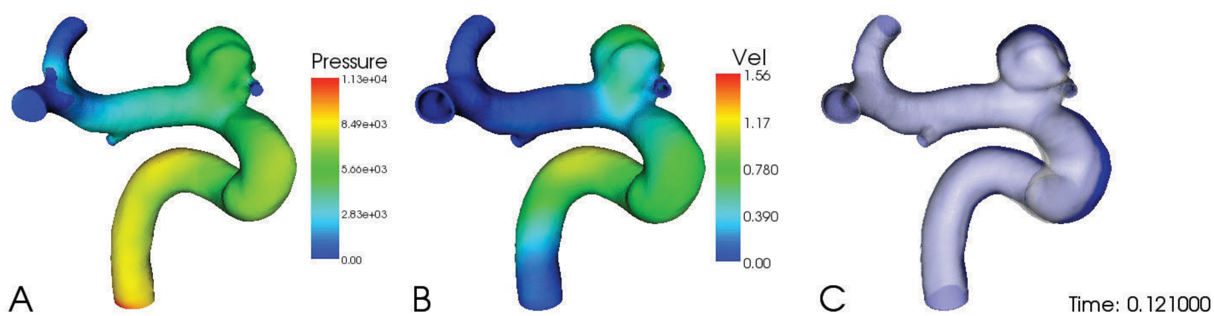


**FIGURE 2.** A - Pressure for the fluid; B - Velocity of the structure; C - Displacement of the structure (gray) in comparison with the initial position (blue); D - Displacement of the structure

*Aknowlegments.* We thank the LHTC, EPFL, for having provided the aorta geometry and the pressure at the aortic valve. We also whish to thank Dr. Gilles Fourestey and Jean Bonnemain for fruitful discussions. All the simulations have been done with LifeV (www.lifev.org)

## REFERENCES

1. S. Badia, A. Quaini, and A. Quarteroni. Modular vs. non-modular preconditioners for fluid-structure systems with large added-mass effect. *Comput. Methods Appl. Mech. Engrg.*, 197(49-50):4216–4232, 2008.
2. P. Crosetto, S. Deparis, G. Fourestey, and A. Quarteroni. Parallel solvers and preconditioners for fluid-structure interaction problems arising in haemodynamics. In preparation.
3. L. Formaggia, A. Quarteroni, and A. Veneziani, editors. *Cardiovascular Mathematics: Modeling and Simulation of the Circulatory System*, volume 1 of *Modeling, Simulation and Applications*. Springer, Milan, 2009.