Optimization of Fluid-Structure Interaction Scheme for Arterial Flow Simulations

Esko Järvinen, Peter Råback and Mikko Lyly
CSC - Scientific Computing Ltd.

Abstract: In this paper we investigate a simple method to improve the convergence of a loose coupled fluid-structure interaction (FSI) models used in arterial flow simulations. The impracticality of straightforward coupling of the fluid and structure equations by relaxation method is first demonstrated, and an alternatively way based on artificial compressibility is presented. The latter is proven to provide a fast and smooth convergence in the coupling iteration. In the computations we apply finite elements discretization for the fluid and elastic solid equations. In order to get the outlet of the test model work correctly, a one dimensional FSI model is combined with the higher dimensional test model.

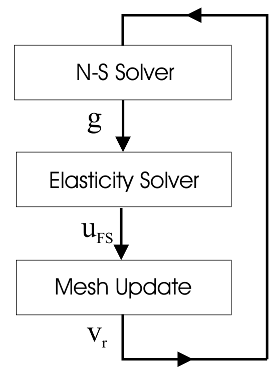

Figure 1: Sequential coupling algorithm.

Introduction

In the simultaneous solution of the Navier-Stokes (N-S) equations and the elasticity equations two basic approaches are available. In the strong coupling the complete system is solved at each fixed time. Even if the flow and the elasticity solvers exists in separate, the strong coupling approach may require almost complete re-writing of the existing program codes. Another much more straightforward approach, and probably at the present the most common in the fluid-structure interaction modelling, is to use loose coupling between the fluid and solid models.

In the loose coupling approach, on each time step we apply a sequential iterative algorithm illustrated in Figure 1. For the formulation of flow equations in the moving domain we use Arbitrary Lagrangian Eulerian (ALE) method. The communication between the fluid and the solid solvers reduces into *i)* transmission of the fluid traction g at the solid/fluid interface boundary (FS) to the elasticity solver, *ii)* giving the new position of the interface boundary u_{FS} to the mesh update solver, and *iii)* passing the new mesh velocity v_r for the flow solver.

A disadvantage of this kind of straightforward coupling scheme is that relaxation of the variables $a_i = [g, u_{FS}, v_r]$ is often needed, see Formaggia et al. [1]. The reason for the need of relaxations is due to the incompressible condition $\nabla \cdot u = 0$. The elasticity equation is not restricted with the incompressibility constraint and therefore the new fluid domain displacement field extended from the new wall position may be such that the incompressibility condition is difficult to fulfil. The system is very ill-conditioned and therefore the substantial strong relaxation is needed.

An alternative way to tackle the coupling problem, the method of artificial compressibility, has been reported in [2, 3].

Method

The artificial compressibility scheme used in this work is a variation of the classical Uzawa algorithm [4], and is basically identical to the artificial compressibility methods developed by Chorin [5], except that in this case the term of pressure time derivative in the continuity equation is used to stabilize the coupling between the incompressible constraint of fluid and an elastic wall, rather than to decouple the momentum and continuity equations and stabilize the solution of the divergence-free Navier-Stokes equations.

The idea of this method, which overcomes the problem with incompressibility constraint of the velocity field, is to allow the density of the fluid vary corresponding the pressure on the structure,

$$d\rho/\rho = c \cdot dp,$$

where c is the artificial compressibility parameter and dp is the local pressure change. Inserting this to the continuity equation for compressible flow and neglecting the space derivative of the density, we get a new continuity equation in the form

$$c(dp/dt) + \nabla \cdot u = 0. \tag{1}$$

During the iterations within the coupling algorithm, the first term on the left side in the equation vanishes. The pressure difference dp refers to pressure change between coupling iterations.

The computations are made by in-house finite element based solver Elmer [6]. As a test geometry we use an axisymmetric segment of a straight tube. A pressure pulse is exposed at the inlet of the tube and the pulse is propagating in an elastic tube to the outlet, which is combined with a one dimensional FSI-model. One dimensional model provide a correctly behaving and non-reflecting outlet boundary condition for the tube segment. For the mesh update we apply the linear elasticity equation.

Our tests with the artificial compressibility method demonstrates a fast, smooth and nearly monotonous convergence of the coupling iterations. Besides the good convergence properties, the artificial compressibility method does not suffer the strict limitations of the relaxation method.

If we consider a linear dependence of the pressure inside the tube on the displacement of the wall, the artificial compressibility parameter c in Eq.(1) can be expressed as

$$c = \frac{1}{V}\frac{dV}{dp}.$$

Expression is closely related to several measures of arterial stiffness commonly applied in clinical diagnostics [7]. The equation may be presented in general form convenient for numerical computations in an arbitrary fluid domains, see [3]. For a simple structure such as a non-branching segment of an artery, a following analytical expression for the optimal size of c can be derived

$$c = (1 - \nu^2)\frac{(2R + \Delta R)}{Eh}, \tag{2}$$

where ΔR is the radial displacement of the wall, a quantity which is not in general case defined in advance, but which magnitude is less than 0.1R in normal physiological situations in healthy arteries. Assuming Poisson ratio ν equal to zero, and that the wall motion is small compared to the tube radius, and therefore neglecting the term with the radial displacement ΔR, the equation above is reduced into form $c = 2R/(Eh)$ [2, 3].

An example of the results is seen in Figure 2. We varied the value of the parameter c with different values of Young's modulus of the tube and looked for an optimal size for c.

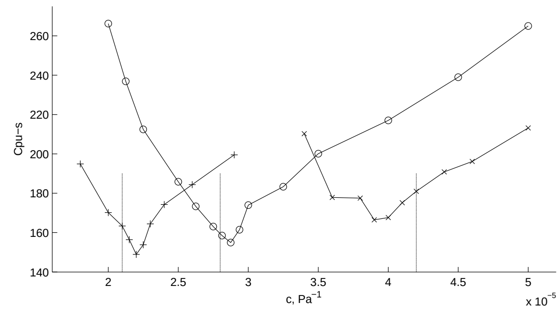

Figure 2: The optimal size of parameter c with three different Young's -modulus: $2.0 \cdot 10^5$ Pa (-x-x-), $3.0 \cdot 10^5$ Pa (-o-o-) and $4.0 \cdot 10^5$ Pa (-+-+-). The vertical lines represents the optimal size of c calculated by the analytical expression.

The results demonstrates that the optimal size of c can be approximated reasonable well with the theoretical expression (2). The optimal size seems to locate in relatively narrow range, implying that its size should be predicted accurately in order to make computations cost effective.

References

[1] Formaggia, L., Gerbeau, J.-F. Nobile, F., Quarteroni, A., *On the coupling of 3D and 1D Navier-Stokes equations for the problems in compliant vessels*, INRIA, Rapport de Recherche nro 3862, 2000

[2] Riemslagh K., Vierendeels J., Dick E., *An efficient coupling procedure for flexible wall fluid-structure interaction*, ECCOMAS 2000, Barcelona, Spain, September 2000

[3] Råback, P., Ruokolainen, J., Lyly, M., Järvinen, E., *Fluid-structure interaction boundary conditions by artificial compressibility*, ECCOMAS 2001, Swansea, UK, September 2000

[4] Glowinski, R., *Handbook of Numerical Analysis. Numerical Methods for Fluids (Part 3)*, North-Holland, 2003

[5] Chorin, A. J, *A Numerical Method for Solving Incompressible Viscous Flow Problems*, Journal of Computational Physics (2), pp. 12-26, 1967

[6] Elmer Software Package, CSC, the Finnish IT center for science, *http://www.csc.fi/elmer/*

[7] O'Rourke, M. F., Staessen, J. A., *Clinical applications of arterial stiffness: Definitions and reference values*, American Journal of Hypertension (15), pp 426-444, 2002