Computers and Fluids 153 (2017) 61–75

Contents lists available at ScienceDirect

Computers and Fluids

journal homepage: www.elsevier.com/locate/compfluid

A parallel monolithic approach for fluid-structure interaction in a cerebral aneurysm

Ali Eken, Mehmet Sahin*

Faculty of Aeronautics and Astronautics, Astronautical Engineering Department, Istanbul Technical University, Maslak, Istanbul, 34469, Turkey

ARTICLE INFO

Article history:
Received 15 January 2017
Revised 17 April 2017
Accepted 9 May 2017
Available online 10 May 2017

Keywords:
Fluid-structure interaction
Cerebral aneurysm
Monolithic method
Unstructured finite volume method
Finite element method

ABSTRACT

A parallel fully-coupled numerical algorithm has been developed for the fluid-structure interaction problem in a cerebral artery with aneurysm. For the fluid part of the problem, an Arbitrary Lagrangian-Eulerian formulation based on the side-centered unstructured finite volume method is employed for the governing incompressible Navier–Stokes equations. The deformation of the solid domain is governed by the constitutive laws for the nonlinear Saint Venant-Kirchhoff material and the classical Galerkin finite element method is used to discretise the governing equations in a Lagrangian frame. The time integration method for the structure domain is based on the Newmark type generalized−α method while the second-order backward difference (BDF2) is used in the fluid domain. A special attention is given to construct an algorithm obeying the local/global discrete geometric conservation laws (DGCL) in order to conserve fluid volume at machine precision when the fluid domain is entirely enclosed by solid domain boundary. Therefore, a compatible kinematic boundary condition is applied at the interface between the solid and fluid domains. The parallel implementation of the present fully coupled unstructured fluid-structure solver is based on the PETSc library and a one-level restricted additive Schwarz preconditioner with a block-incomplete factorization within each partitioned sub-domains is utilized for the resulting fully coupled system. The proposed algorithm is initially validated for a pressure pulse propagating in a flexible tube and the mass conservation accuracy is tested for a thin elastic sphere filled with an incompressible fluid in a circular tube. Then the numerical method is applied to a complicated problem involving unsteady pulsatile blood flow in a cerebral artery with aneurysm as a realistic fluid-structure interaction problem encountered in biomechanics. Various hemodynamic quantities of interest like fluid velocities, blood pressure and wall shear stresses (WSS) are computed as well as the time dependent artery wall deformations.

1. Introduction

Hemodynamics, which is basically the fluid dynamics of blood flow, plays a significant role in cardiovascular physiology in understanding the physical mechanisms that govern the interaction between blood pressure, velocity, shear stresses, vascular resistance, and how these quantities of interest are impacted by the vessel wall geometry and the boundary conditions. The computational models for simulating the mechanics of the blood flow through the veins are mainly based on solving the Navier–Stokes equations of incompressible viscous flow on the computational cardiovascular domain of interest. Although the primary variables of such a numerical model are the flow velocity and the pressure, the hemodynamic quantities of significant importance are mainly the wall shear stresses (WSS) and the wall shear stress gradients (WSSG), since the formation and progression of diverse cardiovascular diseases such as aneurysms, atherogenesis, growth and remodeling of arteries are directly related to these shear stresses, which can be simply derived from the flow velocity field.

Computational hemodynamics has emerged as a powerful non-invasive tool in investigation of cardiovascular diseases, which are influenced by hemodynamic factors. The first applications of pure computational fluid dynamics (CFD) approaches for the simulation of blood flow in patient-specific geometries can be traced back to the studies of Taylor et al. [1], where the authors applied a stabilized finite element based numerical modeling for the solu-

* Corresponding author.
E-mail address: msahin.ae00@gtalumni.org (M. Sahin).

http://dx.doi.org/10.1016/j.compfluid.2017.05.005
0045-7930/

tion of the incompressible Navier–Stokes equations on unstructured tetrahedral meshes generated using octree method. The CFD approaches for cardiovascular flow simulation still have valuable use, for example for cerebral aneurysms [2,3]; coronary arteries [4], [5]; simulation of stenting technology for cerebral aneurysms [6], and coronary arteries [7]. New methods also have been proposed to impose realistic boundary conditions in context of multiscale coupling methods as presented by Vignon-Clementel et al. [8] and Moghadam et al. [9]. Although the use of CFD approach in biomedical applications has enabled advances in understanding of the mechanics of blood flow and how the hemodynamic quantities of interest are interrelated, there is the inherent drawback of the pure CFD approach that the solid boundary, like the blood vessel in the case of cardiovascular systems, is modeled as being rigid, which is physiologically unrealistic. In fact, in many cases large displacements of vessel walls are observed as a result of hemodynamic forces, which further elevate the blood flow pattern in an interactive way to further reshape the hemodynamics of the system. In addition, the deformation velocity at the blood vessel boundary will significantly alter the velocity field within the blood vessel. This is particularly important in the case of elastic waves propagating along the artery walls [10]. In several cardiovascular fluid-structure interaction (FSI) studies, the effects of employing a flexible vessel wall structure opposed to the rigid wall assumption of pure CFD modeling have been demonstrated, for instance in the context of cerebral aneurysms [11] and [12]; for carotid artery [13], and for the total cavopulmonary connection [14]. It has been observed that the rigid wall assumption consistently overestimates the wall shear stresses compared to elastic vessel wall, and this overestimation can be substantially different in FSI and rigid wall simulations [10]. In order to obtain a more realistic solution, the blood flow through the surrounding vessel wall need to be modeled as a coupled FSI problem, although it will introduce increasing modeling complexity and computational demands. In cardiovascular FSI, the blood flow within the moving/deforming elastic vessel walls is generally governed by the incompressible Navier–Stokes equations. One widely used method to formulate the Navier–Stokes equations on moving boundaries in blood flow simulations is the arbitrary Lagrangian–Eulerian (ALE) formulation as depicted in [15–17]. An alternative way is the space-time methodology presented in [11,13,17,18], which somewhat suffers from computational efficiency per time step, but has higher time accuracy [12].

In the ALE method [19], the mesh follows the interface between the fluid and solid boundary and the governing equations are discretized on a moving mesh. This differs from the standard Eulerian formulation in a way that the mesh movement has to fulfill special conditions in order to maintain the accuracy and the stability of the time integration scheme. This condition is satisfied by the enforcement of the so-called discrete geometric conservation law (DGCL) as coined by Thomas and Lombard [20]. Although the GCL is satisfied easily in the continuous sense, its discrete implementation may not be trivially satisfied. The ALE time integration scheme developed by Koobus and Farhat [21] is based on more continuous time integration of the fluxes and offers second-order accuracy in time obeying the GCL. Geuzaine et al. [22] have showed that the GCL is neither a necessary nor a sufficient condition for an ALE scheme to preserve its order of time accuracy established on fixed meshes. The ALE approach was subsequently adopted within the finite element context to solve free surface problems of incompressible viscous fluid flow [23]. In the case of an FSI problem, the deformable fluid-structure interface is taken into account and the fluid points at the fluid-solid interface are moved in a Lagrangian way [24]. In the current work, a special attention is given to construct an FSI algorithm with exact mass conservation at machine precision. The exact mass conservation for FSI simulations is a rather difficult challenge to overcome since it requires not only the satisfaction of continuity equation within each element at machine precision (div-stable discretization) but also a compatible kinematic boundary condition at the fluid-structure interface. The compatible kinematic boundary condition ensures that the interface velocity is in accordance with the global discrete geometric conservation law (DGCL) as well as the discrete form of the continuity equation so that the total fluid mass is conserved at machine precision. To our best knowledge, the exact mass conservation for three-dimensional fluid-structure interaction problems is attempted only in the work of Bazilevs et al. [25] using a NURBS-based isogeometric analysis. However, the mass conservation is achieved on the order of 10^{-4} and the error is attributed to the iterative solver tolerance. Although the Eulerian approaches have also been used for the simulation of the fluid-structure interaction problems around complex geometrical shapes [26–28], the use of these approaches for large mesh deformations requires relatively high mesh resolution in large percentages of the fluid computational domain, or adaptive mesh refinement, in order to properly capture the viscous effects.

There are mainly two solution procedures for the numerical simulation of fluid-structure interaction problems: partitioned (segregated) [29–31] or fully coupled (monolithic) [32–37] methods. In the partitioned approach, separate solvers are utilized for the fluid and structure subproblems. The main advantage of the partitioned approach is the ability to reuse existing solvers which allows the application of different, possibly more efficient, computational methods specifically developed for either the fluid or the structure subproblem. Both explicit or implicit methods can be used in order to couple the fluid and structure solvers in partitioned approaches. In explicit partitioned methods, which are also known as loosely or weakly coupled methods, typically a fixed point (Picard) iteration is employed to obtain a coupled solution. Although the implementation of this approach is relatively easy, it does, however, suffer from some serious drawbacks. The fixed point iterations tend to converge slowly and the iterations may diverge in the presence of strong fluid-structure coupling due to the high fluid/structure density ratio which causes the so-called artificial added mass effect [38]. For strong coupling in partitioned procedures, on the contrary, several fluid and structure computations are performed in a single time-step until a satisfactory convergence tolerance is reached. This approach, however, requires costly sub-iterations and the sub-iteration convergence may not be guaranteed. In addition, the accuracy and the stability of the coupling method used in FSI modeling are strongly dependent on the physical problem that it is being applied to. For instance, in the case of a biomechanics application, such as cardiovascular FSI, the densities of the blood and the surrounding cardiovascular tissue are comparable, which yields a highly nonlinear strong coupling of the fluid and the structure so that the classical segregated time-marching methods may result in unconditionally unstable solutions [39]. In a fully coupled (monolithic) approach, the fluid and structure equations are discretized and solved simultaneously as a single equation system for the entire problem. In the present monolithic approach, the original system of equations is preconditioned with an upper triangular right preconditioner which results in a scaled discrete Laplacian instead of a zero block in the original system due to the divergence-free constraint. Then a one-level restricted additive Schwarz preconditioner with a block-incomplete factorization within each partitioned sub-domains is utilized for the resulting fully coupled system. The implementation of the preconditioned Krylov subspace algorithm, matrix-matrix multiplication and the restricted additive Schwarz preconditioner are carried out using the PETSc [40] software package developed at the Argonne National Laboratories.