Commun. Comput. Phys.
doi: 10.4208/cicp.060510.150511s

Vol. **11**, No. 4, pp. 1279-1299
April 2012

A Parallel Domain Decomposition Algorithm for Simulating Blood Flow with Incompressible Navier-Stokes Equations with Resistive Boundary Condition

Yuqi Wu[1,*] and Xiao-Chuan Cai[2]

[1] *Department of Applied Mathematics, University of Colorado at Boulder, Boulder, CO 80309, USA.*
[2] *Department of Computer Science, University of Colorado at Boulder, Boulder, CO 80309, USA.*

Received 15 May 2010; Accepted (in revised version) 15 May 2011

Available online 30 November 2011

Abstract. We introduce and study a parallel domain decomposition algorithm for the simulation of blood flow in compliant arteries using a fully-coupled system of nonlinear partial differential equations consisting of a linear elasticity equation and the incompressible Navier-Stokes equations with a resistive outflow boundary condition. The system is discretized with a finite element method on unstructured moving meshes and solved by a Newton-Krylov algorithm preconditioned with an overlapping restricted additive Schwarz method. The resistive outflow boundary condition plays an interesting role in the accuracy of the blood flow simulation and we provide a numerical comparison of its accuracy with the standard pressure type boundary condition. We also discuss the parallel performance of the implicit domain decomposition method for solving the fully coupled nonlinear system on a supercomputer with a few hundred processors.

AMS subject classifications: 74F10, 65M55, 35R37, 65Y05, 68W10

Key words: Fluid-structure interaction, blood flow, mesh movement, resistive boundary condition, additive Schwarz, domain decomposition, parallel computing.

---

1 Introduction

Artery diseases, such as the plaque formation, are closely related to flow properties of the blood and to the interaction between the blood and the artery walls. Different from the

*Corresponding author. *Email addresses:* `yuqi.wu@colorado.edu` (Y. Wu), `cai@cs.colorado.edu` (X.-C. Cai)

traditional experimental approach, the computational approach has the ability to simulate the velocity and pressure fields in a virtual environment, which is important in predicting the development of the disease and helps the treatment of the diseases [30]. Unfortunately, computer modeling of blood flow in arteries is a challenging problem [31]. In this paper, we develop a parallel fluid-structure interaction algorithm for the simulation of blood flow in compliant arteries using a fully-coupled system of partial differential equations.

One of the main challenges is the effective coupling of the fluid and the wall deformability. Two well-known formulations for the fluid-structure coupling are iterative and monolithic. In iterative approaches, the fluid, solid equations are solved sequentially, update each other's boundary conditions, until some desired tolerance is reached [17,26]. This enables the use of existing well-established fluid and structure solvers. However, difficulties in the form of lack of convergence have been addressed in a number of situations [24,28]. In monolithic approaches, the fluid, solid and mesh movement equations are solved simultaneously in fully-coupled fashion, where the coupling conditions enforced strongly as part of the system [3–6,19,23]. The fully-coupled approach shows to be more robust. Many of the convergence problems encountered within the iterative approach can be avoided with the monolithic approach [4]. Of course, there is a price to pay in this approach, solving the fully-coupled system is more computationally expensive. In this paper, we use the monolithic approach within the ALE framework.

In the blood flow simulation, the size and complexity of the circulation precludes a computational representation for the complete circuit in human body. Numerical models must invariably be truncated and divided into the upstream domain (modeled domain) and the downstream domain. And appropriate outflow boundary condition must be specified for the modeled domain. The downstream domain includes a vast quantities of smaller arteries, arterioles, capillaries, venules and veins returning blood to the heart. As a consequence, solutions to the governing equations of blood flow in the modeled domain depend closely on the outflow boundary conditions imposed to represent the influence from the downstream vascular system. By ignoring the effect of the downstream circulation, these boundary conditions may result in inaccurate predictions of velocity and pressure fields. In [16,32,33], a suggested solution is to use a reduced dimensional model to represent the downstream vessels and provide boundary conditions for the higher dimensional upstream model, where high-resolution information is needed.

In [4], Barker and Cai successfully developed a scalable parallel method for fluid-structure interaction problem. However, their model only use zero-traction as outlet boundary conditions. The blood pressure is not computed accurately from reports [33] and reference therein. For this reason, this paper describes the extension of Barker's paper to the following two aspects.

• In this paper, a more physically realistic outflow boundary condition is considered, namely the resistance of the flow. Where we assume the pressure P is a constant over the upstream outlets, the relation $P = QR$, representing the resistance to the flow of the down-