ARTICLE IN PRESS

Journal of Biomechanics ∎ (∎∎∎∎) ∎∎∎–∎∎∎

Contents lists available at ScienceDirect

Journal of Biomechanics

journal homepage: www.elsevier.com/locate/jbiomech
www.JBiomech.com

Numerical methods for simulating blood flow at macro, micro, and multi scales

Yohsuke Imai [a,*], Toshihiro Omori [a], Yuji Shimogonya [b], Takami Yamaguchi [c], Takuji Ishikawa [a,c]

[a] School of Engineering, Tohoku University, Sendai, Japan
[b] Frontier Research Institute for Interdisciplinary Science, Tohoku University, Sendai, Japan
[c] Graduate School of Biomedical Engineering, Tohoku University, Sendai, Japan

ARTICLE INFO

Article history:
Accepted 7 November 2015

Keywords:
Hemodynamics
Red blood cell
Computational fluid dynamics

ABSTRACT

In the past decade, numerical methods for the computational biomechanics of blood flow have progressed to overcome difficulties in diverse applications from cellular to organ scales. Such numerical methods may be classified by the type of computational mesh used for the fluid domain, into fixed mesh methods, moving mesh (boundary-fitted mesh) methods, and mesh-free methods. The type of computational mesh used is closely related to the characteristics of each method. We herein provide an overview of numerical methods recently used to simulate blood flow at macro and micro scales, with a focus on computational meshes. We also discuss recent progress in the multi-scale modeling of blood flow.

1. Introduction

In the past decade, much progress has been made in the computational biomechanics of blood flow. First, numerical simulations have been used for more clinical purposes. Arterial blood flow has been simulated using geometries based on medical images to analyze hemodynamics in patient-specific models. The efficiency of endovascular treatments, including stenting and coiling, has also been examined using patient-specific models. In addition, considerable progress has been made in the numerical simulation of cellular flow. The mechanics of red blood cells (RBCs) and their single-cell dynamics have been modeled and simulated, and this work has been extended to multi-cellular flow in the microcirculation.

Numerical methods have also overcome difficulties such as complex geometry, wall deformation, and the hydrodynamic interaction of cells, in diverse applications from the cellular to organ scales. Various numerical methods have been proposed, but the advantages and disadvantages of each method are not clear for researchers who are not specialists in computational mechanics. Numerical methods in the computational biomechanics of blood flow may be classified by the type of computational mesh used for the fluid domain, into fixed mesh methods, moving mesh (boundary-fitted mesh) methods, and mesh-free methods. The type of computational mesh is closely related to the characteristics of the associated method. Therefore, we herein provide an overview of numerical methods recently used for blood flow simulations at macro and micro scales, with a focus on computational meshes. There have also been attempts at the multi-scale modeling of blood flow. Examples include low-dimensional models for inflow and outflow boundary conditions, coarse-grained models of cell membranes, and multi-scale models of the blood coagulation cascade. We also discuss recent progress in such multi-scale models of blood flow.

2. Macro-scale problems

Blood flow simulations at the macro scale often aim to quantify wall shear stress in complex patient-specific geometries, and boundary-fitted meshes are suitable for this purpose. The arterial wall is often assumed to be rigid for computational fluid dynamics (CFD) simulations of cerebral arteries because the wall deformation due to pulsation is relatively small in the cerebrovascular system. By contrast, blood flow in the heart and cardiovascular system must incorporate wall deformation. A computational fluid-structure interaction (FSI) approach is applied to these problems using moving boundary-fitted meshes.

* Correspondence to: School of Engineering, Tohoku University, 6-6-01 Aoba, Aoba, Sendai 980-8579, Japan. Tel.: +81 22 795 7005; fax: +81 22 795 6959.
E-mail address: yimai@pfsl.mech.tohoku.ac.jp (Y. Imai).

http://dx.doi.org/10.1016/j.jbiomech.2015.11.047
0021-9290/

Please cite this article as: Imai, Y., et al., Numerical methods for simulating blood flow at macro, micro, and multi scales. Journal of Biomechanics (2015), http://dx.doi.org/10.1016/j.jbiomech.2015.11.047

2.1. Rigid wall

Image-based modeling techniques for blood flow simulation were pioneered in the late 1990s and early 2000s (Milner et al., 1998; Taylor et al., 1999; Steinman et al., 2003). Steinman et al. (2003) extracted the geometry of a giant intracranial aneurysm from computed rotational angiography volumetric image data. This geometry and a representative flow rate waveform were used as the boundary conditions for CFD simulation to reconstruct the time-varying, three-dimensional (3D) blood flow patterns in the giant intracranial aneurysm. They performed simulations using an in-house CFD solver based on the finite element method (FEM). While structured grids were used previously (Shimogonya et al., 2009), a boundary-fitted mesh is often constructed using unstructured grids comprising tetrahedral, prismatic, and other types of geometric elements. FEM is suitable for boundary-fitted meshes and has been used in CFD simulations of vessels with multiple branches or irregular morphologies. For example, Cebral et al. (2005) used an in-house FEM solver with an unstructured body-fitted mesh to investigate the hemodynamics of cerebral arteries with aneurysms and branches, performing 39 CFD simulations. In addition to in-house solvers, open-source solvers have also been used in CFD studies of large vessels. They include the FEM-based FEniCS (Evju et al., 2013; Valen-Sendstad et al., 2014), the spectral/hp element method-based NEKTAR (Karniadakis and Sherwin, 2005; Baek et al., 2010), and the finite volume method (FVM)-based OpenFOAM (Gambaruto and João, 2012; Shimogonya et al., 2012; Morales and Bonnefous, 2015). Fig. 1 shows an example of blood flow simulation using OpenFOAM in a patient-specific cerebral artery with an aneurysm. The velocity and wall shear stress were successfully calculated using an unstructured boundary-fitted mesh method with a near-wall layer mesh, which enhances the resolution of hemodynamic wall parameters such as wall shear stress (Bazilevs et al., 2010).

Recently, commercial CFD solvers based on FVM have been widely used to efficiently simulate the blood flow of large vessels, especially in large-scale clinical studies of intracranial aneurysms. Xiang et al. (2011) simulated pulsatile blood flow in 119 intracranial aneurysms (38 ruptured and 81 unruptured) using Star-CD. Their study evaluated six morphological and seven hemodynamic parameters, including wall shear stress (WSS) and the oscillatory shear index (OSI), to identify significant parameters that discriminate the aneurysm rupture status. Takao et al. (2012) compared the hemodynamic differences between 87 unruptured and 13 ruptured intracranial aneurysms by conducting CFD simulations with ANSYS CFX. Imai et al. (2008) used Fluent to perform CFD simulations in idealized models of saccular aneurysms. They parametrically examined the effect of parent artery geometry on the inflow patterns and fluxes into saccular aneurysms. A similar study was also conducted by Baharoglu et al. (2010) using Fluent.

Steinman et al. (2013) organized a CFD challenge that sought to assess the variability of CFD solutions for pressure and flow in a giant aneurysm obtained by different research groups. Of the 27 CFD solutions that were submitted to the challenge, 5 used in-house solvers, 5 used open-source solvers, and 17 used commercial solvers. By assessing the variability of the CFD solutions and comparing them with experimental data, the study demonstrated that CFD can consistently predict pressure across a wide range of solvers. The majority of the 27 solutions predicted the inlet-outlet pressure drops at peak systole within 8% of each other. In contrast, velocity in the aneurysm was found to be more variable; iso-surfaces of the velocity magnitude varied largely among the solutions. While the majority of the 27 solutions, which used normal temporal resolutions, reported smooth iso-surfaces in the aneurysm, a few solutions, which used high temporal resolutions (below 0.2 ms), detected flow instabilities. However, other solutions with comparably high temporal resolutions did not show clear flow instabilities. These results indicated that the flow velocity magnitude and patterns in the aneurysm were highly dependent on the choice of the solution strategies, such as the numerical scheme and resolution. Other CFD challenges involving aneurysm stenting (Radaelli et al., 2008) and aneurysm rupture (Janiga et al., 2015) have also been conducted. Although the practical significance of the variability found in these challenges is not clear, one must consider the fact that the solution strategies can markedly affect the CFD solutions. The solution strategies should be determined carefully based on their sensitivity with respect to the simulation results, considering in particular the spatial and temporal discretization schemes, mesh size, mesh design (e.g. presence or absence of boundary layer mesh, layer thickness, and number of layers), and temporal resolution, regardless of the type of CFD solver used (in-house, open-source, or commercial).

2.2. Deformable wall

Peskin (1977) developed the immersed boundary method (IBM) and performed the earliest simulation of blood flow in the heart on a fixed Cartesian mesh. IBM is still one of the most popular methods for examining FSI, but the diffusive interface treatment of this method decreases the numerical accuracy. Hence, recent FSI simulations of the heart and large arteries have used moving mesh methods to impose interfacial boundary conditions more accurately.

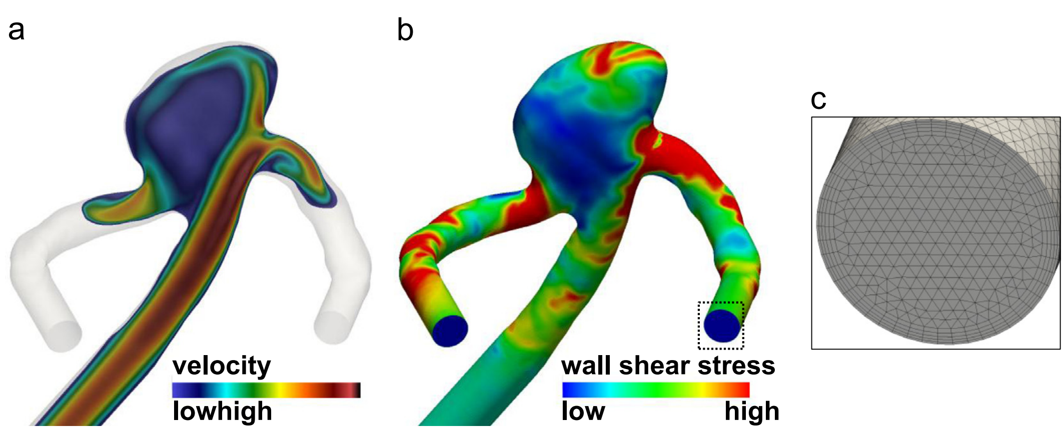

Fig. 1. Numerical simulation of blood flow in a patient-specific cerebral artery with an aneurysm using a boundary-fitted mesh. (a) Velocity. (b) Wall shear stress. (c) Fluid mesh in a cross-section. The ethics committee at National Hospital Organization Kyoto Medical Center, Kyoto, Japan approved use of the patient's medical image data.

Please cite this article as: Imai, Y., et al., Numerical methods for simulating blood flow at macro, micro, and multi scales. Journal of Biomechanics (2015), http://dx.doi.org/10.1016/j.jbiomech.2015.11.047