Journal of Computational Physics 254 (2013) 200–218

Contents lists available at ScienceDirect

Journal of Computational Physics

www.elsevier.com/locate/jcp

A novel multiblock immersed boundary method for large eddy simulation of complex arterial hemodynamics

Kameswararao Anupindi *, Yann Delorme, Dinesh A. Shetty [1], Steven H. Frankel

School of Mechanical Engineering, Purdue University, West Lafayette, IN 47907, USA

ARTICLE INFO

Article history:
Received 18 February 2013
Received in revised form 25 June 2013
Accepted 22 July 2013
Available online 7 August 2013

Keywords:
Large eddy simulation
WENO
Multiblock
Immersed boundary method
High-order finite difference
Incompressible
Biomechanical flows

ABSTRACT

Computational fluid dynamics (CFD) simulations are becoming a reliable tool to understand hemodynamics, disease progression in pathological blood vessels and to predict medical device performance. Immersed boundary method (IBM) emerged as an attractive methodology because of its ability to efficiently handle complex moving and rotating geometries on structured grids. However, its application to study blood flow in complex, branching, patient-specific anatomies is scarce. This is because of the dominance of grid nodes in the exterior of the fluid domain over the useful grid nodes in the interior, rendering an inevitable memory and computational overhead. In order to alleviate this problem, we propose a novel multiblock based IBM that preserves the simplicity and effectiveness of the IBM on structured Cartesian meshes and enables handling of complex, anatomical geometries at a reduced memory overhead by minimizing the grid nodes in the exterior of the fluid domain. As pathological and medical device hemodynamics often involve complex, unsteady transitional or turbulent flow fields, a scale resolving turbulence model such as large eddy simulation (LES) is used in the present work. The proposed solver (here after referred as *WenoHemo*), is developed by enhancing an existing in-house high-order incompressible flow solver that was previously validated for its numerics and several LES models by Shetty et al. (2010) [33]. In the present work, *WenoHemo* is systematically validated for additional numerics introduced, such as IBM and the multiblock approach, by simulating laminar flow over a sphere and laminar flow over a backward facing step respectively. Then, we validate the entire solver methodology by simulating laminar and transitional flow in abdominal aortic aneurysm (AAA). Finally, we perform blood flow simulations in the challenging clinically relevant thoracic aortic aneurysm (TAA), to gain insights into the type of fluid flow patterns that exist in pathological blood vessels. Results obtained from the TAA simulations reveal complex vortical and unsteady flow fields that need to be considered in designing and implanting medical devices such as stent grafts.

1. Introduction

Aortic aneurysm is a local permanent ballooning of the blood filled aorta [20], and is prone to rupture if not treated. Developing reliable rupture risk prediction tools is clinically important to make better aneurysm repair decisions. Knowledge of hemodynamics helps in understanding the factors that lead to initiation and growth of aneurysms. Atherosclerosis, a cardiovascular disease is the primary cause of heart disease and localization of atherosclerosis was shown to be correlated

* Corresponding author.
E-mail address: kamesh.a@gmail.com (K. Anupindi).
[1] Current address: Halliburton, 2107 CityWest Blvd Bldg 2, Houston, TX 77042-2827, United States.

0021-9991/$ – see front matter
http://dx.doi.org/10.1016/j.jcp.2013.07.033

to regions of disturbed flow fields [31]. It was shown that, recirculation zone formed inside the aneurysm promotes thrombus formation and rupture [3]. Computational fluid dynamics (CFD) simulations are becoming a reliable tool to not only understand disease progression in pathological blood vessels, but also design and gauge the performance of several medical device solutions, such as stent grafts and ventricle assist devices [18,8]. Pathological and medical device hemodynamics often involve, transitional or mildly turbulent unsteady disturbed flows with streamline curvature and rotation [37,38]. In order to accurately simulate such flows, a scale resolving turbulence model such as large eddy simulation (LES) is required.

Turbulence modeling based on LES further requires that high-order (greater than 2nd order) methods be used for discretizing and solving the governing equations numerically. However, usage of high-order numerical methods often limits one to use structured grids, which may not be able to handle a variety of complex geometries that arise in arterial flow domains. Immersed boundary method (IBM) emerged as an attractive methodology because of its ability to efficiently handle complex moving and rotating geometries on structured grids. The tedious job of mesh generation for complex flow domains is by-passed in these methods by constructing a global domain containing both the solid and fluid regions. IBM was introduced by Peskin [26], in which the flow field is solved on a Eulerian mesh and the immersed surface is discretized using Lagrangian points and the method was applied to the two-dimensional simulation of flow around a natural mitral valve. IBM simulations can handle moving or deforming bodies with complex surface geometry relatively easily without the need for re-meshing at every time step of the flow simulation as is needed in conventional body-fitted mesh simulations. There have been many works by several authors, in applying IBM to various fluid mechanics problems such as dragonfly flight aerodynamics [23], fish swimming [23,11], human walking as an application of multiple moving immersed objects [5], blood flow in heart [25], fluid–structure interaction of aortic heart valve [21] and turbo machinery [29], to name a few. Certainly, the application list mentioned here is incomplete and the reader is referred to the articles by Mittal et al. [24] and by Peskin et al. [27] to gain a complete insight. Simulations based on IBM can be readily applied to external aerodynamics problems [6,28] where the volume of the solid region is much smaller compared to the fluid region thereby reducing the amount of unnecessary grid. Adaptive mesh refinement (AMR) was used by Vanella et al. [36] as a way of reducing the amount of unnecessary grid and also to increase the resolution only in the regions of interest. Griffith et al. [12] also employed an adaptive, second order accurate IBM to simulate blood flow in heart and great vessels. They achieved enhanced boundary layer resolution in model heart valve by using locally refined mesh methodology. Using AMR one can specifically refine the mesh based on geometric or solution driven parameters.

Although IBM based simulations are quite successful in external aerodynamics problems [24,5,6,28], their applications to internal fluid flow in complex geometries such as blood flow in arteries are scarce. Yokoi et al. [43] used a Cartesian grid approach together with IBM and simulated blood flow in a cerebral artery with multiple aneurysms. They used a 0.6 million Cartesian grid to immerse the cerebral artery. Although, no mention of the percentage of total grid nodes in the fluid region is made in their article, given the ratio of the diameter of the cerebral artery to its lateral extents it is apparent that a large portion of grid nodes were in the exterior of the fluid domain. Delorme et al. [8] performed LES studies of powered Fontan hemodynamics with relatively short vena cavae and long pulmonary arteries in order to reduce the amount of grid nodes lying in the exterior of the fluid domain. However, again given the longitudinal and lateral extents of the total cavopulmonary connection (TCPC) compared to its internal diameter a significant number of grid nodes were located outside of the fluid domain, as was reported in their article. These are few examples of the short comings of IBM directly applied to simulate complex arterial networks. Recently, in an effort to extend IBM to simulate complex arterial geometries, de Zélicourt et al. [7] developed a serial flow solver, using an unstructured Cartesian grid approach and studied blood flow in a real-life TCPC anatomy. As de Zélicourt et al. [7] point out in their article, one possible reason for the scarcity of studies on IBM applied to study blood flow in complex internal flow configurations, could be because of the prohibitive memory and computational demands on the single block grids that arise in order to handle these geometry. Another point that is of concern in handling complex geometries on structured grids is the constraint that all the inflow and outflow boundaries of the geometry have to terminate only on the boundary faces of the global bounding box that encloses both the fluid and solid regions. This requirement could be met in certain cases (as was done in Yokoi et al. [43] and Delorme et al. [8]) by properly truncating the complex geometry and in some cases it is not possible. Such alterations of the complex geometry to make it compatible for single block simulations might result in altering the results obtained compared to the unaltered geometry and sometimes the inflow boundary conditions may not even be known at the altered locations. In order to overcome the aforementioned problems and extend the applicability of IBM to simulate blood flow in complex anatomies, we propose a method based on a combination of multiblock structured grids and IBM on an inherently parallel framework. This particular methodology not only enables simulation of fluid flow in complex geometries but also reduces the amount of unnecessary grid that goes into the solid regions.

The organization of the paper is as follows. In Sections 2 and 3 we present the governing equations, the mirroring immersed boundary method developed by Mark et al. [22] and the multiblock methodology employed in the present work. In Section 4, we perform simulations using *WenoHemo* for three-dimensional laminar flow over a sphere and laminar flow over a backward facing step which validates the IBM and multiblock approaches respectively. Then we validate the combined solver by simulating the laminar and transitional flow in abdominal aortic aneurysm (AAA) and compare them to the experimental results of Asbury et al. [1]. Then we study blood flow in thoracic aortic aneurysm (TAA) which establishes the applicability of the proposed solver to complex arterial geometries.