Numer. Math. Theor. Meth. Appl. Vol. 7, No. 4, pp. 555-574
doi: 10.4208/nmtma.2014.1303si November 2014

Computing Viscous Flow in an Elastic Tube

Yi Li, Ioannis Sgouralis and Anita T. Layton*

Department of Mathematics, Duke University, Durham, NC, USA.

Received 17 April 2013; Accepted 14 February 2014

Available online 11 November 2014

Abstract. We have developed a numerical method for simulating viscous flow through a compliant closed tube, driven by a pair of fluid source and sink. As is natural for tubular flow simulations, the problem is formulated in axisymmetric cylindrical coordinates, with fluid flow described by the Navier-Stokes equations. Because the tubular walls are assumed to be elastic, when stretched or compressed they exert forces on the fluid. Since these forces are singularly supported along the boundaries, the fluid velocity and pressure fields become unsmooth. To accurately compute the solution, we use the velocity decomposition approach, according to which pressure and velocity are decomposed into a singular part and a remainder part. The singular part satisfies the Stokes equations with singular boundary forces. Because the Stokes solution is unsmooth, it is computed to second-order accuracy using the immersed interface method, which incorporates known jump discontinuities in the solution and derivatives into the finite difference stencils. The remainder part, which satisfies the Navier-Stokes equations with a continuous body force, is regular. The equations describing the remainder part are discretized in time using the semi-Lagrangian approach, and then solved using a pressure-free projection method. Numerical results indicate that the computed overall solution is second-order accurate in space, and the velocity is second-order accurate in time.

AMS subject classifications: 65M06, 76D05, 76M20

Key words: Navier-Stokes flow, axisymmetry, immersed boundary, fluid-structure interaction.

1. Introduction

Many biological systems involve viscous flow through contracting or compliant tubes. Such examples include blood flow, food mixing and chyme movement in intestine, transport of spermatozoa in cervical canal, transport of bile in bile ducts, fluid flow through the kidney's collecting duct that is undergoing peristaltic contracts, etc. The interactions between the moving tubular walls and the luminal fluid can give rise

*Corresponding author. *Email addresses:* `yili@math.duke.edu` (Y. Li), `io-sgou@math.duke.edu` (I. Sgouralis), `alayton@math.duke.edu` (A. T. Layton)

to complex fluid dynamics. It is natural to formulate such problems in cylindrical coordinates. To lower computational cost while still capturing the 3D flow features, one may assume axisymmetry.

Numerical approaches for simulating fluid-structure interactions can be grouped into two main categories: conforming approaches, such as the Arbitrary Lagrangian Eulerian (ALE) method [5–7, 18, 19, 23], and non-conforming approaches. A popular non-conforming grid method is the immersed boundary method, which was originally developed in [21] for solving the full incompressible Navier-Stokes equations with moving boundaries, originally for studying blood flow through a beating heart [20]. Moving boundaries are represented by means of Lagrangian markers, at which boundary forces are computed. These forces are then transferred to an underlying Cartesian fluid grid via a regularization using discrete (smoothed) delta functions, and the Navier-Stokes equations are solved on the Cartesian grid.

The immersed boundary method is generally first-order accurate. To improve solution accuracy, especially near the immersed boundary, Li and co-workers developed the immersed interface method [13, 14], which captures jump discontinuities in the solution and its derivatives sharply, and generates approximations with second-order accuracy. The higher accuracy is achieved by incorporating jumps in the solution or its derivatives, which can be computed from boundary forces, into the finite difference schemes. The immersed interface method has been developed for Stokes [11, 14] and Navier-Stokes [16, 24] flows, in Cartesian [14, 16] and polar [10, 17] coordinates.

Recently we developed a numerical method for simulating driven Stokes flows through a compliant closed tube, driven by a pair of internal fluid source and sink [15]. Motivated by models of blood flow through vessels, the problem is formulated in axisymmetric cylindrical coordinates. The method decomposes the pressure and velocity fields into parts due to the tubular boundary force, which is singular, and due to the source and sink, which have compact support. Each part is computed by means of an appropriate method that is second-order accurate, and also efficient given that characteristics of that part of the solution. The singular tubular boundary force induces jump discontinuities in the solution and its derivatives. To compute this part of the solution, we use the immersed interface method, which has not been previously applied to cylindrical coordinates, and for which we derived the jump conditions in axisymmetric cylindrical coordinates. In contrast, the solution due to the fluid source and sink is smooth. If one is interested only in tracking the tubular wall movements, then the source/sink-driven solution can be efficiently calculated along the tubular surface via a boundary integral. The method is second-order accurate and robustly captures the jump discontinuities in the overall solution and its derivatives.

The method developed in [15] is limited to creeping flows or fluid with sufficiently high viscosity. However, many biofluid applications, such as blood flow in arteries and arterioles, have medium to high Reynolds number and are thus more appropriately described as Navier-Stokes flow. In the present study, we aim to extend the method to the Navier-Stokes equations, using the velocity decomposition approach [2]. That approach is motivated by two observations: first, the jump conditions in the solution