Applied Mathematics and Computation 333 (2018) 376–400

Contents lists available at ScienceDirect

Applied Mathematics and Computation

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

Implicit finite element methodology for the numerical modeling of incompressible two-fluid flows with moving hyperelastic interface

Aymen Laadhari

Institut für Bildverarbeitung, Department of Information Technology and Electrical Engineering, Swiss Federal Institute of Technology Zürich (Eidgenössische Technische Hochschule Zürich, ETH-Zentrum), Zürich CH-8092, Switzerland

ARTICLE INFO

Keywords:
Fluid-structure interaction
Hyperelastic membrane
Incompressible flow
Newton
Monolithic
Finite element method

ABSTRACT

We present a numerical methodology based on the use of the Newton and level set methods and tailored for the simulation of incompressible immiscible two-fluid flows with moving hyperelastic membrane. The method features the use of implicit time integration schemes and is based on a consistent Newton–Raphson linearization. The performances are enhanced by using the Kou's method (Kou et al., 2006) which features a third-order convergence behavior without requiring higher order derivatives. To overcome numerical instability issues related to the explicit decoupling, a fully monolithic strategy and a partitioned implicit strategy are devised. We investigate the main features of the proposed strategies, and we report several numerical experiments with the aim of illustrating their robustness and accuracy. We show numerically that the monolithic strategy performs better and remains stable when considering relatively small viscosities or large stiffness, for which the partitioned approach depicts a slow convergence or even fails to converge. However, the partitioned strategy features significant computational savings when it converges within a reasonable number of sub-iterations.

1. Introduction

This computational framework is concerned with the modeling of thin elastic structures immersed in incompressible Newtonian flows. The coupled dynamics of fluids and lightweight elastic membranes with small thickness compared to the membrane dimensions is of significant interest in many real life phenomena and industrial applications, including amongst which biomedicine, aeronautics and structural engineering. In recent years, this problem has generated a lot of interest from different scientific communities, and researches have covered the fields of theoretical biology [2], mechanics [3], physics [4,5], biomedical engineering [6] and applied mathematics [7].

Among a broad spectrum, let us first list two applications we have in mind for this kind of problems. (i) A cardiovascular application is the dynamics of the aortic valve at the exit of the left ventricle. It is composed of three leaflets that undergo fast movement and large deformations during the opening and closing phases, allowing thereby the correct blood ejection and preventing regurgitation [8]. Typically, the leaflets thickness is very small (about 0.5 mm) compared to the dimensions of the valve (height ≈ 18 mm and aorta diameter ≈ 26 mm) [9,10]. (ii) A second target application is the dynamics of biomembranes such as red blood cells (RBCs) and capsules. A RBC has approximately the dimension $8 \times 2\,\mu$m

E-mail addresses: Aymen.Laadhari@math.ch, laadhari@vision.ee.ethz.ch

https://doi.org/10.1016/j.amc.2018.03.074
0096-3003/

and consists of thin lipid bilayer lacking nucleus and organelles with 4/5 nm-thickness juxtaposing a filamentous meshwork of proteins called cytoskeleton of about 60/80 nm-thickness and enclosing a hemoglobin fluid [5,7,11,12]. The RBC's membrane can be considered as a two-dimensional structure suspended in blood and endowed with hyperelastic properties. Unlike RBCs, capsules are deformable phospholipid membranes with a small resistance to stretching. They are found in nature or are synthesized e.g. for cosmetic, industrial, biomedical and pharmaceutical applications on grounds of their mechanical properties and simple structures. In biomedical applications, they are manufactured to allow drug delivery in the body. When suspended in flow, RBCs and capsules are exposed to viscous stresses that can lead to large deformations of the membrane. It is fundamental to model their mechanical properties, as they enable to establish the resistance of the membrane to the applied stresses.

A fluid-structure interaction problem involving a thick structure requires the simultaneous solution of the fluid and structure problems in such a way that the coupling is achieved by enforcing the kinematic conditions describing the continuity of the velocity and the dynamic conditions describing the balance of stresses at the interface. Further difficulties are encountered when extremely thin structures (membranes) are involved. Throughout the literature, the existing methods can be classified based on the framework used to represent the structure in the fluid domain and how the coupling is performed. The most popular approaches are the Arbitrary Lagrangian Eulerian (ALE) methods and the Immersed Boundary Methods (IBM) [13]. (i) The standard ALE approaches consist in the employment of a body-fitted moving mesh that follows the structural deformations. The fluid and structural meshes remain contiguous, while the compatibility conditions are applied at the shared interface. Different Lagrangian and Eulerian representations are traditionally used for the solid and fluid problems, respectively. ALE-type approaches have been successfully employed in problems with relatively small structural deformations. Efficient and robust remeshing tools are required to maintain high quality mesh elements during the simulation, leading thereby to a substantial computational burden. Major concerns are additionally related to the large structural deformations and the multitude of interpolations between different frameworks. (ii) To address the remeshing issues, several approaches have been developed using fixed grids such as the IBM extensively used in this context [14]. The driving idea is that a Lagrangian description is used for the elastic structure (nonconforming mesh), while keeping a separate Eulerian model for the fluid [15–17]. A common domain is then occupied by both the fluid and structure, while additional forcing (i.e. body or singular surface force) is applied to the fluid problem in the surrounding of the solid domain. Several IB-type approaches have been devised, see e.g. [18–22], the finite difference method [23], the immersed finite element method [24], and the fictitious domain method [25]. We further cite other available strategies using the finite element method [12,26,27] the boundary-element method [5,28,29], and the extended finite element method [30]. The IB-type methods seem to be more appropriate than ALE methods when dealing with large structural deformations. However, an appropriate formulation of the solid mechanics and the coupling conditions is required, while the price to be paid is the loss of accuracy due to the regularization procedure [20,31]. They also allow for semi-implicit time schemes at the price of a stability condition that can be severe depending on the physical parameters of the problem [31,32].

The convergence and stability can be a big concern and relies importantly on the techniques used to enforce the coupling conditions. We can distinguish between monolithic and partitioned approaches. (i) Monolithic approaches take into account the coupling into a single system, which is solved using a single solver [33]. (ii) Partitioned approaches solve the two-way fluid/solid coupled problem in a sequential manner. Hence, separate dedicated solvers involving different frameworks for the fluid and solid are used, and the coupling is performed at the interface level. The advantage of this approach is its modularity and the possibility of re-using standard solvers. (ii-1) Performing one or few iterations at each time step leads to a loosely-coupled approach and an explicit decoupling scheme. That can exhibit numerical instabilities due to the artificial added mass effect which results from the effect of an extra mass the fluid force exert on the structural interface degrees of freedom [34]. The numerical difficulties are exacerbated for problems involving extremely slender and lightweight structures in an incompressible fluid [35]. (ii-2) To circumvent instabilities inherent in loosely-coupled schemes, the strongly-coupled approaches represent somewhat a remedy that requires an iterative process at each time level until the convergence is achieved.

The central aim of the present paper is to present a stable computational framework to solve problems involving an elastic membrane in an incompressible fluid. The membrane is tracked using a level set method. This Eulerian framework allows a unified representation and a unique solver for the membrane and fluid. To address stability issues due to the time step restrictions when using explicit methods, we present fully implicit strategies based on the use of an exact Newton method. To improve the convergence properties, we consider a generalization to the multidimensional case of the Kou's variant [1,36], which features a faster convergence behavior without computing higher order derivatives. To our knowledge, such Newton variants have not been investigated for the present problem. We propose both a fully monolithic approach and a strongly-coupled approach, and we investigate their stability properties and the corresponding computational costs.

We have arranged the remainder of the paper as follows. Section 2 presents the mathematical formulation of the problem. Section 3 focuses on the numerical method and describes in detail the fully monolithic and the strongly coupled partitioned approaches. The two approaches use the cubically-convergent Kou's method. In Section 4, we report several numerical examples to illustrate the efficiency and the robustness of the proposed methods. We close with some comments about the model limitations and forthcoming extensions in Section 5.