Journal of Computational Physics 300 (2015) 814–843

Contents lists available at ScienceDirect

Journal of Computational Physics

www.elsevier.com/locate/jcp

A numerical approach for simulating fluid structure interaction of flexible thin shells undergoing arbitrarily large deformations in complex domains

Anvar Gilmanov [a], Trung Bao Le [a], Fotis Sotiropoulos [a,b,*]

[a] Saint Anthony Falls Laboratory, University of Minnesota, Minneapolis, MN 55414, United States
[b] Department of Civil, Environmental and Geo-Engineering, University of Minnesota, Minneapolis, MN 55414, United States

ARTICLE INFO

Article history:
Received 13 November 2014
Received in revised form 19 May 2015
Accepted 4 August 2015
Available online 11 August 2015

Keywords:
Fluid–structure interaction
Immersed boundary method
Finite element
Thin shells
Rotation-free approach

ABSTRACT

We present a new numerical methodology for simulating fluid–structure interaction (FSI) problems involving thin flexible bodies in an incompressible fluid. The FSI algorithm uses the Dirichlet–Neumann partitioning technique. The curvilinear immersed boundary method (CURVIB) is coupled with a rotation-free finite element (FE) model for thin shells enabling the efficient simulation of FSI problems with arbitrarily large deformation. Turbulent flow problems are handled using large-eddy simulation with the dynamic Smagorinsky model in conjunction with a wall model to reconstruct boundary conditions near immersed boundaries. The CURVIB and FE solvers are coupled together on the flexible solid–fluid interfaces where the structural nodal positions, displacements, velocities and loads are calculated and exchanged between the two solvers. Loose and strong coupling FSI schemes are employed enhanced by the Aitken acceleration technique to ensure robust coupling and fast convergence especially for low mass ratio problems. The coupled CURVIB-FE-FSI method is validated by applying it to simulate two FSI problems involving thin flexible structures: 1) vortex-induced vibrations of a cantilever mounted in the wake of a square cylinder at different mass ratios and at low Reynolds number; and 2) the more challenging high Reynolds number problem involving the oscillation of an inverted elastic flag. For both cases the computed results are in excellent agreement with previous numerical simulations and/or experiential measurements. Grid convergence tests/studies are carried out for both the cantilever and inverted flag problems, which show that the CURVIB-FE-FSI method provides their convergence. Finally, the capability of the new methodology in simulations of complex cardiovascular flows is demonstrated by applying it to simulate the FSI of a tri-leaflet, prosthetic heart valve in an anatomic aorta and under physiologic pulsatile conditions.

1. Introduction

Fluid–structure interaction problems involving three-dimensional, thin, flexible structures are encountered in a broad range of problems in engineering and biology. Examples include, among others, inflating parachutes, energy harvesting

* Corresponding author.
E-mail addresses: agilmano@umn.edu (A. Gilmanov), lebao002@umn.edu (T.B. Le), fotis@umn.edu (F. Sotiropoulos).

http://dx.doi.org/10.1016/j.jcp.2015.08.008

devices, swimming aquatic organisms, and native and prosthetic heart valves. Such problems occur across a broad range of Reynolds numbers and flow regimes, often take place in geometrically complex domains, and commonly involve arbitrarily large deformations of the flexible structures. These attributes, and especially the highly non-linear nature of the ensuing FSI, present unique challenges to numerical methods for simulating such problems. Such challenges arise from, among others, *i)* the need to model geometric and hyperelastic non-linearities of the solid bodies; *ii)* the complexity of the flow domains and the arbitrarily large amplitude of the deformation thin flexible structures may undergo; and *iii)* the challenges in obtaining robust and efficient FSI algorithms especially in problems with low mass ratios [1,2], which are commonly encountered in heart valve simulations. These challenges along with approaches that have previously been developed in the literature for tackling them are discussed in more detail below.

There are two general approaches for simulating complex flows with deformable boundaries: the boundary conforming *Arbitrary Lagrangian Eulerian (ALE)* approach and *Immersed Boundary (IB)* methods. The ALE approach [3,4] is well suited for simulating high Reynolds number flows due to its inherent body-fitted mesh structure that conforms to boundaries at all times. However, ALE methods are cumbersome to apply to problems with large deformations since they require frequent remeshing in order to prevent the mesh from becoming severely distorted when large deformation develop. The remeshing procedure is computationally expensive making ALE methods difficult to apply in complex three-dimensional problems. Fixed, non-boundary conforming, grid methods provide another alternative to solving problems with deformable boundaries and complex geometry. Such methods are generally referred to as immersed boundary (IB) methods and are especially attractive for simulations of complex flows in engineering and biology because they do not require remeshing and can readily handle arbitrarily large deformations. The various types of IB methods have been recently reviewed by Sotiropoulos and Yang [1]. The interested reader is referred to this paper as well as the earlier review by Mittal and Iaccarino [5] for details. Here we focus our literature review exclusively on numerical approaches proposed for handling FSI of flexible structures in complex domains. In our review, we distinguish and pay special attention to the discretization techniques used to handle the flow and structural governing equations since a range of formulations have been proposed. These include pure finite-difference (FD) [6–10] or finite-element (FE) [11–13] methods for both the flow and structural equations as well as mixed formulations combining FD (or finite volume) discretization for the flow with FE for the structural equations [14,15].

Diffused-interface IB methods use finite-differencing for both the fluid and structural solvers [6] due to the simplicity in obtaining the loading condition on the structural surface by incorporating appropriately defined body forces in the governing equations. A number of successful applications of such IB-FD-FD methods have been reported over the years [8,9,7]. The accuracy of such methods can be improved by incorporating local mesh refinement as was done in [6]. One potential difficulty with this class of methods, however, arises from treating the solid surface as diffused interface, which complicates the calculation of the details of the wall shear stress field on the surface. Yet, such detailed calculations may be required in heart valve flow simulations, where complex wall shear stress patterns on the valve leaflets have been linked with increased potential for aortic valve disease [16].

A sharp-interface IB method using FD formulations for both the flow and structural equations was proposed by Luo et al. [10]. This method was formulated for solving linear viscoelastic solids and applied to simulate two-dimensional FSI in laryngeal aerodynamics. The same formulation was later modified to incorporate a FE formulation for the structural equations and applied to simulate FSI of a high mass ratio 3D flapping wing at very low Reynolds number ($Re = 50$) [17]. Tian et al. [18] further extended this method to simulate several complex FSI problems at low Reynolds numbers ($Re \sim 10^2$). An ALE formulation utilizing a so-called embedded boundary approach was proposed by Farhat and K. Lakshminarayan [15] for solving compressible FSI problems for external aerodynamics applications at high Reynolds numbers. This approach employs finite volume discretization for the fluid equations with finite elements for the structural equations. While this approach can work well for structures in unbounded domains, remeshing difficulties may arise when the structure is embedded within a complex confined domain. A pure finite-element based formulation, for both the flow and the structural equations, was recently proposed by Kamensky et al. [19]. This method employs the *immersogeometric FSI* approach and was applied to simulate FSI of a bioprosthetic heart valve in a straight aorta.

In FSI simulations of biological tissues, e.g. heart valve leaflet interaction with blood flow, it is critical to use a relevant and efficient structural model that is able to realistically represent the deformation of the tissue under loads imposed by the pulsatile blood flow. Such undertaking, however, is not a trivial task since the large deformations of the tissue and its underlying geometric non-linearity pose major modeling challenges. To circumvent these challenges recent studies attempting to simulate FSI of tissue valves chose to either use simplified membrane-like materials [20] or treat the valve leaflets as thick bodies [18]. However, biological tissues of leaflets are normally thin and they exhibit significant bending. Therefore, a shell model for the solid body is a more appropriate choice [21,19]. Most finite-element (FE) methodologies for handling shells, however, are computationally very demanding as they employ two or three nodal rotations alongside with three nodal translations, i.e. 5 or 6 degree of freedom per node. An exhaustive review of this large body of literature is beyond the scope of this paper but the reader is referred to a number of recent review papers on the topic [22,23]. Note that the efficiency of the FE shell model becomes of paramount concern in FSI simulations of complex problems where the need to couple the fluid and structural solvers together can dramatically increase the computational cost per time step. For that, in this work we adapt and incorporate in the FSI methodology a previously developed nonlinear, rotation-free triangular shell element formulation [24], which has already been shown to provide accurate and robust solutions of various thin shell FE problems. Such an approach, however, has not been coupled before with a flow solver to simulate FSI problems and it is this coupling that constitutes one of the important contributions of our work.