

Simulating Two-Phase Viscoelastic Flows Using Moving Finite Element Methods

Yubo Zhang¹, Heyu Wang^{2*} and Tao Tang¹

¹ Department of Mathematics, Hong Kong Baptist University, Kowloon Tong, Hong Kong.

² Department of Mathematics, Zhejiang University, Hangzhou, Zhejiang 310027, China.

Received 4 February 2009; Accepted (in revised version) 8 June 2009

Available online 24 August 2009

Abstract. Phase-field models provide a way to model fluid interfaces as having finite thickness; the interface between two immiscible fluids is treated as a thin mixing layer across which physical properties vary steeply but continuously. One of the main challenges of this approach is in resolving the sharp gradients at the interface. In this paper, moving finite-element methods are used to simulate interfacial dynamics of two-phase viscoelastic flows. The finite-element scheme can easily accommodate complex flow geometry and the moving mesh strategy can cluster more grid points near the thin interfacial areas where the solutions have large gradients. A diffused monitor function is used to ensure high quality meshes near the interface. Several numerical experiments are carried out to demonstrate the effectiveness of the moving mesh strategy.

AMS subject classifications: 65M05, 76M10, 76T10

Key words: Moving finite-element method, two-phase flow, viscoelastic flow, non-Newtonian flow.

1 Introduction

Modeling and simulating two-phase viscoelastic flows have been challenging both mathematically and technically. There have been many computational techniques developed to tackle the problem, including diffuse-interface methods [1], interface tracking methods [17], level-set methods [16, 20], finite-element methods with adaptive mesh refinements [22] and spectral methods with adaptive mesh redistribution [7]. The governing

*Corresponding author. Email addresses: zybx@msn.com (Y. Zhang), wangheyu@zju.edu.cn (H. Wang), ttang@math.hkbu.edu.hk (T. Tang)

equations require additional constitutive equations for stress tensor and numerical computations require higher mesh resolution around fluid interfaces. Unlike the Newtonian flow whose local stress is proportional to the local strain rate, for polymer fluids the local stress usually depends on its deformation history due to the long chain molecular structure. Therefore an additional constitutive relation between the stress and the strain of flow such as Upper Convected Maxwell (UCM) model by Oldroyd [13] should be coupled into the flow system.

When comes to multi-phase flows, problems arise due to the sharp interface between fluids. The classical jump conditions and surface tension are introduced into the Navier-Stokes equations in which the interface has zero width. Different approaches such as volume-of-fluid (VOF) and level-set method have been developed to handle this problem, see, e.g., [16, 20]. Compared to sharp interface methods, diffuse interface methods are based on a different theoretical model where the interface is treated as a smooth transition region from one phase to another. The original idea can be found in [2, 3], and [1] is a useful review of the diffuse interface model. In [12, 22], diffuse interface model is applied for Newtonian and non-Newtonian flows.

The main challenge for simulating the phase-field evolution is that very fine meshes are needed for resolving thin interfaces. In order to produce physically correct results one needs a very thin interface and it is almost impossible to solve the problem practically when using uniform meshes. In past years, many adaptive mesh techniques have been proposed which can be classified as adaptive mesh refinement methods and adaptive mesh redistribution methods, see [14]. In this work, we will simulate multi-phase flows using a moving mesh method (i.e., adaptive mesh redistribution method in the sense of [14]). In particular, we will use the moving mesh algorithms developed in Li et al. [10, 11] which redistribute mesh nodes based on harmonic mapping. The moving mesh method based on harmonic mapping has been applied successfully to several complex problems including incompressible flow [5, 6], reaction-diffusion systems [15], and dendritic growth [18, 19]. The goal of the moving mesh method is to reduce the computational cost and to enhance the accuracy in resolving the diffuse interfaces.

In this work, the Oldroyd-B model for constitutive relation of viscoelastic fluids is used. The following section will briefly review the Oldroyd-B model for viscoelastic flow and the phase-field model for two-phase flows. Section 3 will describe the coupled equations and the finite-element formulation used in our computations. The moving mesh methods and the corresponding monitor function will be discussed in Section 4 and several numerical tests will be given in Section 5. The last section draws the conclusion with some discussions on the future works.

2 The Oldroyd-B model and the phase-field model

In this section, we will briefly review the governing equations for viscoelastic flows and the phase-field model for multi-phase flows.