Abstract

Matter at small scales is not a continuum. Whenever we are dealing with phenomena which have disparate time and length scales, we have to rely on multiscale modelling approach in order to capture the complete physics. A common scenario in multiphase flow simulations is the formation of thin fluid films in between colliding fluid masses or between a fluid mass and a surface. These films are very thin (O(100nm)) and during the Direct Numerical Simulations (DNS) of multiphase flows it is impractical to resolve their thickness fully due to highly disparate time and length scales. Our approach here is to couple a complex thin film model derived analytically to a finite volume solver (and can also be used for standard interface capturing technique like Volume of Fluid method) so that we can capture the formation and evolution of thin films which come into existence in the sub-grid thickness. In the present work, we have formulated a thin film model where viscous forces, surface tension forces and long-range intermolecular forces play the dominant role. Since the modelled equation is highly stiff in nature and these films in realistic scenarios are spread over a large area, we have developed a parallel solver for solving the resulting set of equations. We propose an algorithm to couple the thin film model to a finite volume solver. First, we study a simple square domain containing a single phase, undergoing a shear flow with periodic flow in the horizontal direction, that is simulated using finite volume method coupled to a complex thin film model. Next, we develop a robust multiphase flow solver using Coupled Level-set Volume of Fluid method as the interface capturing technique and propose an algorithm to couple the rigorously validated multiphase solver to the thin film model.