Euler-Lagrange Modeling of Large-Scale Gas-Liquid Bubbly Flows

R. Sungkorn and J.G. Khinast

Institute for Process and Particle Engineering, Graz University of Technology, A-8010 Graz, Austria

Abstract

In the present paper, an attempt to assess the feasibility of using computational fluid dynamics (CFD) modeling as an engineering tool for simulation of large-scale gas-liquid bubble flows will be discussed. A modeling technique which provides sophisticated level of accuracy in conjunction with reasonable amount of computational expense will be exploited.

In this modeling technique, the continuous liquid phase is simulated using the lattice Boltzmann (LB) scheme developed by Eggels and Somers (*Int. J. Heat and Fluid Flow* **1995**, 16, 357-364) and Derksen and Van den Akker (*AIChE Journal* **1999**, 45, 209-221). The scheme solves the large-scale motions of turbulent flow using the filtered conservation equations, where the Smagorinsky subgrid-scale model has been used to model the effects of the sub-filter scales. The equation of motion on the basis of the Lagrangian approach has been used to model the trajectory of the individual bubbles. The effect of the fluid fluctuation along the bubble trajectory is modeled using the Langevin equation model of Sommerfeld (*Proceedings of the Ninth Symposium on Turbulent Shear Flows* **1993**, 15-1). Collisions between bubble and bubble are described by the stochastic inter-particle collision model according to kinetic theory, introduced by Sommerfeld (*Int. J. Multiphase Flow* **2001**, 27, 1829-1858). It has been found that, the collision model not only dramatically decreases computing time compared to the direct collision method but also provides an excellent scalability on parallel platforms. Furthermore, it was found that the presented modeling technique provides reasonably good agreement with experimental data for mean and fluctuating velocity components.