Numerical simulation of a single bubble suspension in polyol resin

Dongjin Seo, Yun Mee Lim, and Jae Ryoun Youn
School of Materials Science and Engineering, Seoul National University, Seoul, 151-742, Korea

Abstract

Dilute bubble suspensions are prepared by introducing carbon dioxide bubbles into polyol resin. The apparent shear viscosity is measured with a wide gap parallel plate rheometer. A numerical simulation for deformation of a single bubble suspended in a Newtonian fluid is conducted by using a finite volume method (FVM) where multigrid algorithms are incorporated. Transient and steady results of bubble deformation were obtained and were in good agreement with experimental results. At high capillary number, viscosity of the suspension increases as the volume fraction increases, while at low capillary number, the viscosity decreases as the volume fraction increases.

Introduction

The rheological behavior of bubble suspensions has been studied intensively because of its practical significance and complexity of the phenomena. By adding gaseous bubbles into a Newtonian fluid, the suspension fluid exhibits non-Newtonian behaviors, such as elastic effects and shear- and time-dependent viscosity (Macosko, 1994).

In this study various suspending fluids were prepared with different volume fraction and bubble radii, and the shear viscosity was measured with a wide-gap parallel plate rheometer. In addition to the experiment, we performed a numerical simulation of single bubble deformation between two shearing parallel plates by using a finite volume method with a multigrid algorithm. Deformed bubble shapes, pressure, and velocity fields were obtained with respect to time. The numerical results were compared with experimental and theoretical results.

Experimental

The suspending fluid used in our experiment is propylene oxide based polyol that is surfactant free. Suspensions are prepared by mechanical mixing after carbon dioxide gas is injected into polyol. After suspensions are prepared, they are transferred to the BROOKFIELD DV-Ⅱ+ viscometer with rotating wide-gap parallel plates and shear viscosity at the imposed shear rate is measured.

Numerical Modeling

A pressure based finite volume method for unstructured meshes that includes the SIMPLE algorithm (Patankar, 1981) was used. Cell-based, co-located storage is used for all physical variables. For treating the moving interface, an explicit high resolution scheme that is similar to the CICSAM method (Ubbink and Issa, 1999) is used. The bubble suspension is modeled as two phases of Newtonian fluids with different viscosities. In addition, multigrid algorithms are incorporated into the numerical code to increase the rate of convergence and reduce the calculation time compared with equivalent single-grid schemes.

Computational domain is filled with two different fluids, suspending fluid and air, and has a moving interface. Assuming an isothermal incompressible Newtonian fluid, general governing equations can be written as follows:

$$\nabla \cdot \mathbf{v} = 0 \qquad (1)$$

$$\rho\left(\frac{\partial \mathbf{v}}{\partial t} + \mathbf{v}\cdot\nabla\mathbf{v}\right) = -\nabla p + \mu\nabla^2\mathbf{v} + \mathbf{f}_\sigma \qquad (2)$$

where ρ is density, t is time, $\mathbf{v}$ is the velocity vector, p is pressure, μ is the viscosity, and $\mathbf{f}_\sigma$ is the force due to surface tension.

Surface tension force is formulated with the continuum surface force (CSF) concept (Brackbill et al., 1993) and given by

$$\mathbf{f}_\sigma = -(\Gamma\nabla f)\nabla\cdot\left(\frac{\nabla f}{|\nabla f|}\right) \qquad (3)$$

where Γ is the surface tension and f is the fractional volume function.

Results and Discussion

Shearing of a bubble between two parallel plates is studied numerically. The upper plate and lower plate move in opposite direction with the same speed of $U/2$. Density of fluid and bubble is set to the same value and the viscosity ratio λ = 0.01. For the system of bubble suspension, capillary number and Reynolds number are defined as follows

$$Ca \equiv \frac{\mu_m \dot{\gamma} r}{\Gamma} = \frac{2U\mu_m r}{\Gamma d} \quad (5)$$

$$\mathrm{Re} \equiv \frac{\rho_m \dot{\gamma} r^2}{\mu_m} = \frac{2U\rho_m r^2}{\mu_m d} \quad (6)$$

Here μ_m is viscosity of matrix, r is bubble radius, d is distance between the plates, and U is 0.1 m/s so that Re is small enough to assume creeping flow. Periodic boundary condition is applied to the left and right side of the computation domain. When the surface tension is large (small Ca), it is difficult to achieve convergence and obtain physically meaningful solutions due to the mesh dependence of the solution. In the case of large surface tension, small wiggles in the surface result in large changes in the solutions. Shapes of the deformed bubble with ϕ = 0.15 at t = 8 s are shown in Fig. 1 for different capillary numbers. As surface tension increases, the bubble resists deforming from its original spherical shape. In the case of Ca = 0.01, the bubble shape is almost the same as the sphere.

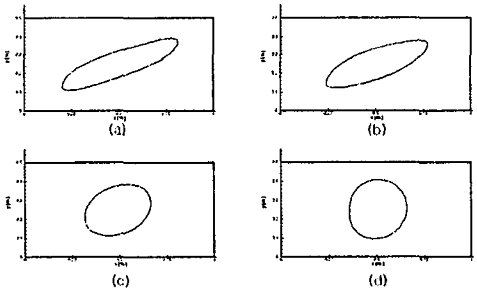

Fig. 1. Shape of the deformed bubble with ϕ = 0.15 at t = 8 s; (a) Ca = 10, (b) Ca = 1, (c) Ca = 0.1, and (d) Ca = 0.01.

Figure 2 shows the relative viscosities with respect to the capillary number. The simulation results are compared with the results from general equation determined based on experimental data and the Frankel and Acrivos equation (1970). For the large capillary number and ϕ = 0.15, the relative viscosity predicted by the numerical simulation is about 0.80, but Frankel and Acrivos equation yields about 0.77. The numerical simulation is based on a two-dimensional approach where the bubble is considered as a long circular cylinder. Furthermore polydispersity of the bubbles and the interactions between bubbles are not considered. Therefore numerical results cannot be compared with experimental results directly. Despite of these limitations, simulation results agree well with the general constitutive equation in high capillary number.

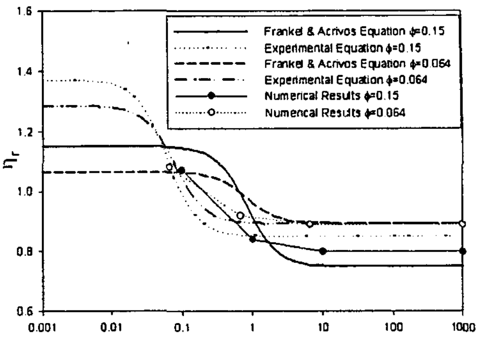

Fig. 2. Effects of capillary number on the relative viscosity when the bubble is slightly deformed.

Conclusions

A numerical simulation was developed and carried out for large capillary number of unsteady region. Although numerical simulation has some restrictions, the numerical results are in good agreement with the experimental data. At low capillary number, increasing bubble volume fraction leads to an increase in viscosity, whereas at relatively high capillary number, viscosity decreases as bubble volume fraction increases. Three dimensional numerical simulation that can consider interactions between bubbles will be developed in the future.

References

1. Brackbill, J. U., D. B. Kothe, and C. Zemach, 1992, A continuum method for modeling surface tension, Journal of Computational Physics, 100, 335.
2. Frankel, N. A. and A. Acrivos, 1970, The constitutive equation for a dilute emulsion, Journal of Fluid Mechanics, 44, 65.
3. Macosko, C. V., 1994, Rheology: Principles, Measurements, and Applications, Willy, New York, 425.
4. Patankar, S. V., 1980, Numerical heat transfer and fluid flow, McGraw-Hill, New York.
5. Seo, D., J. R. Youn, and C. L. Tucker III, 2003, Numerical simulation of mold filling in foam reaction injection molding, International Journal for Numerical Methods in Fluids, 42, 1105.
6. Ubbink, O. and R. I. Issa, 1999, A Method for Capturing Sharp Fluid Interfaces on Arbitrary Meshes, Journal of Computational Physics, 153, 26.