On the Use of SPH Method for Simulating Gas Bubbles Rising in Viscoelastic Liquids

Mohammad VAHABI and Kayvan SADEGHY

School of Mechanical Engineering, College of Engineering, University of Tehran, Center of Excellence in Design and Optimization of Energy Systems (CEDOES) Tehran, Iran, P.O. Box: 11155-4563

(Received: June 10, 2014)

The upward motion and deformation of a single large-scale two-dimensional planar gas bubble rising in an otherwise quiescent viscoelastic liquid obeying the Oldroyd-B rheological model is numerically investigated using the weakly-compressible smoothed particle hydrodynamics (WC-SPH) method. It is shown that unlike the incompressible version of this mesh-less method (I-SPH), the WC-SPH method combined with the continuum surface force (CSF) idea for modelling surface tension is well capable of capturing the cusped-shape trailing edge during its rise in viscoelastic liquids. The failure of the I-SPH method for predicting the cusp is attributed to the several stages of weighted arithmetic and harmonic interpolations used for smoothing the transport parameters of the constituents. The WC-SPH code developed in the present work has enabled us to obtain results at high density and viscosity ratios not reported in similar works. The results suggest that WC-SPH is better suited than I-SPH for handling bubble rise problem in viscoelastic liquids even at large density and viscosity ratios typical of underwater explosions.

Key Words: Two-phase flow / Bubble rise / WC-SPH method / CSF model / Oldroyd-B model

1. INTRODUCTION

The rise and deformation of gas bubbles in liquids are encountered in a variety of industrial and physiological systems. One can mention, for example, underwater explosion, polymer foaming, and blood aeration, among others.15 Several experimental and theoretical studies have been carried out in the past1-6 in order to figure out the parameters affecting bubble’s shape during its rise. In the majority of cases, the liquid surrounding the bubble has been assumed to be Newtonian. However, as is well-established in the literature, most fluids of industrial/physiological importance are known to be more or less non-Newtonian, with viscoelasticity being a common effect.7 To that end, the rise of gas bubbles in polymer solutions has been the subject of several experimental studies in the past.8-10 Through such studies it has already been established that in viscoelastic liquids rising bubbles develop a cusp at their trailing edge. Also, a negative wake has been shown to form behind bubbles rising in polymer solutions.8-10

Tracking large bubbles during its rise in non-Newtonian fluids is not an easy experimental task (particularly over large distances). Thus, it should not be surprising that numerical simulations have played a key role in furthering our knowledge in this area. Several grid-based Eulerian and Lagrangian methods have successfully been used in the past for this purpose.11-13 During the last couple of decades, mesh-less methods such as smoothed particle hydrodynamics (SPH) have been used with great success for this purpose—thanks to their ease of tracking interfaces undergoing large deformations.14 In its original form, the method treats fluids (even liquids) as partially-compressible material through invoking an artificial equation of state—the so-called WC-SPH method. A strictly incompressible variation of the method—the so-called I-SPH method—was later proposed for dealing with two-phase flows in which instead of an equation of state, the Poisson equation is used for obtaining the pressure field. It should be noted that since 1997 a large number of fluid dynamics problems such as fluid-solid interaction15, Rayleigh-Taylor instability16, multi-phase flow17, and free surface flows of Newtonian18 and viscoelastic fluids19 have been solved by the SPH method. In the same vein, the SPH method has successfully been used for simulating the bubble rise problem in Newtonian fluids.20-23
As to the bubble rise problem in non-Newtonian fluids, Zainali et al.\textsuperscript{26} appear to be the only researchers who have relied on the SPH method for this purpose. Using the incompressible version of this robust numerical method, they simulated bubble’s shape during its rise in a viscoelastic fluid obeying the Oldroyd-B model. They have been able to successfully capture the negative wake as observed in experimental studies. However, they could not capture the cusped-shape trailing edge for bubbles rising in viscoelastic liquids. Also, they could not obtain numerical results for density and viscosity ratios larger than 10. (Small density and viscosity ratios are not of much interest for most situations of practical interest.) Surprisingly, however, Zainali et al.\textsuperscript{26} did not address the source(s) of such discrepancies/shortcomings in their work. In the present work, the problem of bubble rise in an Oldroyd-B viscoelastic liquid is revisited with an objective to fill the above gaps. It will be shown that by incorporating an appropriate equation of state in the SPH method (i.e., by relying on the WC-SPH method instead of the I-SPH method) the cusped-shape can be predicted at the trailing edge for bubbles rising in viscoelastic liquids. It will also be shown that obtaining numerical results at high density and viscosity ratios (typical of gas/liquid systems encountered in the real world) is not a problem with the WC-SPH method.

To reach its objectives, the paper is organized as follows: in the next section, the equations governing the rise of a bubble in a viscoelastic fluid obeying the Oldroyd-B model are presented. The next three sections then describe the details of the proposed WC-SPH method encompassing the physical boundary conditions and the technique of parallel-processing used to obtain the numerical results within a reasonable time frame. Typical numerical results are then presented addressing the effect of the fluid’s elasticity on the shape of a rising bubble. The paper is concluded by highlighting its main findings.

2. GOVERNING EQUATIONS

In this study, the buoyancy-driven rise of a gas bubble is simulated in a polymer solution which obeys Oldroyd-B liquid model as its constitutive equation. For ease of analysis, and also for code-verification purposes, the bubble is assumed to be planar initially and remain planar during its rise. Fig. 1 shows schematically the two-dimensional rectangular column filled with the viscoelastic fluid of interest. As can be seen in this figure, the column includes a planar gas bubble which itself is assumed to be Newtonian. The equations governing the flow induced in the surrounding liquid are the conservation of mass and momentum which must be solved over the whole computational domain. The bubble is immiscible with the surrounding fluid so that a force due to the interfacial tension must be included in the analysis. Based on the CSF method, this force can be treated like a volumetric force in a thin layer around the interface.\textsuperscript{25} In Lagrangian framework, the conservation of mass and momentum are as follows:

\[
\frac{Dp}{Dt} = -\rho \nabla \cdot \mathbf{V} \tag{1}
\]

\[
\rho \frac{D\mathbf{V}}{Dt} = -\nabla p + \mathbf{V} \cdot \tau + \mathbf{\hat{f}}_\sigma + \rho \mathbf{g} \tag{2}
\]

where \(\frac{D}{Dt}\) stands for the material derivative, and \(\rho, \mathbf{V}, p, \tau,\) and \(g\) are the density, time, velocity vector, pressure, stress tensor, and the acceleration of gravity, respectively. Also, \(\mathbf{\hat{f}}_\sigma\) is the force due to interfacial tension. Based on the CSF method, this surface force can be treated like a body force so that we have\textsuperscript{25},

\[
\mathbf{\hat{f}}_\sigma = \sigma \kappa \delta \mathbf{n} \tag{3}
\]

where \(\sigma, \kappa, d,\) and \(\mathbf{n}\) are the surface tension coefficient, curvature of the interface, Dirac’s delta function (concentrated on the interface), and unit outward normal vector of the interface, respectively. In order to avoid direct calculation of the curvature (which is prone to numerical error), in the present work an extension of the CSF is used in which the surface tension force is considered as the divergence of a tensor called the “capillary tensor”. That is, we substitute \(\mathbf{\hat{f}}_\sigma = -\mathbf{V} \cdot \mathbf{T}\) where \(\mathbf{T}\) is the capillary tensor defined by\textsuperscript{26},

\[
\mathbf{T} = -\sigma (\mathbf{I} - \delta \hat{\nabla} \delta) \tag{4a}
\]

where \(\mathbf{I}\) is the identity tensor. Defining \(C\) as the phase index (which is set equal to “zero” for one of the phases and equal to “one” for the other phase), the unit outward vector and Dirac’s delta of the interface could be obtained based on the phase index gradient as \(\mathbf{\hat{V}} C / |\mathbf{\hat{V}} C|\) and \(|\mathbf{\hat{V}} C|\), respectively. Thus \(\mathbf{T}\) is calculated as\textsuperscript{26},

\[
\mathbf{T} = -\sigma \left( \frac{\mathbf{\hat{V}} C^2 - \mathbf{\hat{V}} C \otimes \mathbf{\hat{V}} C}{|\mathbf{\hat{V}} C|} \right) \tag{4b}
\]

In this study it is assumed that both fluids are weakly compressible so that the pressure can be related to the density through an appropriate equation of state.\textsuperscript{10} The following equation of state has been shown to be appropriate for this purpose\textsuperscript{20},
where \( c_0 \) is the speed of sound, \( \rho_0 \) is the initial value of the density, and \( \gamma \) is the polytropic constant. In order to limit density variations (say, to lower than 0.01) the speed of sound is assumed to be at least ten times larger than the maximum velocity encountered in the domain.\(^{20} \) As to the stress tensor in Eq. 2, we assume that the liquid surrounding the bubble obeys Oldroyd-B model as its constitutive equation. For this rheological model, the total stress is composed of two parts so that we have\(^{20} \),

$$\tau = \tau_s + \tau \rho$$

(6)

where \( \tau_s \) is the Newtonian solvent’s contribution, and \( \tau \rho \) is the polymer’s contribution to the total stress. For each component, we have\(^{20} \),

$$\tau_s = \mu_s (\nabla \nabla - (\nabla \nabla)')$$

(7)

$$\tau_s + \lambda \left( \frac{D \tau_s}{Dt} - (\nabla \nabla)'. \nabla \nabla - (\nabla \nabla)' \right) = \mu_s (\nabla \nabla)' + (\nabla \nabla)'$$

(8)

where \( \lambda \) is the relaxation time of the solution. The solution itself is assumed to have been obtained by dissolving a polymer (such as PEO) in a Newtonian solvent. In fact, in the above equations \( \mu_s \) and \( \mu_p \) are the solvent and polymer contributions to the total viscosity of the polymeric liquid \( \mu = \mu_s + \mu_p \). Also, if “c” is the polymer concentration (by weight) in the solution, we can write: \( \mu_s = \frac{1}{1+c} \mu_0 \) and \( \mu_p = \frac{c}{1+c} \mu_0 \). It is easy to check that the Oldroyd-B model reduces to the Newtonian fluid model by simply setting \( \lambda \) and \( c \) equal to zero. Now, to close the problem, the above set of governing equations must be solved together with the following kinematic equation,

$$\frac{D \mathbf{x}}{Dt} = \mathbf{V}$$

(9)

where \( \mathbf{x} \) is the particle’s position vector. As the next step, we try to make the governing equations dimensionless. To that end, we substitute

$$x^* = \frac{x}{D_0}, \quad t^* = \frac{t}{\sqrt{D_0/g}}, \quad \mathbf{V}^* = \frac{\mathbf{V}}{\sqrt{gD_0}}, \quad \rho^* = \frac{\rho}{\rho_1}, \quad \mu^* = \frac{\mu}{\mu_1}$$

(10)

where subscript 1 refers to the liquid phase, and \( D_0 \) is the bubble’s initial diameter. Also, the dimensionless gravity acceleration vector, namely \( \mathbf{e}_g \), is achieved by dividing it by its own magnitude. By substituting these dimensionless parameters into Eqs. 1 and 2, we obtain,

$$\frac{D \rho}{Dt} = -\rho \nabla \cdot \mathbf{V}$$

(11)

$$\frac{D \mathbf{V}}{Dt} = -\mathbf{V} + \frac{1}{Re} \mathbf{V} \cdot \nabla + \frac{1}{Bo} \kappa \mathbf{e}_n + \rho \mathbf{e}_g$$

(12)

where we have dropped asterisks above dimensionless parameters for convenience. In these equations, \( Re \) is the Reynolds number, and \( Bo \) is the Bond number defined respectively by,

$$Re = \frac{\rho_1 D_0 \sqrt{gD_0}}{\mu_1}$$

(13a)

$$Bo = \frac{\rho_1 gD_0}{\sigma}$$

(13b)

It is also easy to show that the dimensionless form of Eq. 8 includes the Deborah number which is defined by\(^{30} \),

$$De = \frac{\lambda}{\sqrt{D_0/g}}$$

(14)

As to the boundary conditions required to close the problem, we rely on either the no-slip or the free-slip conditions at the walls of the domain depending on the test case we choose for code-verification purposes. (That is to say that, our code could easily handle both types of boundary conditions when the need arises.) In addition, the fluid is assumed to be initially stress-free.

Fig. 1. Schematic of the flow configuration.
3. NUMERICAL SCHEME

In SPH methods, the physical domain is treated like a bunch of particles (with an initial arbitrary distribution) which transport field properties such as density, velocity, pressure, etc. These field properties are then represented in integral form by means of kernel functions.\textsuperscript{14} The integral approximation of an arbitrary field function, \( f \), at particle’s position, \( i \), depends on its values in the neighboring particle values, \( f_j \), so that we have,

\[
f_i = f(\vec{x}_i) = \int f(\vec{x}_j) W(\vec{x}_i - \vec{x}_j, h) \, \forall_j
\]

where \( W, h, \) and \( \forall \) are kernel function, smoothing length, and volume, respectively. The kernel functions must satisfy specific properties such as delta-function property, utility condition, and compact condition.\textsuperscript{14} Herein, we rely on the normalized Gaussian kernel function for this purpose\textsuperscript{22}:

\[
W(\vec{x}_i - \vec{x}_j, h) = W_\beta = \frac{1}{\pi h^2 (1 - 10 e^{-\alpha})} \left[ e^{-\left(\frac{1}{h^2} \left| \vec{x}_i - \vec{x}_j \right| \right)^2} - e^{-\alpha} \right]
\]

\[
0 \leq |\vec{x}_i - \vec{x}_j| \leq 3h
\]

where the smoothing length is set equal to 1.3 times of the particles’ initial distances. In SPH methods the integral forms of the field functions (as represented by Eq. 15) are converted to algebraic summations over all neighboring particles in a local domain (called the support domain) as follows:

\[
f_i = \sum_j f_j W_\beta \forall_j
\]

It is worth-mentioning that because the kernel function is smooth and has compact support, the derivative of any field function could be written as\textsuperscript{14},

\[
\nabla f_i = \sum_j (f_j - f_i) \nabla W_\beta \forall_j
\]

where the summation must be repeated at each time step due to the particles’ movements. In practice, the governing partial differential equations are converted to a system of ordinary differential equations which must be solved through invoking an appropriate time-integration algorithm. At the end of each time step the particles are moved to their new positions and the whole operation is repeated.

3.1 Two-Phase WC-SPH

In this study, the multi-phase treatment strategy as discussed in Ref.\textsuperscript{22} is adopted with some minor modifications. In this reference, the mass of particles are considered to be constant and the conservation of mass is used to evaluate the specific volume of particles. The equation representing conservation of mass (Eq. 1) then becomes,

\[
\frac{D \rho_i}{Dt} = -\rho_i (\nabla \cdot \vec{V})_i \Rightarrow \frac{D (\rho \vec{V})_i}{Dt} = -\frac{1}{\rho_i} (\nabla \cdot \vec{V})_i
\]

\[
\frac{D \rho_i}{Dt} = -\rho_i (\nabla \cdot \vec{V})_i \Rightarrow \frac{D \rho_i}{Dt} = -\frac{1}{\rho_i} (\nabla \cdot \vec{V})_i
\]

where \( \rho \) is the specific volume. Before using kernel function in Eqs. 17 and 18, we further normalized it to be safe of particles’ deficiency, namely the so-called Sheppard kernel is utilized,

\[
W_b = \frac{W_\beta}{\Gamma_i}, \quad \Gamma_i = \sum_k W_{ik} \forall_k
\]

where \( \Gamma \) is a normalizing factor close to 1 (due to the unity condition of the kernel function). The numerical scheme, which is repeated at each time step, is as follows: First, the density of each particle is evaluated by substituting \( f \) with \( \rho \) in Eq. 17 as,

\[
\rho_i = \sum_j \rho_j W_\beta \forall_j \Rightarrow \rho_i = \sum_{j \in \chi} \frac{\rho_j W_{ij} \forall_j}{\Gamma_j}
\]

\[
\rho_i = \sum_j \rho_j W_\beta \forall_j \Rightarrow \rho_i = \sum_{j \in \chi} \frac{\rho_j W_{ij} \forall_j}{\Gamma_j}
\]

where \( \chi \) is one of the two phases and \( m \) is the particle’s mass. Since the density is discontinuous at the interface, hence the previous summation is only performed over the neighbor particles belonging to the same phase.\textsuperscript{23} Having done that, the pressure of particles are calculated using Eq. 3 (separately for each phase). Next, the pressure gradient at particle \( i \) position is calculated as\textsuperscript{20},

\[
(\nabla p_i) = \sum_j \left[ \frac{p_j}{\Gamma_j} \frac{p_i}{\Gamma_i} \right] \cdot \nabla W_{ij} \forall_j
\]

\[
(\nabla p_i) = \sum_j \left[ \frac{p_j}{\Gamma_j} \frac{p_i}{\Gamma_i} \right] \cdot \nabla W_{ij} \forall_j
\]

Note that this symmetric form of the pressure gradient guarantees that the Newton’s 3rd law is locally preserved. As to the divergence of the velocity vector at each particle’s position, we have,

\[
(\nabla \cdot \vec{V})_i = \sum_{j \in \chi} \left( \frac{\nabla W_{ij} \forall_j}{\Gamma_j} \right)
\]

\[
(\nabla \cdot \vec{V})_i = \sum_{j \in \chi} \left( \frac{\nabla W_{ij} \forall_j}{\Gamma_j} \right)
\]

Next, specific volumes of all particles are updated by integrating Eq. 19,

\[
\frac{D\rho_i}{Dt} = \frac{\rho_i}{\rho_i} (\nabla \cdot \vec{V})_i \Rightarrow \rho_i^{\text{new}} = \rho_i^{\text{old}} + \Delta t \cdot v_i (\nabla \cdot \vec{V})_i
\]
And after that the volume of each particle is updated by manipulating its mass and specific volume. The new values of the stress components are calculated by integrating Eq. 8,

\[ \tau_{p,i}^{\text{new}} = \tau_{p,i}^{\text{old}} + \Delta t \times \left[ (\nabla \nabla)^T \cdot \tau_{p,i} \right]^{\text{old}} \]

(25)

And, finally, the new velocity and position of all particles are evaluated by integrating Eqs. 2 and 9 as follows:

\[ \vec{V}_i^{\text{new}} = \vec{V}_i^{\text{old}} + \Delta t \times \left( -\frac{1}{\rho_i} \nabla P_i + \frac{\vec{V}_i}{\rho_i} + \frac{\vec{F}^n_i}{\rho_i} + \frac{\vec{F}^p_i}{\rho_i} + \vec{g} \right) \]

(26)

\[ \vec{x}_i^{\text{new}} = \vec{x}_i^{\text{old}} + \Delta t \times \vec{V}_i \]

(27)

where we the Newtonian and polymeric forces are calculated as\(^{25}\):

\[ \vec{F}^n_i = \sum_{j \neq k} \frac{2\mu_{ij}}{\rho_i \rho_j} \left( \frac{1}{\Gamma_i} + \frac{1}{\Gamma_j} \right) \left( \vec{x}_i - \vec{x}_j \right)^T \cdot \vec{W}_{ij} \]

\[ \vec{F}^p_i = \sum_{j \neq k} \left( \frac{\tau_{ij}}{\Gamma_i} + \frac{\tau_{ji}}{\Gamma_j} \right) \cdot \vec{W}_{ij} \]

(28)

(29)

In order to calculate the surface tension force, Eq. 5 is used together with an artificial surface tension proposed in Ref. 22. The artificial surface tension is deemed to smooth out the unwanted (numeric) spurious fragmentation of the interface. In practice, the surface tension force is calculated as:

\[ \vec{F}^s_i = \sum_{j \neq k} \left( \frac{T_{ij}}{\Gamma_i} + \frac{T_{ji}}{\Gamma_j} \right) \vec{W}_{ij} \cdot \vec{V}_{ij} + \varepsilon \sum_{j \neq k} \left( \frac{p_i}{\Gamma_i} + \frac{p_j}{\Gamma_j} \right) \vec{W}_{ij} \cdot \vec{V}_{ij} \]

(30)

where \( \varepsilon \) is an adjusting parameter (set equal to 0.01). For time integration, a simple two step predictor-corrector is utilized. Between prediction and correction steps, the particles are artificially displaced as follows to avoid particle clustering problem\(^{24}\):

\[ \vec{x}_i = \vec{x}_i + \alpha \sum_{j=1}^{N} \frac{\vec{x}_j - \vec{x}_i}{|\vec{x}_j - \vec{x}_i|^2} \left( \sum_{j=1}^{N} \frac{\vec{x}_j - \vec{x}_i}{N} \right)^2 \vec{V}_{mn} \Delta t \]

(31)

where \( \alpha \) is an adjusting parameter set equal to 0.007. Since explicit time marching is used, the time step must be limited in accordance to the following criteria\(^{27}\):

\[ \Delta t = 0.25 \times \min \left( \frac{h}{c_g + V_{\text{max}}} \right), \left( \frac{\rho h^2}{2\mu} \right)^{1/2}, \left( \frac{\rho h^4}{2\pi\sigma} \right)^{1/2} \]

(32)

where the first term inside the parenthesis is the well-known Courant-Friedrichs-Levy (CFL) condition \( c_g \) is the numerical sound speed utilized in Eq. 5. The next three terms in the parenthesis have been incorporated in order to control the particles’ movement by viscous diffusion, acceleration of gravity, and surface tension, respectively.\(^{27}\) To avoid clustering problem, particles are initially distributed in a random manner as proposed by Ref. 28; that is:

\[ x_{i,0} = (i - 1) \Delta x + \Delta x/2 + 0.1 \times \text{rand}(-0.5,0.5) \times \Delta x \quad ; \quad i = 1,...,m \]

\[ y_{j,0} = (j - 1) \Delta y + \Delta y/2 + 0.1 \times \text{rand}(-0.5,0.5) \times \Delta y \quad ; \quad j = 1,...,n \]

(33)

### 3.2 Initial and Boundary Conditions

As the initial conditions, it is assumed that at \( t = 0 \) both phases are at rest. Thus, the velocities of all particles are set equal to zero. Also, the viscoelastic liquid surrounding the bubble is assumed to be initially stress-free. The particles’ initial densities are evaluated such that it creates a hydrostatic pressure distribution as described in Ref. 29. As to the boundary conditions, in this study some dummy particles are located out of the domain and in front of each near-boundary inner particles in the mirror position. Pressure, polymeric stress, and density of a dummy particle are dictated from its twin inner particle. Furthermore, to model free-slip condition on the side walls the tangential velocities of these dummy particles are the same as their twins whereas their normal velocities have different signs. At the top and bottom walls of the domain, the no-slip condition is imposed. These dummy particles are generated and destroyed at each time step.\(^{20}\) In practice, the no-slip boundary condition is imposed by negating both of these velocity components. It should be noticed that these dummy particles are generated and destroyed in each time step.

### 3.3 Parallel Processing

A serious limitation of the SPH method (particularly in its dealing with complex problems) is its huge CPU time requirements. To circumvent this problem, we have decided to rely on parallel-processing computations. For this purpose, the MPICH2 open source package is utilized with 4 and 16 cores. In practice, each particle is defined as a structure of its
own data. The problem domain is then partitioned between the CPUs. That is, at the beginning of each time step, particles are distributed between CPUs according to their positions by the zero CPU. Because properties of each particle in a typical CPU depends on its neighboring particles, particles information in a layer having the thickness of the effective influence area (namely 3h) from the neighboring CPUs must be sent to that CPU. These shared particles’ properties are only updated in their own CPU. For the momentum conservation subroutine, some of the updated properties of the shared particles are needed. Thus an additional data transferring between the CPUs is obviously a must. It is found that current parallel-processing algorithm is efficient enough to carry out all the required simulations.

4. RESULTS AND DISCUSSION

4.1 Code-Verification

The SPH code developed in this work is verified using Newtonian results reported in Ref. 5 for the bubble rise problem. To that end, a bubble with an initial dimensionless radius of 0.25 cm is placed at location of (0.5 cm, 0.5 cm) in the rectangular domain shown in in Fig. 1 having a dimension of 1 cm × 2 cm. Following Ref. 5, no-slip condition is applied on the top and bottom walls while free-slip is imposed on the side walls. The physical properties pertinent to the problem have been given in Table I for two different test cases, as used in Ref. 5. (Although in Ref. 5 the system of units has not been mentioned, by trial-and-error we have figured out that they have actually relied on the “cgs” system of units for the physical parameters used in their computations. That is to say that, in Table I densities are in g/cm³, viscosities are in g/cm.s, surface tension is in dyne/cm, and g is in cm/s².) As can be seen in this table, for the first test case the density and viscosity ratios are both equal to 10. Also, for this test case we have: Re = 35 and Bo = 10. In contrast, for the second test case, the density ratio is equal to 1000 and the viscosity ratio is equal to 100. For this test case we have: Re = 35 and Bo = 125. (The second test case is expected to be more challenging than the first one by exhibiting the break-up phenomenon, as will be shown shortly.) The bubble shape and its vertical position at different time steps are then compared with published data reported in Ref. 5 obtained using the TP2D finite-element/level-set code. Fig. 2 shows results obtained for the first test case, and in Fig. 3 we have shown the results for the second test case. As can be seen in these figures, the comparison is very good even though we have relied on just 12,800 particles for obtaining our SPH results. As a matter of fact, even for the second test case, which captures the break-up effect, the SPH code is working remarkably well.

To further verify the code its performance is compared with the performance of the finite-element/level-set code used in Ref. 11 for a bubble rising in an Oldroyd-B viscoelastic liquid. For this comparison, a bubble having an initial radius of 0.5 cm is placed in the column shown in Fig. 1 having a width of 8 cm and a height of 10 cm. The bubble’s center is positioned 2 cm below the bottom and 4 cm from the side walls of the domain. In the “cgs” system of units, the physical properties used for these simulations are: \( \rho_b = 0.2 \text{ g/cm}^3, \mu_b = 200 \text{ g/cm.s}, \rho_l = 1 \text{ g/cm}^3, \) and \( \mu_l = 200 \text{ g/cm.s} \) where subscript “b” refers to the bubble and subscript “l” refers to the liquid. The polymeric liquid surrounding the bubble is assumed to have been obtained by dissolving a water-soluble polymer in water. The polymer concentration is set at 1.0 wt%, and the relaxation time of the liquid is set at 3 s [see Ref. 11]. Also, the surface tension coefficient between the two phases and the acceleration of gravity are set at 10.0 g/s² and 980 cm/s², respectively. Based on this set of parameters, the Re, Bo, and De numbers are equal to 0.16, 98, and 93.9, respectively. Like Ref. 11, the no-slip condition is imposed on all walls of the domain. Fig. 4 shows a comparison between our WC-SPH results with FEM/LS results reported in Ref. 11 at \( t = 3.5 \text{ s} \).

Table I. Physical properties used for the Newtonian test case (in “cgs” system of units).

<table>
<thead>
<tr>
<th>Test Case</th>
<th>( \rho_b )</th>
<th>( \mu_b )</th>
<th>( \rho_l )</th>
<th>( \mu_l )</th>
<th>( \sigma )</th>
<th>( g )</th>
</tr>
</thead>
<tbody>
<tr>
<td>#1</td>
<td>100</td>
<td>1</td>
<td>1000</td>
<td>10</td>
<td>24.5</td>
<td>0.98</td>
</tr>
<tr>
<td>#2</td>
<td>1</td>
<td>0.1</td>
<td>1000</td>
<td>10</td>
<td>1.96</td>
<td>0.98</td>
</tr>
</tbody>
</table>

Fig. 2. A comparison between our Newtonian results with the TP2D results for the first test case in Table I: a) bubble’s shape at \( t = 3 \), b) the vertical position of the bubble’s center of mass.
Note that in this figure, and all subsequent figures, the red symbols represent particles forming the gas side while the blue ones denote the particles representing the surrounding liquid. As can be seen in this figure, the two sets of results are virtually the same. A more stringent test is the vertical length of the bubble during its rise. As can be seen in Fig. 5, the comparison appears to be reasonably good even in this respect (particularly at longer times). As a matter of fact, at short times SPH results appear to be more realistic than the FEM/LS results.

We have also tried to verify the results obtained using our WC-SPH code with results obtained by Zinali et al\textsuperscript{24} using their I-SPH code. To that end, a bubble with an initial radius of 0.3 cm is placed at \( x = 1 \) cm and \( y = 1 \) cm in a rectangular domain (see Fig. 1) having a width of 2 cm and a height of 4 cm. The physical properties are set at \( \rho_b = 0.1 \) g/cm\(^3\), \( \mu_b = 1.025 \) g/cm.s, \( \rho_l = 1 \) g/cm\(^3\), \( \mu_l = 10.25 \) g/cm.s. The surrounding fluid obeys the Oldroyd-B model and its relaxation time and polymeric concentration are set at 0.2 s and 13.286 wt\%, respectively [see Ref. 24]. Also, the surface tension coefficient between the two phases and the acceleration of gravity are set at 10.0 g/s\(^2\) and 980 cm/s\(^2\), respectively. Based on these parameter settings, the Re, Bo, and De numbers are equal to 1.419, 35.28, and 8.083, respectively. Again, no-slip condition is applied at all walls, as used in Ref. 24.

Fig. 6 shows a comparison between the results obtained using our WC-SPH code and the I-SPH code in Ref. 24 at \( t = 0.13 \) s. As can be seen in this figure, while our WC-SPH code could easily capture the cusped-shape trailing edge, the I-SPH code fails to predict this well-known effect. Surprisingly, however, there is no mentioning of this shortcoming of the I-SPH code in Ref. 24. With the speculation that the failure of

\begin{figure}
\centering
\includegraphics[width=\textwidth]{fig3}
\caption{A comparison between our Newtonian results with the TP2D results\textsuperscript{5} for the second test case in Table I: a) bubble’s shape at \( t = 1.8 \) s, b) bubble’s shape at \( t = 2.4 \) s, c) bubble’s shape at \( t = 3.0 \) s, d) vertical position of the bubble’s center of mass.}
\end{figure}

\begin{figure}
\centering
\includegraphics[width=\textwidth]{fig4}
\caption{A comparison between our WC-SPH results (left figure) for the shape of bubble rising in an Oldroyd-B liquid with FEM/LS results reported in Ref. 11 (right figure) at \( t = 3.5 \) s.}
\end{figure}

\begin{figure}
\centering
\includegraphics[width=\textwidth]{fig5}
\caption{A comparison between our WC-SPH results for the vertical length of the bubble rising in an Oldroyd-B liquid with FEM/LS results reported in Ref. 11 at various times.}
\end{figure}
the I-SPH code in predicting the cusp might have been caused by the fact that Zinaí et al.\textsuperscript{24} have modelled surface tension in a different way, we have resorted to the same idea as used in Ref. 24 for modelling the surface tension. To our surprise, however, our WC-SPH code was still able to predict the cusp at almost the same time step.\textsuperscript{30} Therefore, it appears to us that the main culprit for the failure of the I-SPH method as used in Ref. 24 for predicting the cusp is related to the several stages of smoothing used for handling the transport properties. That is to say that, the combined usage of the weighted arithmetic and harmonic interpolations used for smoothing the transport parameters of the constituents appear to be the main culprit for the odd results reported in Ref. 24 in relation to the cusp. The cusp (shown more clearly in Fig. 7) is a real effect for elastic liquids and its capture is mandatory for robust numerical methods. The cusp, which is always accompanied by a negative vortex beneath the bubble’s trailing edge (see Fig. 7.b), is often attributed to the generation of large normal stresses in the vicinity of the bubble close to its trailing edge.\textsuperscript{11}

Numerical results obtained using our WC-SPH code shows that this is indeed the case, as can be seen in Fig. 8.

### 4.2 Large Density and Viscosity Ratio Results

The results reported in Ref. 24 (and also in Refs. 11-13) are limited to small density and viscosity ratios, apparently due to numerical instability. In practice, however, results obtained at large density ratios are more common in real-world situations. As such, obtaining numerical results at large density ratios is of prime importance for viscoelastic liquids. To see if our WC-SPH code can work at large density ratios, we conducted simulations for the same domain and parameter setting as used in Ref. 11 with the only difference being that the bubble’s density and viscosity were changed from 1 g/cm\textsuperscript{3} and 200 g/cm.s to 0.001 g/cm\textsuperscript{3} and 1 g/cm.s, respectively (while the liquid’s properties remained the same). As can be seen in Fig. 9, due to an increase in the density ratio (from 10 to 1000) this new bubble moves much faster than the previous one. Also, the cusped shape is predicted to emerge sooner (say, at \( t = 0.45 \) s).

### 4.3 Parameter Study

Although the effect of relaxation time and polymer concentration have been addressed in Ref. 12 using FEM/LS method, such simulations have not previously been performed using SPH methods.\textsuperscript{24} As such, we have decided to investigate the performance of our developed SPH algorithm in figuring out the effect of these two parameters on the bubble dynamics. The physical properties and the flow geometry are the same as that used in Ref. 11. Figure 10 shows the effect of the
relaxation time, $\lambda$, on the bubble’s shape obtained for $c = 1.0$ wt%. As can be seen in this figure, by increasing $\lambda$ from 0.75 s to 12 s the bubble’s rise speed is slightly increased, as manifested by its vertical position. Also, the bubble tends to retain its initial circular shape by an increase in the relaxation time. These results are in qualitative agreement with those reported in Ref. 12. It can be shown that [see Ref. 30] the maximum of the longitudinal polymeric stress is decreased from 170.5 g/cm$^2$s$^2$ to 55.3 g/cm$^2$s$^2$ when the relaxation time is increased from 0.75 s to 12 s. Obviously, the surrounding liquid has more time to relax the induced stress by an increase in the relaxation time.

Fig. 11 shows the effect of ‘$c$’ on the bubble’s shape obtained at $t = 1.5$ s obtained for $\lambda = 3$ s. As can be seen in this figure, increasing ‘$c$’ from 0.5 wt% to 4 wt% may lead to an increase in the bubble’s rise speed which is in agreement with results reported in Ref. 12. Interestingly, the cusp shape in the trailing edge exhibits itself sooner the larger the $c$. To explain these results, it should be noted that since the total viscosity is held constant, therefore by increasing ‘$c$’, the shear viscosity is decreased. As a result, the bubble moves easier and consequently faster. On the other hand, by an increase in $c$ the polymer contribution to the overall viscosity is increased, so that the viscoelastic behavior (namely the emergence of the cusp) is expedited. It is notable that the maximum magnitude of the longitudinal polymeric stress is increased from 69.8 g/cm$^2$s$^2$ to 991.3 g/cm$^2$s$^2$ as polymer concentration is increased from 0.5 wt% to 4 wt%.

Fig. 8. WC-SPH computation of the two normal polymeric stress components induced in the Oldroyd-B liquid during bubble rise obtained at $t = 0.13$ s for $c = 13.286$ wt% and $\lambda = 0.2$ s: a) contours of $t_p^{yy}$ in g/cm$^2$s, b) contours of $t_p^{yy}$ in g/cm$^2$s.

Fig. 9. The gaseous bubble shape rising in an Oldroyd-B fluid obtained at $t = 0.45$ s ($c = 1$ wt%, $\lambda = 3$ s, $\rho_l/\rho_b = 1000$, $\mu_l/\mu_b = 200$): a) full domain, b) enlarged view.

Fig. 10. The effect of relaxation time on a bubble rising in an Oldroyd-B fluid obtained at $t = 1.5$ s: a) $\lambda = 0.75$ s, b) $\lambda = 1.5$ s, c) $\lambda = 6.0$ s, d) $\lambda = 12.0$ s.
5. CONCLUDING REMARKS

Numerical results reported in this work suggest that SPH method is well competitive with the FEM-LS method in its dealing with the bubble rise problem even if the surrounding liquid is viscoelastic. Still, it is advised that use should preferably be made of the WC-SPH method instead of the I-SPH method if capturing the details of the bubble’s shape during its rise (such as its cusped-shape trailing edge) is of interest. The failure of the I-SPH method used in Ref. 24 for predicting the cusp is related to the usage of several stages of weighted arithmetic and harmonic interpolations for smoothing the transport parameters of the constituents. Irrespective of the fact that unlike I-SPH, the WC-SPH method relies on an artificial equation of state (meaning the pressure field obtained using I-SPH is more precise), the WC-SPH code could still correctly predict the effect of polymer concentration and its relaxation time on the bubble shape and its rise speed. Based on the results obtained using our SPH code, the viscoelastic behavior of the surrounding liquid drastically affects the bubble shape as it deforms from a rounded spherical shape to a cusped shape. An increase in the polymer concentration and/or the relaxation time of the liquid are predicted to increase the bubble’s rise speed with the trailing cusp exhibiting itself sooner. These SPH results qualitatively comply with published data for viscoelastic liquids.\textsuperscript{12)}

ACKNOWLEDGEMENTS

The authors wish to express their sincerest thanks to Iran National Science Foundation (INSF) for supporting this work under contract number 92011995. Special thanks are also due to the respectful reviewers for their constructive comments.

REFERENCES


