Bubbles Alive

Jeong-Mo Hong∗ Ho-Young Lee∗ Jong-Chul Yoon∗ Chang-Hun Kim∗
Korea University Korea University Korea University Korea University

Abstract

We propose a hybrid method for simulating multiphase fluids such as bubbly water. The appearance of subgrid visual details is improved by incorporating a new bubble model based on smoothed particle hydrodynamics (SPH) into an Eulerian grid-based simulation that handles background flows of large bodies of water and air. To overcome the difficulty in simulating small bubbles in the context of the multiphase flows on a coarse grid, we heuristically model the interphase properties of water and air by means of the interactions between bubble particles. As a result, we can animate lively motion of bubbly water with small scale details efficiently.


Keywords: Fluid simulation, bubbles, grid-based simulation, smoothed particle hydrodynamics, multiphase fluids

1 Introduction

The lively but chaotic motion of bubbles has enchanted and challenged many scientists. Besides the engineering applications, including ship hydrodynamics, cooling of nuclear reactors, and laundry machines, an understanding of bubbles is indispensable to the visual realism of computer-generated animations that show the multiphase characteristics of fluids. In computer graphics, many researchers are struggling to get more realistic bubbles and foams by means of physics-based fluid animation, powered by computational fluid dynamics.

The two major approaches, based on Eulerian grids and Lagrangian particles, have been competing with each other, but are now being combined. This is desirable because they are complementary methods: a particle system based on smoothed particle hydrodynamics (SPH) can be much more flexible and controllable if it concentrates on small-scale details, while large bodies of water and air can be handled efficiently and faithfully by a grid-based solver, without requiring excessive resolution.

The hybrid approach to multiphase flows (including bubbles) has received less attention than the simulation of splashes and droplets, because the difference in scale as compared to the background flow is more severe for bubbles than droplets. Water dominates the inertia because its density is 800 times higher than that of air, and thus bubbles require the surrounding water to be simulated in more detail than the air around a splash. In our experience, each bubble should occupy at least $3^3$ nodes (or $3^2$ in 2D) to have numerical meaning. This makes it infeasible to refine the grid sufficiently to capture all the small details, especially in a graphics context. That is why it is desirable to develop a dynamic model appropriate for representing details at the subgrid scale.

In this paper, we propose a hybrid method for simulating multiphase fluids, especially focusing on bubbles. To avoid excessive refinement of the background grid, while maintaining the subgrid details of bubble motion including path instability, we model the interphase properties of water and air in terms of the interactions between bubble particles. While this is ultimately a heuristic approach, it is underpinned by the SPH vorticity confinement method and an analysis of the cohesive forces that generate subgrid turbulence. This combination enables us to capture the natural look of moving bubbles in a way that harmonizes with an underlying grid-based simulation of multiphase flows.

2 Previous Work

The success of grid-based liquid animation techniques that use a free surface single-phase model (see [Foster and Metaxas 1996], [Foster and Fedkiw 2001], and [Enright et al. 2002b] for examples) led to work on the direct numerical simulation of multiphase phenomena [Hong and Kim 2003; Song et al. 2005; Kim et al. 2007a; Kim et al. 2007b; Hong and Kim 2005; Mihalef et al. 2006; Losasso et al. 2006; Hong et al. 2007].

Premoze et al. 2003] presented a particle-based method for fluid simulations that can handle multiphase liquids. [Müller et al. 2005] applied the SPH method to multiple phases, and [Cleary et al. 2007] modeled the nucleation, collision, and drag interactions of bubbles and foams, based on a background SPH simulation.

Kim et al. 2006] used the SPH method to model escaped parti-
icles within the particle level set method [Enright et al. 2002a], so as to resolve subgrid splashes. [Losasso et al. 2008] improved this approach by coupling a model of dense water volume to diffuse sprays. [Greenwood and House 2004] also modeled escaped particles to give a more detailed look to bubbles and foams, but without using SPH. [Thuerey et al. 2007] coupled SPH bubbles to shallow-water simulations using locally defined vortices on particles.

3 A Hybrid Approach

We use the Eulerian method to model the background motions of water and air bodies which are large enough to be captured using a simulation grid which can be managed by an ordinary single-CPU computer. The bubbling details that are too small to be handled on such a grid are simulated by SPH particles. We built our system on the particle level set fluid solver [Enright et al. 2002b] in order to generate bubble particles by incorporating the escaped particles back into the SPH system as bubbles, similar to [Greenwood and House 2004]. However, we believe that this hybrid framework and our bubble model would also integrate well with other grid-based techniques such as the CIP method [Song et al. 2005], the BFECC method [Kim et al. 2007b], the CLSVOF method [Mihailef al. 2006], or the Lattice-Boltzmann method [Thuerey 2007], if appropriate ways of generating bubbles were available.

Figure 2 is a schematic overview of our hybrid system. Since we accelerate our solver by using an octree grid [Losasso et al. 2004], the scale difference between the grid-spacing and the particle radius is large. This difficulty is resolved by our subgrid-scale bubble dynamics, which we develop in Section 4.

3.1 Grid-based Background Simulation

The Navier-Stokes equations describing inviscid incompressible fluid motion are

\[ u_t + (u \cdot \nabla)u + \nabla p/\rho = f \]

\[ \nabla \cdot u = 0, \]

where \( u \) is the velocity, \( p \) is the pressure, \( \rho \) is the density, and \( f \) is the aggregate of the external forces including gravity and the momentum exchange from the SPH bubbles that occurs during coupling. Since numerical methods of solving Equations (1) and (2) are well known, we refer readers to [Enright et al. 2002b; Losasso et al. 2004; Hong and Kim 2005] for details.

3.2 SPH Overview

The acceleration of a particle \( i \) is determined by a sum of forces exerted by adjacent particles, \( f_{ij} \), as follows:

\[ u_i = \sum_j f_{ij}/\rho_i, \]

where the density of a particle \( i \) is defined as \( \rho_i = \sum m_i W(x_{ij}, r_i) \). We use the radially symmetric kernel functions \( W(x, r) \) with support \( r \), as defined in [Müller et al. 2003]. The velocity and the position of a particle can be determined by sequential Euler integrations such as \( \mathbf{v}^{t+\Delta t} = \mathbf{v}^t + \mathbf{a}^t \Delta t \) and \( \mathbf{p}^{t+\Delta t} = \mathbf{p}^t + \mathbf{v}^t \Delta t + \frac{1}{2} \mathbf{a}^t \Delta t^2 \), where \( \Delta t \) is a time step. Following the adaptive-radius approach of [Adams et al. 2007], which provides a versatile description of bubble details, the pressure force can be expressed as

\[ f^\text{pressure}_{ij} = -V_i V_j (P_i + P_j) \left( \nabla W(x_{ij}, r_i) + \nabla W(x_{ij}, r_j) \right) / 2, \]

where the volume \( V_i \) is \( m_i / \rho_i \), \( r \) is the radius, the mass \( m_i \) is proportional to \( r_i^3 \), \( x_{ij} = x_j - x_i \), and the pressure \( P_i = k \rho_i \) with a control parameter \( k \). In general, SPH systems largely depend on viscosity, especially to improve stability when they are used to simulate large bodies. Since we use a grid-based solver to deal with large bodies, the viscous forces can be omitted.

3.3 Two-way Coupling

The major coupling forces which make the bubble particles follow the background flows are drag and lift forces [Cleary et al. 2007; Magnaudet and Eames 2000], given by

\[ f_{ij}^{\text{drag}} = -k_{\text{drag}} r_i^2 |v_i - u_i| (v_i - u_i) \]

\[ f_{ij}^{\text{lift}} = -k_{\text{lift}} V_i (v_i - u_i) \times \omega_i, \]

where \( u_i \) and \( \omega_i = \nabla \times u_i \) are the velocity and the vorticity, which are interpolated at \( p_i \) from the grid values. Initially, we tried to simulate the path instability of bubbles with lift forces, but this did not work well enough since Equation (6) relies on the vorticity field around \( p_i \), being highly refined. This is one of the motivations to develop the heuristic bubble model of Section 4.

The forces reacting to these coupling forces are transferred to the surrounding fluid through Equation (1) after being distributed across a number of adjacent nodes. We also used reaction forces to model the popping of bubbles when they merge with the ambient air. In many cases, the SPH time step needs to be smaller than the grid simulation time step. Since the reaction forces change the grid velocities and repeated updating makes their values diverge, they must be stored separately and only added to the right-hand side of Equation (1) once per grid simulation time step.

4 Bubbles

4.1 SPH Vorticity Confinement

Unlike droplets moving through ambient air, bubble particles are subject to strong velocity diffusion because they are coupled to the surrounding fluid by drag and lift forces. Furthermore, these forces are determined from values interpolated on the coarse grid. To simulate the motion of bubbles in more detail, we therefore introduce a
heuristic representation of the vorticity confinement [Fedkiw et al. 2001] into the SPH method.

First, we measure the vorticity $\omega = \nabla \times \mathbf{v}$ at the mass center of two SPH particles, $p_i = (m_i \mathbf{p}_i + m_j \mathbf{p}_j) / (m_i + m_j)$. In contrast to the grid-based method of [Fedkiw et al. 2001], we are able to express the vorticity location vector $\eta$ as $\eta = \mathbf{p}_i + \mathbf{p}_j$. We can use a normalized version of $\eta$, $N = \eta / |\eta|$, to determine the confinement force:

$$f_{ij}^{\text{vorticity}} = \epsilon \left( N \times \frac{\omega}{|\omega|} \right) \rho_i. \tag{7}$$

The original vorticity confinement method used by [Fedkiw et al. 2001] can amplify the existing vorticity over time because the incompressibility enforced by the projection step ensures stability. Taking a similar approach makes the SPH system diverge and we therefore used a normalized $\omega$.

### 4.2 Cohesive Forces

Due to the very large density ratio of water to air, water exerts a high pressure on air bubbles causing them to merge rapidly. To achieve a physically accurate simulation, multiphase SPH methods such as those of [Hu and Adams 2006] and [Müller et al. 2005] are desirable. However, because we simulate the water on a coarse grid, we have to take care of the multiphase interactions without explicit models of water particles or detailed velocities around air particles. By assuming that air particles are surrounded by water except where they are explicitly modeled, we can handle this multiphase property by simulating the attraction forces between touching particles, rather than attempting to model the forces exerted by water particles on air particles. Finally, we introduce a cohesive attraction force between particles:

$$f_{ij}^{\text{attraction}} = k_{\text{attraction}} W_{\text{attraction}}(r_i - r_j) \rho_i. \tag{8}$$

We use a constant-valued function for $W_{\text{attraction}}$ to make it easy to establish a force that balances the pressure forces. The pressure force in Equation (4) pushes adjacent particles outward when the density $\rho_i$ becomes high due to the attraction forces. We note that [Becker and Teschner 2007] introduced a similar force to represent surface tension, but this can be adequately modeled by the intrinsic properties of SPH in the physical situation with which we are dealing.

One way of inducing clustering would be to use the pressure kernels of [Becker and Teschner 2007] or [Adams et al. 2007] with a negative term so that attraction forces are exerted on adjacent particles when the particle density is low. This is a reasonable approach, but we believe that our attraction force is physically more plausible for bubbles under large water pressure. It also works better with the vorticity confinement techniques explained in the previous section, since it can suppress the scattering of particles by centrifugal effects to the extent required.

### 4.3 Subgrid Turbulence

The beauty of bubble motions is mainly a result of their unstable paths. Even a single bubble rising in calm water moves along a zigzag or spiral path due to its own wake (see [Shew and Pinton 2006] for an example). Our combination of a cohesive attraction force and SPH vorticity confinement approximates this characteristic motion (see Figure 3) when two or more particles are close together. For single bubbles, it is simplest to add disturbances to the particles’ velocities based on random numbers. This also helps to generate an initial vorticity and our system generates the natural look of turbulent bubble motion with the combination of these techniques.

**Figure 3:** Rising bubbles in calm water. This example shows the realistic motion of bubbles generated by our bubble model coupled to background flows. Simulation took 3 hours on an octree grid with an effective resolution of 256 $\times$ 128$^2$. A maximum of 2,600 SPH particles were used.

### 4.4 Buoyancy

The rising velocities of bubbles are determined by the balance between drag and buoyancy, which establishes a terminal upward velocity. We generally make the buoyant force $f_{\text{buoyancy}}$ proportional to the volume of each particle. An alternative is to make the buoyant force proportional to the difference between the current velocity and a terminal velocity approximately proportional to the particle radius [Mendelson 1967]. This could be used to improve the upward motion of the bubbles.

### 5 Examples

Both water surfaces and bubbles can be ray-traced as a single level set surface by performing on-the-fly Boolean operations that subtruct air bubbles from water bodies. We set the particle radii to between 0.3 and 0.8 of the grid spacing. Simulations were performed on a PC with an Intel Core2 CPU running at 3 GHz.

Figure 3 shows bubbles freely rising in water. In this example, bubble particles are seeded randomly at the bottom and then rise, demonstrating the basic capabilities of our bubble model. The lively and natural motion of bubbles, including flickering, merging, separation, and spiral path instability were simulated successfully. On the accompanying video there are animations with and without our vorticity confinement heuristic, which show that simply adding random disturbances is not adequate. Our bubbles pop as soon as they reach the surface, rather than persisting as foam, which is out of the scope of this paper but could be implemented using the methods already investigated by [Greenwood and House 2004] and [Cleary et al. 2007]. Figure 1 shows water being poured.
The atomization of large bodies of air is naturally modeled by escaped particles, and the coupled motion of level set surfaces and SPH particles achieves realistic bubbly water.

6 Conclusion

We have presented a hybrid of Eulerian grid-based simulation and Lagrangian SPH for the realistic simulation of multiphase fluids, focusing on bubbles. Using our heuristic bubble model, we can generate natural looking computer generated bubbly water.

Acknowledgements

This work was partly supported by the IT R&D program of MIC/IITA [2007-S001-01, Development of wearable interface technology for emergency rescue] and accomplished as the result of the research project 2007 culture technology development supported by KOCCA. We would also like to thank Tamar Shinar for many helpful suggestions.

References


