3D Simulations of Cavitation Bubble Breakup over a Lifting Surface

Chao-Tsung Hsiao, Jingsen Ma, Georges L. Chahine

DYNAFLOW, INC., Jessup, Maryland, USA

ABSTRACT

Two non-spherical bubble dynamics models are used to simulate a traveling cavitation bubble over a hydrofoil. The first is based on a Boundary Element Method modified to account for the non-uniform flow and shear, which treats the bubble in a Lagrangian fashion and discretizes finely its bubble to allow tracking of the non-spherical deformations. The other is based on Navier Stokes solution with a Level-Set Method to describe the bubble surface, which enables continuation of the description of the bubble dynamics beyond breakup. Using these methods, the numerical simulations of the traveling cavitation bubbles over the hydrofoil begin with the release from upstream of pre-existing nuclei with various sizes. The effects of the initial bubble size, release locations, and the cavitation number on the traveling cavitation bubble behaviors and shape variation are studied in order to improve bubble breakup parameters to be used in the Eulerian-Lagrangian approach.

Keywords

Non-spherical Bubble Dynamics, Two-Phase Flow, Bubble Breakup, Traveling Cavitation bubble

1 INTRODUCTION

Previous studies of the interaction between a lifting surface and a dispersed bubbly flow have shown that gas diffusion and cavitation can significantly alter bubble size distribution and increase bubble number downstream of a hydrofoil or propeller (Hsiao et al. 2012). Previous results can be illustrated with the computed bubble distribution over the foil and in the wake shown in Figure 1, which are very similar to experimental observations and in the change in the bubble size distribution shown in Figure 2. Additional studies have then shown that bubble breakup can further significantly increase bubble numbers and alter the size distribution (Hsiao et al. 2018). These numerical simulations were conducted using an Eulerian-Lagrangian approach, which uses equivalent spherical bubbles, which split into multiple daughter bubbles during collapse based on a couple of heuristic breakup models. The bubble splitting models included one based on stability analysis (Brennen, 1995) or another one based on empirical observations of the breakup in a cavitating venturi flow (Hsiao et al. 2018). Observations a bubbly flow over a hydrofoil (e.g. in Russell et al. 2018) show that the bubbles deform significantly during their violent dynamics. These observations show that the bubble nuclei grow to large bubbles in the low-pressure region of the hydrofoil and flatten over the rigid surface while being sheared in the boundary layer. Such non-spherical deformations may require bubble breakup criteria different from those obtained from spherical shape stability analysis.

Figure 1. Dispersed bubbly flow over the finite-span hydrofoil with gas diffusion and breakup considered (from Hsiao et al. 2018).

Figure 2. Comparison of the downstream bubble size distribution with and without considering bubble breakup (from Hsiao et al. 2018).

To consider the bubble non-spherical deformations due to flow inhomogeneity near a foil rigid wall and account for boundary layer shearing effects on bubble breakup, we have conducted 3D simulations of a traveling cavitation bubble over a hydrofoil using two 3D bubble dynamics models. The first one is based on a viscous multi-scale
two-phase flow model (Hsiao et al. 2017 and Ma et al. 2017), which combines viscous solution with a Level Set Method description of cavities. The other one is based on a Boundary Element Method (BEM), which accounts for the non-uniform flow and shear (Chahine, 1995). The BEM treats the bubble in a Lagrangian fashion and discretizes finely its interface to allow tracking of non-spherical deformations. It uses the viscous flow generated by the foil as a background flow through a Helmholtz decomposition. The BEM model is known to provide high accuracy description of bubble interfaces and reentrant jets, but suffers when the bubble becomes multi-connected prior to splitting. On the other hand, the Level Set model enables continuation of description of the bubble dynamics beyond breakup but provides less accurate description of the interfaces.

2 NUMERICAL MODELS
2.1 Multi-Scale Two-Phase Flow Model
The multi-scale two-phase flow model (Hsiao et al. 2017 and Ma et al. 2017) is capable of accurately representing two-phase bubbly flows at various relevant scales and includes three major components: a) In the macroscale liquid/gas two-phase flow continuum-based phase averaged Navier-Stokes equations are solved using an Eulerian approach. In this scale the gas/liquid interface of large bubbles or cavities are resolved directly. b) A Lagrangian approach, dubbed Discrete Singularity Model (DSM) is used to track sub-grid bubbles. c) A transition model is used to transform a sub-grid bubble into a large-scale bubble tracked by the Level Set Method (LSM) once a bubble grows beyond grid size during its dynamics.

2.1.1 Eulerian Continuum-Based Mixture Model
The two-phase flow continuum model uses the Navier-Stokes equations solver, 3DYNAFS-VIS® (VIS), which satisfies the continuity and momentum equations:

\[ \frac{\partial \rho_m}{\partial t} + \nabla \cdot \left( \rho_m \mathbf{u} \right) = 0, \]  
\[ \rho_m \frac{D \mathbf{u}}{Dt} = -\nabla p + \mu \nabla^2 \mathbf{u}, \]  

where the subscript \( m \) represents the mixture properties. \( \mathbf{u} \) is the mixture velocity and \( p \) is the pressure. The mixture density, \( \rho_m \), and the mixture viscosity, \( \mu_m \), are functions of the gas volume fraction, \( \alpha \):

\[ \rho_m = (1 - \alpha) \rho_l + \alpha \rho_g, \quad \mu_m = (1 - \alpha) \mu_l + \alpha \mu_g, \]  

where the subscript \( l \) represents the liquid and the subscript \( g \) represents the gas. The medium density and viscosity are time- and space-dependent.

The system of equations is solved by an artificial compressibility method (Chorin, 1967) in which a pseudo-time derivative of the pressure multiplied by the artificial compressibility factor, \( \beta \), is added to the continuity equation as:

\[ \frac{1}{\beta} \frac{\partial p}{\partial t} + \frac{\partial \rho_m}{\partial t} + \nabla \cdot \left( \rho_m \mathbf{u} \right) = 0. \]

As a consequence, a hyperbolic system of equations is formed and can be solved using a time marching scheme. The solution is iterated in the pseudo-time until convergence. To obtain a time-dependent solution, a Newton iterative procedure is performed at each physical time step in order to satisfy the continuity equation.

2.1.2 DSM for Sub-grid Bubbles
When a bubble cannot be resolved accurately by the grid, it is modeled as a discrete singularity and tracked in a Lagrangian fashion. The Surface Averaged Pressure (Ma et al. 2015b) modified version of the Keller-Herring equation (Keller and Kolodner, 1954) is used to describe the bubble dynamics,

\[ \frac{1}{\rho_m} \left[ \left( \frac{\hat{R}}{c_m} \right) R \frac{d R}{d t} + \left( 1 - \frac{\hat{R}}{3 c_m} \right) \frac{d \hat{R}}{d t} \right] = \frac{u_i^2}{4} + \left( 1 + \frac{\hat{R}}{c_m} \right) \frac{R d}{d t} \left( p_v + \rho_{genc} - p_{enc} - \frac{2 \gamma}{R} - 4 \mu_m \hat{R} \right). \]

Here \( c_m \) is the local sound speed in the mixture. \( R \) and \( R_0 \) are the bubble radii at times \( t \) and \( 0 \). \( p_v \) is the liquid vapor pressure, and \( \mathbf{u}_{enc} \) and \( p_{enc} \) are respectively the averages of the liquid velocities and pressures over the bubble surface. The slip velocity, \( \mathbf{u}_s = \mathbf{u}_{enc} - \mathbf{u}_l \), is the difference between \( \mathbf{u}_{enc} \) and the bubble translation velocity, \( \mathbf{u}_l \). The gas pressure, \( p_g \), is obtained from a polytropic compression law if no gas mass transfer effects are included. When gas diffusion is included, the gas pressure is obtained from the solution of the gas diffusion problem and energy balance.

The bubble trajectory is obtained from the following bubble motion equation:

\[ \frac{d \mathbf{u}_l}{d t} = \frac{\rho_l}{\rho_g} \left[ \frac{3 C_D}{8 R} \left| \mathbf{u}_l \right| \left| \mathbf{u}_l \right| + \frac{1}{2} \left( \frac{d \mathbf{u}_{enc}}{d t} - \frac{d \mathbf{u}_l}{d t} \right) \right] + \frac{3 \hat{R}}{2 R} \left( - \frac{\nabla p}{\rho_l} + \frac{\rho_{genc} - \rho_l}{\rho_l} g + \frac{3 C_L}{4 \pi R} \sqrt{R} \left[ \mathbf{u}_l \times \Omega \right] \right), \]

where \( C_D \) is the bubble drag coefficient, given by an empirical equation, such as from Haberman and Morton (1953). \( C_L \) is a lift coefficient and \( \Omega \) is the vorticity vector. The first right hand side term is a drag force. The second and third terms account for the added mass. The fourth term accounts for the pressure gradient, while the fifth term accounts for gravity and the sixth term is a lift force (Saffman 1965).

2.1.3 LSM for Resolved Non-Spherical Bubbles
The liquid/gas interface of the resolved cavitation bubble is captured by the Level Set method. In this method, a smooth distance function \( \varphi(x, y, z; t) \), whose zero value coincides with the liquid/gas interface, is defined in the whole physical domain (i.e. in both liquid and gas phases) as \( d(x,y,z) \), the signed distance from the interface:

\[ \varphi(x, y, z, t) = d(x, y, z). \]
This function is enforced to be a material surface at each time step by imposing:
\[
\frac{\partial \varphi}{\partial t} + u \cdot \nabla \varphi = 0.
\]  
(8)

To avoid that the value of \( \varphi \) gets diffused by numerical viscosity and that the level set be distorted by the flow field, a new distance function is constructed by solving a “re-initialization equation” through pseudo-time iterations:
\[
\frac{\partial \varphi}{\partial \tau} = S(\varphi)\left[1 - |\nabla \varphi|\right],
\]  
(9)

where \( \tau \) is the pseudo time, \( \varphi_b \) is the initial distribution of \( \varphi \), and \( S(\varphi_b) \) is a sign function, which is zero on the interface.

A Ghost Fluid Method (Sussman, 1998) is then used to impose dynamic boundary conditions at the interface without applying any smoothing. The dynamics condition satisfies continuity of stresses across the surface. If shear stresses due to the gas inside the cavity are neglected, the dynamic boundary conditions (balance of normal stresses and zero shear) can be written:
\[
p = p + \frac{\gamma}{\rho} + \tau_u n u, \quad \tau_u n u = 0, \quad \tau_u n u^2 = 0,
\]  
(10)

where \( p \) is the acceleration of gravity, \( \gamma \) is the surface tension parameter and \( \kappa = \nabla \nabla \varphi \) is the surface curvature. \( \vec{n}, \vec{i}^1 \) and \( \vec{i}^2 \) are the surface normal and two tangential unit vectors, respectively.

![Figure 3. Illustration of the transition from a DSM bubble to a Level Set resolved cavity. \( \varphi_i \) is defined as the shortest distance between a cell \( i \) and the bubble \( j \) being activated.](image)

### 2.1.4 Transition Scheme from DSM to LSM

Since DSM bubbles have sub-grid size, by definition, a criterion based on the bubble equivalent radius is set to “activate” the bubbles and transform them into cavities resolved using the level set field (Hsiao et al. 2017, Ma et al. 2017). As denoted in Figure 3, for each cell \( i \), a value of the distance function is determined using:
\[
\varphi_i = \min(\varphi_{\text{sub}}, \varphi_j) \quad j = 1, N_j
\]  
(11)

where \( \varphi_{\text{sub}} \) is the initialization distance or that from the previous level set computation and \( \varphi_j \) is the new distance function value using the closest distance to the newly activated bubble \( j \) surface. \( N_j \) is the number of bubbles, which are “activated” around cell \( j \) at the particular time step. This scheme enables a singularity to become a level set resolved cavity (Hsiao et al. 2017, Ma et al. 2017).

The size criterion used to determine which bubble to “activate” is the following:
\[
R \geq \max(R_{\text{th}} m_{\text{th}}, \Delta L)
\]  
(12)

where \( \Delta L \) is the size of local grid, which hosts the bubble, \( R_{\text{th}} \) is a threshold bubble radius and \( m_{\text{th}} \) is a threshold grid factor. This expresses that a sub-grid bubble represented by singularities is switched to be represented by a level set free surface only when the bubble grows its equivalent radius becomes larger than the threshold bubble radius and a user selected multiple of the local grid size, \( R_{\text{th}} \). The latter ensures good grid resolution to resolve the activated cavity. In the present study, the value \( m_{\text{th}} = 1.0 \) was used.

### 2.5 BEM Non-Spherical Bubble Dynamics Model

To study non-spherical bubble dynamics in the foil flow field, a modified Boundary Element Method (BEM) is applied. This model assumes that the flow due to the bubble dynamics is potential and that the background viscous flow time and space dependence is known. The model is developed using the fact that any velocity field \( u \) can be expressed, via the Helmholtz decomposition, as the sum of the gradient of a scalar potential \( \phi \) and the curl of a vector potential \( A \) with \( \nabla^2 A = -\omega \) (Chahine et al. 1997),
\[
u = u_p + u_e = \nabla \phi + \nabla \times A.
\]  
(13)

The flow due to the bubble dynamics is assumed to be potential and expressed by \( u_e = \nabla \phi \) and the vortical flow field is assumed to be that in absence of the bubble. Since the potential flow field due to the bubble presence satisfies the Laplace equation \( \nabla^2 \phi = 0 \), Green’s identity can be applied to construct the integral equation for the potential \( \phi \) and the normal derivative of the potential \( \partial \phi / \partial n \) as:
\[
\Omega \phi = \int [\phi(x') \frac{\partial G}{\partial n}(x, x') - \frac{\partial \phi}{\partial n}(x')G(x, x')] dS(x'),
\]  
(14)

where \( \Omega \) is the solid angle subtended by the fluid at the point \( x \) and \( G(x, x') = |x - x'|^{-1} \) is the free space Green’s function. To solve Equation (14) with the BEM, we discretize the bubble surface by triangular elements and then rewrite the equation in a discretized form as
\[
\Omega \phi(x) = \sum_{j=1}^{N} \left[A_j(x, x') \frac{\partial \phi}{\partial n}(x') - B_j(x, x') \phi(x')\right],
\]  
(15)

where \( A_{ij} \) and \( B_{ij} \) are elements of the influence matrices in (15) and \( N \) is the number of the discretized nodes on the bubble surface. Equation (15) can be applied to determine the normal velocity \( \partial \phi / \partial n \) on the bubble surface provided that \( \phi \) is known over the bubble surface. In the time stepping procedure, all the nodes on the bubble surface are moved to their new positions using the displacement \( \partial \phi / \partial n \cdot \vec{e}_n + V \cdot \vec{e}_t \), where \( \vec{e}_n \) and \( \vec{e}_t \) are the unit normal and tangential vectors at the bubble surface.
and $V_t$ is the tangential velocity. At the next time step, $\phi$ is updated by

$$\phi^{n+1} = \phi^n + \left( \frac{\partial \phi^n}{\partial t} + (u_n + u_{\omega}) \cdot \nabla \phi^n \right) \Delta t. \quad (16)$$

To find an expression for $\partial \phi / \partial t$ on the bubble surface, the Helmholtz decomposition (7) is substituted into the Navier-Stokes equation to obtain:

$$\frac{\partial (u_p + u_{\omega})}{\partial t} = - \frac{\nabla p}{\rho} + \nabla \cdot (u_p + u_{\omega}). \quad (17)$$

where the pressure field $p$ is the sum of the pressure $p_{\omega}$ due to the vortex field and $p_o$ is the pressure due to the bubble dynamics. Note that the background vortical flow field also satisfies the Navier-Stokes equation (1), which can be also written as:

$$\frac{\partial u_p}{\partial t} - u_p \times (\nabla \times u_p) + \nabla \frac{1}{2} |u_p|^2 = - \frac{\nabla p_{\omega}}{\rho} + \rho \nabla \phi. \quad (18)$$

Subtracting Equation (18) from (17), we can deduce the modified Bernoulli’s equation, which accounts for the presence of the vortical flow in the background.

$$\nabla \left( \frac{\partial \phi}{\partial t} + \frac{1}{2} |\nabla \phi|^2 + u_{\omega} \cdot \nabla \phi + \frac{p - p_{\omega}}{\rho} \right) = \nabla \phi \times (\nabla \times u_{\omega}). \quad (19)$$

In the present study, the vorticity in the vortex field is further assumed to be predominant only in the spanwise direction $e_z$. With $\nabla \times v = \omega, e_z$, this results in

$$\frac{\partial}{\partial t} \left( \frac{p - p_{\omega}}{\rho} + \frac{\partial \phi}{\partial t} + \frac{1}{2} |\nabla \phi|^2 + v \cdot \nabla \phi \right) = 0. \quad (20)$$

Because the disturbances due to the bubble, i.e. $\phi$ and $p - p_{\omega}$, decay to zero at infinity, we have

$$\frac{\partial \phi}{\partial t} = - \frac{p - p_{\omega}}{\rho} - \frac{1}{2} |\nabla \phi|^2 - v \cdot \nabla \phi. \quad (21)$$

The pressure inside a bubble is assumed homogeneous, and the gas inside the bubble is assumed to be composed of both vapor of the liquid and non-condensable gas. The pressure at any instant is given by the sum of the partial pressures of the liquid vapor, $p_v$, and of the non-condensable gas, $p_c$. The non-condensable gas is assumed to satisfy a polytropic law with an exponent $k$, and thus $p \theta^k$ remains constant regardless of the bubble volume $V(t)$ that varies with time.

$$p = p_v + p_{c0} \left( \frac{\Psi}{\Psi(t)} \right)^k - \gamma \mathcal{E}. \quad (22)$$

Here, $\Psi$ is the initial volume of the bubble and $\mathcal{E}$ is the local surface curvature given by $\mathcal{E} = \nabla \cdot n$. Equations (21) and (22) are combined to obtain an expression for $\partial \phi / \partial t$ used in (16) and become

$$\phi(t + \delta t) = \phi(t) +$$

$$\left[ \frac{1}{2} |\nabla \phi(t)|^2 + \frac{1}{\rho} \left( p_{\omega} - p_v - p_{c0} \right) \left( \frac{\Psi}{\Psi(t)} \right)^k + \gamma \mathcal{E} \right] \delta t. \quad (23)$$

The integral equation is then collocated at each node, and the resulting matrix vector equation is solved by using a standard LU decomposition technique.

Once the solution is obtained at any time step, the pressure at given field points can be calculated by using the Green identity and the unsteady Bernoulli equation. First, the Green identity is used to calculate the potential flow due to the bubble, i.e. $\phi$, and then the velocity is obtained from numerical differentiation. Finally, the pressure can be calculated by the Bernoulli equation.

3 RESULTS AND DISCUSSION

3.1 Unsteady Single-Phase Flow

The present study considers the unsteady flow field over a NACA63A015 hydrofoil with a chord length of 15 cm and a span width of 5 cm as in (Russell et al. 2018). The computational domain has all far-field boundaries located 6 chord lengths away from the foil and is discretized using an H-H type grid with a total of 1.8 million grid points (201×61×151). 121×61 grid points are used to discretize both the suction and pressure side of the hydrofoil surface. The first grid above the hydrofoil surface is at $y^+=1$ in order to properly describe the boundary layer.

Figure 4. Non-dimensional pressure contours on the central plane above foil surface at a selected time after reaching limited cycle oscillations.

We consider an incoming uniform flow at an angle of attack of $3.5^\circ$. Freestream velocities and pressures are specified in the inflow boundary and in the far-field side boundaries. A first order extrapolation for all variables is used at the outflow boundary. A symmetry boundary condition is applied at the foil root section and no-slip flow and zero normal pressure gradient conditions are imposed on the foil surface. The flow is directly simulated, without a turbulence model, at a Reynolds number, $Re=1.5\times10^5$ corresponding to a liquid velocity, $U_{le}=10$ m/s. In absence of nuclei, unsteady
Flow separation with vortex shedding is observed in the liquid and the computations are conducted until limit cycle oscillations are reached. Figure 4 shows the non-dimensional pressure contours, \(\frac{(p - p_e)}{0.5 \rho V_e^2}\), at a selected time after reaching limit cycle oscillations.

### 3.2 LSM Non-spherical Bubble Simulation

To gain more insight into bubble dynamics and breakup on the hydrofoil, a 3D simulation of the dynamics of a nucleus traveling very close to the foil surface and its breakup is conducted. The setup is the same as in the above simulations, with the exception that to reduce computation times, a finite width of 0.1 chord length is considered in the span direction for the viscous liquid flow with periodic boundary conditions imposed at both sides.

Using the multiscale two-phase flow model described above in Section 2, the simulation starts from the fully developed liquid flow field as in Section 3.1. A small nucleus with an initial radius of 20 \(\mu\)m is released upstream of the hydrofoil. The bubble is initially tracked in the flow field using DSM. The level-set based cavity model (LSM) is then turned on when the size of the bubble reaches the switch criteria, 1 grid size, as described above in Section 2.

![Figure 4. Non-dimensional pressure contours at a selected time after reaching limit cycle oscillations.](image)

**Figure 4.** Non-dimensional pressure contours at a selected time after reaching limit cycle oscillations.

As shown in Figure 5, as the small bubble travels into the low-pressure region on the suction side of the foil, it grows, strongly deforms, then breaks up downstream into multiple smaller bubbles as it encounters higher pressures and shear flow near the foil boundary.

This is further highlighted in Figure 6, which displays a zoomed front view of the bubble interacting with the liquid flow around it. From this figure it is seen, that the bubble elongates in the stream wise direction due to the viscous shear flow. The low-pressure region before bubble splitting is due to the wake behind the bubble when it travels forward. At the beginning of the dynamics, the lower part of the bubble touches the foil surface and the bubble significantly slows down due to the presence of the wall and the near zero velocities in the boundary layer. On the other hand, the upper part of the bubble continues to be moved downstream since it sees much higher velocities. This causes significant distortion of the bubble shape. On the other hand, as illustrated in Figure 7, the presence of the bubble also modifies temporarily the local velocity profiles near the boundary layer. These effects combine to result in bubble shearing and splitting.

![Figure 5. A time sequence of the bubble travelling over the foil surface. The silver iso-surface is the 3D bubble surface and the colors are the streamwise velocity contours in the domain mid-plane.](image)

**Figure 5.** A time sequence of the bubble travelling over the foil surface. The silver iso-surface is the 3D bubble surface and the colors are the streamwise velocity contours in the domain mid-plane.

As shown in Figure 5, as the small bubble travels into the low-pressure region on the suction side of the foil, it grows, strongly deforms, then breaks up downstream into multiple smaller bubbles as it encounters higher pressures and shear flow near the foil boundary.

This is further highlighted in Figure 6, which displays a zoomed front view of the bubble interacting with the liquid flow around it. From this figure it is seen, that the bubble elongates in the stream wise direction due to the viscous shear flow. The low-pressure region before bubble splitting is due to the wake behind the bubble when it travels forward. At the beginning of the dynamics, the lower part of the bubble touches the foil surface and the bubble significantly slows down due to the presence of the wall and the near zero velocities in the boundary layer. On the other hand, the upper part of the bubble continues to be moved downstream since it sees much higher velocities. This causes significant distortion of the bubble shape. On the other hand, as illustrated in Figure 7, the presence of the bubble also modifies temporarily the local velocity profiles near the boundary layer. These effects combine to result in bubble shearing and splitting.

![Figure 6. Zoomed view of the bubble shape being distorted by the shear flow over the hydrofoil before and after splitting. Top row: with streamwise velocity contours. Bottom row: with pressure contours.](image)

**Figure 6.** Zoomed view of the bubble shape being distorted by the shear flow over the hydrofoil before and after splitting. Top row: with streamwise velocity contours. Bottom row: with pressure contours.

![Figure 7. Streamwise velocity profiles at the three different streamwise locations indicated in the upper contours.](image)

**Figure 7.** Streamwise velocity profiles at the three different streamwise locations indicated in the upper contours.

To gain further insights on the effects of shear on the bubble dynamics and breakup, we consider next the effect of the initial bubble release height on the results. A quick estimate of the bubble trajectory and size history is shown first in Figure 8 using the DSM computations. Figure 9 shows the 3D simulations for the same three initial
elevations of the bubble release location. We can observe very different bubble shapes and daughter bubble sizes while slightly changing the initial bubble release height. When the initial release height is at 1.0 cm, the bubble part stretched over the foil surface has a much larger volume than the daughter bubble that splits from it.

The opposite is seen when the bubble is released 2 mm higher, \( Z_0 = 1.2 \) cm, with the daughter bubble now becoming the larger of the two remaining bubbles. For \( Z_0 = 1.1 \) cm the two daughter bubbles have about the same size. This can be explained by the large velocity gradient (see above in Figure 7) in the shear layer due to the boundary layer effects. When the bubble trajectory takes it closer to the foil surface, it encounters more of the low velocity in the boundary layer during its travel to downstream so that the bottom part is dragged more significantly and this results in a larger daughter bubble in the bottom part.

These simulations underline the important role played by the local vorticity on the bubble deformation and show the bubble-induced flow disturbance during the bubble dynamics and breakup. In the on-going studies such simulations will help uncover breakup criteria to correlate the breakup parameters with local flow conditions such as vorticity and turbulence, etc. However, an issue with the above procedure, which we uncovered while conducting this study, is that the transition from the Lagrangian DSM bubble to the VIS Level Set model appears to affect significantly the bubble growth rate. It is as if the liquid inertia due to the bubble strong growth is either rapidly damped out when the viscous flow is introduced or this inertia is not transferred properly and conserved in the transition procedure. We hope to find the reason for this discrepancy soon. This discontinuity can be clearly seen in Figure 10, which compares the bubble equivalent radius versus time for the different approached.

### Figure 8
Comparison of the bubble trajectory and size for bubbles released at different heights upstream of the hydrofoil.

### Figure 9
Comparison of the bubble shape at splitting into two bubbles for bubbles initially released at different (but very close) vertical locations near the hydrofoil.

### Figure 10
Transition from Lagrangian DSM bubble dynamics (green line) to Eulerian LSM bubble dynamics at different times (here corresponding to instants where the equivalent bubble radius reaches different values).

The green dashed line shows the bubble radius versus time in the foil viscous flow field obtained from the DSM Lagrangian computation. In this DSM simulations, the pressures and velocities from the liquid-only viscous solution drive the bubble dynamics and no feedback to the viscous solution is accounted for. The black, blue, and red curves are the viscous solutions using LSM where the transition is made when the bubble equivalent radius reaches 2, 4, and 5 mm, respectively. We can see that, contrary to what we should expect, the solution depends significantly on the selected transition time and the maximum radius achieved is smaller when the transition is made earlier, all indicating that the inertia of the liquid due to the explosive bubble growth is lost at the transition time, resulting in a strong drop in the slope of the bubble radius versus time. This is one reason why we turned to the Boundary Element Method to investigate whether it could be helpful in this research.

### 3.3 BEM Non-spherical Bubble Simulation
The transition from DSM to BEM accounting for the liquid only viscous flow does not have this issue. This is
illustrated in Figure 12. The black curve shows the DSM solution, which is able to describe bubble growth and oscillation over several cycles. Three BEM solutions are shown, each transitioning from the DSM solution (i.e., using the bubble center location, radius, initial gas pressure, and initial radius time derivative as initial conditions to start the BEM computation) at different times (or locations along their traverse over the foil). As we can see in the figure, the differences between the BEM solutions are very minor. This is contrary to the VIS green curve (as in Figure 11), which shows a large drop in the initial bubble growth rate and a significant reduction of the bubble growth rate and radius during the remainder of the computation.

As we can see from Figure 11, the DSM and BEM solutions result in bubble sizes much larger than the VIS solution. Consequently, for these cases most of the bubble volume is outside the boundary layer and viscous effects are not as important. The bubble shape and dynamics are controlled by inertia effects, shear, as well as by the local pressure gradients generated by the flow and the presence of the wall. Therefore, as seen in the sequence of 3D shapes in Figure 12, the bubble elongates in the flow direction, flattens over the wall during most of its dynamics and finally collapses with the formation of a strong reentrant jet directed upstream due to the pressure gradient (higher pressure downstream) and due to the presence of the wall.

Details of the very fast bubble shape evolution at the end of the collapse can be seen in Figure 13. A very wide reentrant jet, with a shape folded in the transverse direction, develops first and probably results, as in the VIS cases, in top parts of the bubble being torn out from the larger part close to the wall. The smoothing procedures used in the BEM method, unfortunately, do not allow capture of this splitting, and the small split parts simply recede and disappear. The remaining bubble close to the wall finally collapses into a classical bubble ring shape, which would result in a large number of small bubbles once the reentrant jet pierces the opposite side of the bubble.

In order to consider the cases where the bubbles do not grow as much as in Figure 12 and Figure 13, a BEM computation for a case of nucleation from the wall is presented in Figure 14. A 3 mm bubble with internal pressure 1 atm is emitted from the foil wall as x=-1cm from the foil center and is allowed to grow in the flow field. In this case, the bubble maximum radius achieved is

![Figure 11. Effects on the bubble equivalent radius versus time of the transition from DSM spherical bubble to 3D simulations accounting for nonuniform flow field. The green curve is the Navier-Stokes solution with LSM. BEM solutions with three transitions time are shown as well as one solution where the BEM is artificially forced to lose inertia at the transition time. P_{amb}=22,140 Pa, R_{0}=20 \mu m, Z_0=1 mm.](image1)

To test whether ignoring the initial growth rate has a large effect on the solution, the red dashed curve is a fourth BEM solution where the liquid inertia was artificially forced to be zero at the transition time. This affected the solution, but to a much smaller extent than for the VIS solution, and the BEM recovered the bubble dynamics in the following time steps. One possibility is that the changes in the flow and pressure field, as shown in Figure 7, due to the interaction between the bubble and the flow field, affect significantly the dynamics, but this need to be confirmed after the inertia non conservation issue is resolved.

![Figure 12. 3D bubble shape evolution during collapse showing strong reentrant jet directed upstream. P_{amb}=22,140 Pa, R_0=20 \mu m, Z_0=1 mm.](image2)

![Figure 13. 3D bubble shape evolution over time during the final phase of the bubble collapse. P_{amb}=22,140 Pa, R_{0}=20 \mu m, Z_0=1 mm.](image3)

![Figure 14. Top and side views of the bubble dynamics in the case of nucleation from foil surface. 3 mm bubble with an initial internal pressure of 1 atm activated at x=-1 cm.](image4)
only 3 mm and the shear effects are substantial. The bubble flattens and elongates substantially but it again ends its dynamics with a collapse producing a upstream directed reentrant jet, which should breakup the bubble after the formation of a toroidal bubble.

3 CONCLUSIONS
Starting with a fully developed viscous flow field over a 3D NACA63A015 hydrofoil, both Boundary Element Method and Level Set Method are used to simulate 3D dynamics of traveling cavitation bubbles over the hydrofoil. The simulations begin with the release from upstream of pre-existing nuclei with various conditions. Particularly, the effects of the initial release locations, the cavitation number and the initial bubble size on the traveling cavitation bubble behaviors and shape variation are studied and analyzed. The results aim to help understand the bubble breakup mechanisms and thus to improve the criteria to be used for bubble splitting in the Eulerian-Lagrangian simulations of bubbly flow over lifting surfaces, such as a hydrofoil or propeller. However, the Boundary Element Method and Level Set Method are presently not giving similar results and differences between the two need to be resolved.

ACKNOWLEDGEMENTS
This work was supported by the Office of Naval Research under contract N00014-18-C-1024 monitored by Dr. Ki-Han Kim. We sincerely appreciate this support.

REFERENCES


