Multi-Scale Two-Phase Flow Modeling of Sheet and Cloud Cavitation

C.-T. Hsiao, J. Ma, and G. L. Chahine
(DYNAFLOW, INC., USA)

ABSTRACT

A multi-scale two-phase flow model based on an Eulerian/Lagrangian coupled approach is developed to capture the sheet cavitation formation and unsteady breakup and cloud shedding on a hydrofoil. No assumptions are needed on mass transfer. Instead, natural free field nuclei and solid boundary nucleation are modelled and enable capture of the dynamics. The multi-scale model includes a micro-scale model for tracking the bubbles, a macro-scale model for describing large cavity dynamics and a transition scheme to bridge the micro and macro scales. With this multi-scale model small nuclei are seen to grow into large bubbles, which eventually merge to form a large scale sheet cavity. A reentrant jet forms under the sheet cavity, travels upstream, and breaks the cavity, resulting in high pressure peaks as the broken pockets shrink and collapse while travelling downstream. The results for a 2D NACA0015 foil are in good agreement with published experimental measurements in terms of sheet cavity lengths and shedding frequencies. Sensitivity assessment of the model parameters and importance of 3D effects on the predicted major cavity dynamics are also discussed in details.

INTRODUCTION

Various types of cavitation occur over a propeller blade. These include traveling bubble cavitation, tip vortex cavitation, sheet cavitation, cloud cavitation, supercavitation, and combinations of the above. Among these, the transition from sheet to cloud cavitation is recognized as the most erosive and is quite difficult to accurately model. The sheet cavity starts near the leading edge, effectuates large amplitude length/volume oscillations, and breaks up periodically releasing clouds of bubbles and cavitating vortical structures, which collapse violently generating high local pressures on the blade. Accurate simulation and modeling of the physics involved is challenging since the problems of sheet cavity dynamics, cloud cavity dynamics, and sheet-to-cloud transition involve length- and time-scales of many different orders of magnitude.

Experimental observations have indicated weak or little dependence of the sheet cavity inception and dynamics on the nuclei distribution in the test facility, which has led many numerical modelers to believe that sheet cavitation is not connected with bubble dynamics. Actually, over the years, several researchers (e.g. Kodama 1981, Kuiper 2010) have observed that the presence of free field nuclei affected significantly the repeatability of the experimental results. In experiments where nuclei count was very low, the value of the incipient cavitation number showed large scatter, whereas in water rich in nuclei the results were more repeatable and were not affected by further addition of nuclei. One potential explanation to the above observations is that nucleation from the solid surface of the blades could play a much more important role than the free field nuclei, and this may have been overlooked in previous studies of sheet cavitation even though bubble nucleation has been extensively investigated (e.g. Harvey et al. 1944, Yount 1979, Mørch 2009).

Conventional numerical approaches for modeling sheet cavities assume that the cavity is filled with vapor at the liquid vapor pressure and then arbitrarily define as a liquid-cavity interface the liquid iso-pressure surface with the vapor pressure value. The flow solution in the liquid is obtained using either a Navier-Stokes solver (Deshpande et al. 1993, Chen and Heister 1994, Hirschi et al 1998) or a potential flow solver using a boundary element method (Kinnas and Fine 1993, Chahine and Hsiao 2000, Varghese et al. 2005). Since the developed sheet cavity is very unsteady and dynamic, numerical methods requiring a moving grid to
resolve the flow field are accurate but encounter numerical difficulties to describe folding or breaking up interfaces. Other approaches have thus been developed to avoid using moving grids to track the liquid-gas interface. They are mainly in two categories: Volume of Fluid methods (VoF) (Merkle et al. 1998, Kunz et al. 2000, Yuan et al. 2001, Singhal et al. 2002) and Level-Set Methods (LSM) (Kawamura and Sakoda 2003, Dabiri et al. 2007, 2008, Kinzel et al 2009). While both interface capturing methods solve the Navier-Stokes equations using a fixed computational grid, the liquid/gas interface is captured differently. In the VoF method, an advection equation for the volume fraction of the vapor is solved and the two-phase medium density is computed knowing the volume fraction of each phase. This method does not consider a sharp interface with boundary conditions, but uses mass transfer equations between the liquid and the cavity to model liquid/vapor exchange at the interface using ‘calibration’ parameters to match experimental results. In the level set method, the interface location is determined by solving an advection equation for the level set function with the interface represented by the zero level (or iso-surface) of the level-set function. This method provides a sharp interface on which boundary conditions similar to those used in an interface tracking scheme (e.g. a constant pressure value, surface tension, and zero shear stress) can be imposed.

Interface capturing approaches have been successfully applied to describe the dynamic evolution of a cavitation sheet and its breakup but not to the dispersed bubble phase and to resolving the transition from sheet to bubbles since this would require an extremely fine spatial resolution relative to bubble size (Fedkiw et al. 1999, Osher and Fedkiw 2001). Resolving sheet cavity together with the microbubbles involves characteristic lengths different by orders of magnitude and using the smallest scale requires computational resources not currently available.

Instead of resolving all individual bubbles, continuum-based Eulerian approaches (Biesheuvel and Van Wijngaarden 1984, Druzhinin and Elghobashi 1998, 1999, 2001, Zhang and Prosperetti 1994 a,b, Ma et al. 2011, 2012), which assume the bubbly phase to be a continuum, are typically used for bubbles much smaller than the characteristic lengths associated with the motion of the overall mixture. In this case, the precise location and properties of individual bubbles are not directly apparent at the global mixture flow scale. As a result, this approach is not able to address appropriately the bubble dynamics and interactions which are important in cavitation and more so in cloud cavitation.

Approaches combining both continuum and discrete bubbles have been proposed (Chahine and Duraiswami 1992, Balachandar and Eaton 2010, Raju et al. 2011, Shams et al. 2011, Hsiao et al. 2013). In these approaches the carrier fluid is modeled using standard continuum fluid methods (Eulerian approach) and dispersed bubbles are tracked directly (Lagrangian approach). The interaction between the two phases is achieved through a locally distributed force field imposed in the momentum equation (Ferrante and Elghobashi 2004, Xu et al. 2002) or through the inclusion of local density variation due to bubble motion and dynamics (Raju et al. 2011, Hsiao et al. 2013, Ma et al. 2014).

The approach presented in this paper aims at addressing various scales of the problem; notably a macro-scale at the order of the blade length, a micro-scale for the bubble nuclei, an intermediary scale for the bubble clouds, and transition and coupling procedures between these scales. To resolve cavitation inception including sheet formation, we conduct the present study using the same concepts that we have pioneered to study cavitation inception (Chahine, 2004, Hsiao and Chahine 2004, 2008, Chahine, 2009). This starts from the precept that water always contains microscopic bubble nuclei and solid/liquid interfaces do not have perfect contact and thus entrap nanoscopic or microscopic gas pockets, which are sources of nucleation (freeing of nuclei into the liquid) when appropriate local conditions occur. Inception of all cavitation types is initiated through explosive growth of these nuclei.

The sketch in Figure 1 presents an overview of the multi-scale model developed in the present study. This model aims at addressing at the various scales the physics involved in sheet and cloud cavitation:

- At the micro-scale, transport of nuclei and microbubbles, nucleation from solid surfaces, and corresponding bubble dynamics are considered. The model addresses dispersed pre-existing nuclei

![Figure 1: Overview of the various scales involved in the problem of cavitation on a foil and sketch of the modeling strategy.](image-url)
in the liquid, nuclei originating from solid boundaries, and micro-bubbles resulting from disintegration of sheet and cloud cavities or bubbles breaking up from vapor/gaseous cavities. At this scale, the model tracks the bubbles in a Lagrangian fashion.

- At the macro-scale, a two-phase continuum-based flow is solved on an Eulerian grid. At this scale gas/liquid interfaces of large bubbles, air pockets, and cavities are directly discretized and resolved through tracking the gas-liquid interfaces with a Level Set method.

- Inter-scale transition schemes are used to bridge micro- and macro-scales as bubbles grow or merge to form large cavities or as bubbles shrink or break up from a large cavity. At this scale, model transition between micro-scale and macro-scale and vice-versa is realized by using information from both the Discrete Singularity Model (DSM) and the Level Set method (LSM).

An Eulerian/Lagrangian coupled two-phase flow model is used. The Eulerian model addresses the continuum-based two-phase model and is coupled with a discrete bubble model tracking in a Lagrangian fashion the bubbles’ motion.

### Eulerian Continuum-Based Two-Phase Model

The two-phase medium continuum model uses our viscous flow solver, 3DYNAFS-VIS\textsuperscript{©}, to solve the Navier-Stokes equations and satisfy the continuity and momentum equations:

\[
\begin{align*}
\frac{\partial \rho_m}{\partial t} + \nabla \cdot (\rho_m \mathbf{u}) &= 0, \\
\rho_m \frac{D \mathbf{u}}{Dt} &= -\nabla p + \nabla \left( \mu_m \left( \nabla \cdot \mathbf{u} \right) \delta \right), \\
\tau_{ij} &= \mu_m \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right),
\end{align*}
\] (1)

where the subscript \( m \) represents the mixture properties. \( \mathbf{u} \) is the mixture velocity, \( p \) is its pressure and \( \delta \) is the Kronecker delta. The mixture density, \( \rho_m \), and the mixture viscosity, \( \mu_m \), can be expressed as functions of the void volume fraction \( \alpha \):

\[
\rho_m = (1-\alpha) \rho_l + \alpha \rho_g, \quad \mu_m = (1-\alpha) \mu_l + \alpha \mu_g,
\] (3)

where the subscript \( l \) represents the liquid and the subscript \( g \) represents the gas.

The equivalent medium has a time and space dependent density since the void fraction \( \alpha \) varies in both space and time. This makes the overall flow field problem similar to a compressible flow problem. In our approach, which couples the continuum medium with the discrete bubbles, the mixture density is not an explicit function of the pressure through an equation of state. Instead, tracking the bubbles and knowing their concentration provides \( \alpha \) and \( \rho_m \) as functions of space and time. This is achieved by using a Gaussian distribution scheme which smoothly “spreads” each bubble’s volume over a selected radius including over neighboring cells while conserving the total bubble volumes (Ma et al., 2014). Specifically, the void fraction in a fluid cell \( i \) is computed by:

\[
\alpha_i = \frac{\sum_{j \in N_i} f_{i,j} V^b_j}{\sum_{k \in N_{\text{cells}}} f_{i,k} V^\text{cell}_k},
\] (4)

where \( V^b_j \) and \( V^\text{cell}_k \) are the volumes of a bubble \( j \) and cell \( k \) respectively, \( N_i \) is the number of bubbles which are influencing cell \( i \). \( N_{\text{cells}} \) is the total number of cells “influenced” by a bubble \( j \). \( f_{i,j} \) is the weight of the contribution of bubble \( j \) to cell \( i \) and is determined by the Gaussian distribution function. This scheme has been found to significantly increase numerical stability and to enable the handling of high-void bubbly flow simulations.

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

\[
\frac{1}{\beta} \frac{\partial p}{\partial t} + \frac{\partial \rho_m}{\partial t} + \nabla \cdot (\rho_m \mathbf{u}) = 0.
\] (5)

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

3DYNAFS-VIS\textsuperscript{©} uses a finite volume formulation. First-order Euler implicit differencing is applied to the time derivatives. The spatial differencing of the convective terms uses a flux-difference splitting scheme based on Roe’s method (Roe 1981) and van Leer’s MUSCL method (van Leer 1979) to obtain the first-order and the third-order fluxes, respectively. A second-order central differencing is used for the viscous terms, which are simplified using a thin-layer approximation. The flux Jacobians required in an implicit scheme are obtained numerically. The resulting system of algebraic equations is solved using the Discretized Newton Relaxation method (Vanden, and
Whitfield 1995) in which symmetric block Gauss-Seidel sub-iterations are performed before the solution is updated at each Newton iteration.

**Level-Set Approach**

In order to enable 3DYNAFS-VIS© to simulate large free surface deformations such as folding and breakup in the sheet cavity problem, a Level-Set method is used. A smooth function, \( \phi(x,y,z,t) \), whose zero level coincides with the position of the liquid/gas interface when the level set is introduced, is defined in the whole physical domain (i.e. in both liquid and gas phases) as the signed distance \( d(x,y,z) \) from the interface:

\[
\phi(x,y,z,0) = d(x,y,z). \tag{6}
\]

This function is enforced to be a material surface at each time step using

\[
\frac{\partial \phi}{\partial t} + u'_j \frac{\partial \phi}{\partial x_j} = 0, \tag{7}
\]

where \( u' \) is the velocity of the interface. Integration of Equation (7) does not ensure that the thickness of the interface region remains constant in space and time during the computations due to numerical diffusion and to distortion by the flow field. To avoid this problem, a new distance function is constructed by solving a “re-initialization equation” in pseudo-time iterations (Sussman 1998):

\[
\frac{\partial \phi}{\partial \tau} = S(\phi_0)[1-|\nabla \phi|]. \tag{8}
\]

In (8) \( \tau \) is the pseudo time, \( \phi_0 \) is the initial distribution of \( \phi \) and \( S(\phi) \) is a sign function, which is zero at the interface.

Although the Ghost Fluid Method can help maintain a sharp interface, additional computational cost is required for the cells in the ignored gaseous phase. To further reduce the computational cost we apply a single-phase level-set approach in which only the liquid phase of the fluid is solved, while the cells belonging to the gas phase are deactivated during the computations. To solve Equation (7) we use the velocity information from the gas cells which are near the interface. To obtain this information, we implement the following boundary condition at the interface:

\[
n \nabla u_i = 0 \quad \text{with} \quad n = \nabla \phi/|\nabla \phi|, \tag{11}
\]

which means there is no normal component of the velocity gradient at the interface. With this assumption we can extend the velocity from the liquid phase along the normal direction of the distance function to the gas phase by solving the pseudo-time iteration equation:

\[
\frac{\partial u_i}{\partial \tau} + n \cdot \nabla u_i = 0. \tag{12}
\]
Lagrangian Discrete Bubble Model

The Lagrangian discrete bubble model is based on the discrete singularity model, 3DYNFDS-DSM, which uses a Surface Average Pressure (SAP) approach (Hsiao et al., 2003, Chahine, 2004, Choi et al., 2004) to average fluid quantities over the bubble surface. This model has been shown to produce accurate results when compared to full 3D two-way interaction computations (Hsiao and Chahine, 2003). The averaging scheme allows one to consider the spherical equivalent bubble and uses a modified Rayleigh-Plesset equation to describe the bubble dynamics,

\[
\rho \left( R^2 + \frac{3}{2} \frac{R^2}{R_0} \right) = \rho_v + p_{v} \left( \frac{R}{R_0} \right)^{3/2} - p_{oc} - 2 \frac{\gamma}{R} + \frac{4 \mu R}{R} + \rho_s |u_s|^2 \frac{1}{4},
\]

where \( u_s = u_{oc} - u_v \) is the difference between the encountered (surface averaged) liquid velocity, \( u_{oc} \), and the bubble translation velocity, \( u_v \). \( R \) and \( R_0 \) are the bubble radii at time \( t \) and \( 0 \). \( \rho_v \) is the liquid vapor pressure. \( p_{v0} \) is the initial bubble gas pressure. \( k \) is the polytropic compression constant and \( p_{oc} \) is the average SAP pressure “seen” by the bubble during its travel. With the SAP model, \( u_{oc} \) and \( p_{oc} \) are respectively the averages of the liquid velocities and pressures over the bubble surface.

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

\[
\frac{d\mathbf{u}_b}{dt} = \left( \frac{\rho_v}{\rho} \right) \left[ \frac{3}{8R} C_D \left| \mathbf{u}_b \right| \mathbf{u}_b + \frac{1}{2} \left( \frac{du_{oc}}{dt} - \frac{du_v}{dt} \right) + \frac{3R}{2R} \mathbf{u}_b \right] - \frac{\mathbf{v}_p}{\rho} + \frac{\mathbf{\rho}_e - \mathbf{\rho}_v}{\rho} - g + \frac{3C_L}{4\pi R} \left( \mathbf{u}_b \times \mathbf{\Omega} \right),
\]

where \( \rho_v \) is the density of the bubble, \( C_D \) is the drag coefficient given by an empirical equation from Haberman and Morton (1953), \( C_L \) is the lift coefficient, and \( \mathbf{\Omega} \) is the vorticity vector. The 1st right hand side term of (14) is a drag force. The 2nd and 3rd terms account for the added mass. The 4th term accounts for the presence of a pressure gradient, while the 5th term accounts for gravity and the 6th term is a lift force (Saffman, 1965).

Transition Model between Micro- and Macro-Scale

The transition between micro- and macro-scale models includes two scenarios:

a. transition from collecting and coalescing microbubbles into large two-phase cavities,
b. transition from a large cavity into a set of micro-scale dispersed bubbles.

In the first scenario a criterion based on bubble / cell relative size is set to “activate” bubbles for computation of a level set function:

\[
R \geq \max(R_{thr}, m_{thr} \Delta L),
\]

where \( R_{thr} \) is a threshold bubble radius, \( \Delta L \) is the size of the local grid which hosts the bubble, and \( m_{thr} \) is a threshold grid factor. Therefore a bubble interface is switched to be represented by the zero level set when the bubble radius exceeds a threshold value and becomes larger than a multiple of the local grid size.

At the beginning of the simulation where no cavity exists, all cells have the same level set value, \( \varphi_{LSB} \), fixed to be a very large negative value. When bubbles are detected to satisfy (15), the level set value of each cell in the computation domain is modified based on the distance from the cell center to the bubbles’ surfaces, using the following equation:

\[
\varphi_i = \min(\varphi_{LSB} - \varphi_{bc}), \quad j = 1, N_i,
\]

where \( \varphi_{bc} \) is the local distance function computed based on the surface of bubble \( j \), and \( N_i \) is the number of bubbles which are “activated” around cell \( i \).

This scheme allows multiple bubbles to merge together into a large cavity, or isolated bubbles to coalesce with a previously defined large cavity.

In the second scenario, we use the zero level-set to identify the gas/liquid interface of the cavity. The volume of the cavity is tracked at each time step to determine when the cavity collapse occurs. As the cavity collapses, surface instabilities and cavity disintegration are accounted for using empirical criteria based on experimental observations. An amount of micro-bubbles of the same volume is initiated around the cavity to replace volume loss between time steps. The procedure is similar to that used to simulate bubble entrainment in breaking waves. Micro-bubbles are entrained into the liquid once the speed of the gas/liquid interface exceeds the local normal velocity. This leads to a simple expression for the location and rate of bubble generation that is proportional to the liquid’s local turbulent kinetic energy times the gradient of the liquid velocity in the liquid normal direction. The procedure is similar to that in Ma et al., 2011 and Hsiao et al., 2013.

Figure 3 illustrates the basic transition concepts between macro and micro scale.
theoretical studies, we used in the present work a molecular model for weak spots, which are the source for air pockets. One can expect that clusters of such insoluble hydrocarbon chain) and capture nanoscopic water-soluble group attached to a nonpolar water-amphiphilic molecules (molecules having a polar solid surfaces submerged in a liquid tend to absorb also be used to describe the wall gas bubbles because that proposed by Yount (1979) for dispersed nuclei can more depth and suggested that a skin model similar to the proposed by Youn (1979) for dispersed nuclei can also be used to describe the wall gas bubbles because solid surfaces submerged in a liquid tend to absorb amphiphilic molecules (molecules having a polar water-soluble group attached to a nonpolar water-insoluble hydrocarbon chain) and capture nanoscopic air pockets. One can expect that clusters of such molecules form weak spots, which are the source for bubble nucleation from the solid surface when low pressure conditions are achieved. Briggs (2004), for instance, found that scrupulous cleaning was decisive for obtaining a high tensile strength of water, and that contamination of interfaces is a primary factor in cavitation. Atchley and Prosperetti (1989) modeled nucleation out of crevices with liquid pressure drop. They identified unstable motion of the liquid-solid-gas contact line in the crevice and unstable growth of the nucleus volume due to loss of mechanical stability (force balance). Specifically, they related the threshold pressure to crevice aperture, surface tension, and gas concentration etc.

Based on existing experimental observations and theoretical studies, we used in the present work a nucleation model with the following characteristic parameters:

- $P_{th}$: a nucleation pressure threshold,
- $N_c$: a number density of nucleation sites per unit area,
- $f_n$: a nucleation time emission rate, function of the local flow conditions,
- $R_0$: an initial nuclei size, function of the local flow condition (several values can be used).

In this nucleation model, at each time step, nuclei are released from the rigid boundary computation cells when the pressure at a cell center drops below $P_{th}$. The cell releases $N$ nuclei during the time interval $\Delta t$, obtained as follows:

$$N = N_c f_n \Delta t \Delta A,$$

where $\Delta A$ is the surface area of the grid cell.

Since the Lagrangian solver for bubble tracking usually requires time steps much smaller than the Eulerian flow solver, the $N$ nuclei can be released randomly in space from the area $\Delta A$ and in time during the Eulerian time step $\Delta t$.

The above parameters are expected to also be functions of surface roughness, temperature, and other physico-chemical parameters not explicitly considered here. In this study, we use this simplified nucleation model to investigate whether it enables with the above described overall multi-scale approach to recover the main characteristics of sheet cavitation.

**SHEET CAVITATION ON 2D HYDROFOIL**

**Case Setup**

We consider first sheet cavitation over a 2D NACA0015 hydrofoil to study the sensitivity of the solution to the values of the parameters used in the numerical model. An $H-H$ type grid is generated with $181 \times 101$ grid points in the stream wise and normal direction, respectively. Only 2 grid points are used along the spanwise direction to enforce two-dimensionality. 101 grid points are used on each side of the hydrofoil surface where non-slip wall boundary conditions are imposed. The computational domain is subdivided into 12 blocks and extends three chord lengths upstream and five chord lengths downstream. The grid also extends two chord lengths vertically away from the foil surface where far-field boundary conditions are applied. The grid is selected to provide high clustering around the hydrofoil surface as shown in Figure 5. In particular, the first grid above the hydrofoil surface is such that $y^+ < 1$ in order to properly capture the boundary layer and apply turbulence modeling. Grid resolution was determined following grid convergence studies in previous work.
In this section, the foil considered at two angles of attack: 8° and 10°. The chord length is 0.12 m and the incoming flow from far-field upstream has a mean velocity of 10 m/s, resulting in a Reynolds number of $1.2 \times 10^6$. Both nuclei in the bulk of the liquid and released from the surface of the foil using the solid surface nucleation model are considered. The field nuclei have radii ranging from 20 μm to 60 μm, and correspond to an initial void fraction of $5.5 \times 10^{-8}$.

Nucleation from the solid wall is initiated when the local pressure drops below a critical value, here selected to be the vapor pressure. The initial size of the nuclei emitted is selected to be 10 μm and the number of bubbles is determined by Equation (17) with the base nucleation sites per unit area value selected to be $N_s = 224 / \text{cm}^2$ and the base nucleation frequency selected to be $f_n = 22$ kHz. The microbubbles form a cavity described by a Level Set surface when they grow larger than the size of the local grid and exceed 350 μm (~10 times of the median initial nuclei sizes). The effects of these selected values of the parameters on the solution are investigated below.

**Figure 4:** Importance of the inclusion of the wall nucleation on the proper modeling of the sheet cavity using the Eulerian-Lagrangian approach and the level set method. Field nuclei alone do not reach the wall and grow enough to fill the cavity. $\sigma = 1.65$, $\alpha = 8^\circ$, $U_\infty = 10$ m/s, $L = 0.12$ m, $N_s = 224 \text{ cm}^{-2}$, $F_n = 22$ kHz.

Comparison of the numerical results with experimental observations are very satisfactory, as described in more detail in the next section, when wall nucleation is taken into account. This is illustrated in Figure 4, which clearly shows a major difference between time-averaged cavity size when wall nucleation is included or when it is ignored. Accounting for field nuclei only is unable to capture the sheet length properly unless the void fraction is relatively high (of the order of $10^{-8}$). This is very much in agreement with the experimental observations. Considering the wall nucleation alone is capable of capturing the sheet length but is insufficient to capture properly events behind the cavity closure such as cloud cavities and development of cavitating vortical structures.

**Figure 5:** Zoomed-in view of the multi-block grid used in this study for the simulation of sheet cavitation on a NACA0015 hydrofoil.

**Dynamics of the Sheet Cavity**

A time sequence of sheet cavity shape oscillation and cloud cavitation shedding for one time cycle is shown in Figure 6 for $\sigma = 1.65$. Nucleation initiates at the hydrofoil surface near the leading edge where the pressure drops below the selected nucleation threshold, $p_{th} = p_v$. Initially, bubble nuclei from the bulk of the liquid being convected downstream, as well as nuclei released from the solid surface, grow, coalesce and form a discrete small cavity described with a level set function. Later, free field nuclei have difficulty to enter in the cavity and just move downward, while wall nucleation keeps feeding vapor and gas into the sheet cavity which grows to become a fully developed sheet cavity.

At the back of the cavity a stagnation flow forms (Shen and Dimotakis 1989) as shown in the zoomed view in Figure 7. The reverse flow results in a re-entrant jet, which moves upstream under the sheet cavity and ultimately contributes to breaking the cavity into two. The shed cavity collapses as it travels downstream generating high pressures. At the same time a new sheet cavity forms and expands, and the cycle repeats.

This periodic shedding phenomenon can be further quantified by analyzing the time histories of the instantaneous pressures and stream-wise velocities in the flow. For example Figure 8 shows the pressures and velocities monitored at the first grid point in the liquid on the suction side of the solid surface at the location $x = 0.4$ L downstream of the leading edge. This is at about the same location as the maximal extent of the trailing edge of the fully developed sheet cavity. This location alternatively experiences positive and negative stream-wise velocities with magnitudes as high as the incoming flow, $U_\infty$. The time interval between consecutive negative velocity maxima is repeatable and
has a value close to $1.15 \frac{L}{U_\infty}$. The corresponding oscillation frequency is about $0.86 \frac{U_\infty}{L}$, which matches very well with the experimentally measured shedding frequency for the same cavitation number (Berntsen et al. 2001).

Figure 6: Time sequence of sheet cavitation formation and cloud shedding with pressure contours. ($\sigma=1.65$, $\alpha=8^\circ$, $U_\infty=10$ m/s, $L = 0.12$ m). The white dots are discrete bubbles and the white lines are the zero level-set lines denoting the liquid-vapor interface.

Figure 8 also shows that each cycle of flow velocity oscillations coincides with the alternation of peaks and minimum pressure values with the same oscillation frequency. This illustrates a strong correlation between reentrant jet initiation and the development of pressure fluctuations during each cycle of the sheet cavitation.

Figure 7: Zoom views of the velocity field and pressure contours around the sheet cavity (Top) and a zoom view of the red-boxed region of the streamlines in the reverse re-entrant jet flow region (Bottom).

Figure 8: Time histories of the pressures and streamwise velocities at $x = 0.4L$ at the solid wall near the sheet cavity trailing edge. $\sigma = 1.65$.

Sensitivity of Results to Model Parameters

Nucleation Model Parameters

The effects of the selection of the numerical values of the nucleation model parameters on the simulation results are examined in this section. The nucleation parameters, $f_0$ and $N_0$, which express the number density and frequency of the nuclei emitted from the wall, as shown in Equation (17), are investigated.
Figure 9: Effect of varying \( f_n \) (a) and \( N_s \) (b) on the time-averaged cavity shape for \( \sigma = 1.65 \).

Figure 9a compares the time-averaged sheet cavity shapes for different assumed nucleation rates, \( f_n \), and for the same value of the number of nucleation sites per unit area, \( N_s = 224 \, \text{cm}^{-2} \). Figure 9b, on the other hand, compares the averaged shapes for different assumed nucleation sites per unit area, \( N_s \), for the same value of the nucleation rate, \( f_n = 22 \, \text{kHz} \). The same information is shown in quantitative form in Figure 10. It is seen that the time-averaged cavity length increases with the nucleation rate when \( f_n \) is small, but then remains almost unchanged for values of \( f_n \) larger than 22 kHz. A similar trend can be seen when \( N_s \) is increased for a constant nucleation frequency, \( f_n \).

The time-averaged cavity length increases with the number density, \( N_s \), when \( N_s \) is small, but then remains almost unchanged for values of \( N_s \) larger than 112 cm\(^{-2} \). These trends imply that when the nucleation rate and the number of nucleation sites per unit area are larger than some values, the cavity length reaches a saturation status and shows negligible dependency on those parameters. Since these two parameters are associated with surface roughness, the implication of the results is that surface treatment of the lifting surface can significantly influence the development of sheet cavitation.

The same can be seen in Figure 10 concerning the effect of the selection of the initial radius, \( R_0 \), of the bubbles nucleated from the wall. For the whole range of radii tested between 10 \( \mu \text{m} \) and 100 \( \mu \text{m} \), both the cavity length and the frequency of oscillations remain approximately the same.

**Transition Model Parameters**

In the model for transition between micro-scale and macro-scale, criteria are introduced to switch discrete singularity bubbles to non-spherical deformable cavities represented by the zero level set.

Figure 10: Effects of varying the wall nucleation parameters \( f_n \), \( N_s \), and \( R_0 \), on the normalized time-averaged cavity length (a), and on the normalized cavity shedding frequency (b) for \( \sigma = 1.65 \). (\( f_{n,0} = 22 \, \text{kHz}, N_{s,0} = 224/\text{cm}^2, R_{0,0} = 50 \, \mu \text{m} \)).

In order to demonstrate the generality of these criteria, a parametric study is conducted on the effects on the solution of the choice of the threshold values \( R_{thr} \) and \( m_{thr} \) in Equation (15). Figure 11 compares the time-averaged sheet length and the shedding frequency for different assumed threshold bubble radii, \( R_{thr} \), with the same value of the mesh factor, \( m_{thr} = 1.0 \). It is seen that the time-averaged cavity length increases as the threshold bubble radius increases when \( R_{thr} \) is small, but then remains almost unchanged when \( R_{thr} \) exceeds 200 \( \mu \text{m} \). The same figure also shows little dependency on the threshold grid factor, \( m_{thr} \), for a constant \( R_{thr} = 350 \, \mu \text{m} \). The same conclusions apply for the predicted shedding frequency. This strongly indicates little dependency of the results on the switching criteria as long as the switching of the bubbles to a cavity is not done too early, i.e. when the bubbles have not grown significantly.
Figure 11: Effects of varying the threshold values, $R_{thr}$ and $m_{thr}$, in the switching from singular bubbles to large cavities, on the normalized time-averaged cavity length (a), and on the normalized cavity shedding frequency (b) $\sigma = 1.65$, $R_{thr,0} = 350 \mu m$, and $m_{thr,0} = 1$.

Validation against Experimental Measurements

In the following, the results of the present model are compared against published experimental data for different cavitation numbers and angles of attack. To do so, the cavitation number is varied in the computations by adjusting the far field pressure, $p_\infty$. Figure 12 compares the instantaneous pressures and streamwise velocities at $x = 0.4 L$ for $\sigma = 1.65$ and $\sigma = 1.50$. Similar periodic oscillations of the local pressure and reversal of jet direction are seen in both cases. However, for $\sigma = 1.50$ most of the time the streamwise velocity remains negative, but from time to time it is interrupted by sharp positive peaks. This is because at this lower cavitation number the cavity extends further downstream than $x = 0.4 L$. As a result, during most of the time this location is below the cavity and sees the reverse re-entrant jet. The sharp peaks appear when the sheet cavity shrinks due to the periodic shedding. This can be further explained by the pressure curves in Figure 12b. The pressure mostly stays at the vapor pressure, $C_v = -1.5$, with repeated peak pressure increases at a nearly constant frequency. The observation point for the $\sigma = 1.65$ case does not see vapor pressure but still senses the repeated pressure peaks at a frequency slightly higher than that for $\sigma = 1.50$. This is made apparent in Figure 13, which shows the frequency spectrum of the signal through a Fast Fourier Transform (FFT) analysis of both the re-entrant jet velocity and the pressure oscillations.

Quantitative comparisons of these frequencies with the experimental data, which measured cloud shedding frequencies for three different cavitation numbers, $\sigma = 1.50$, 1.65, and 1.80 and two different angle of attacks, $\alpha = 8^\circ$ and $10^\circ$, are displayed in Figure 14. The results are presented in terms of the non-dimensional shedding frequency, $f_{sheet} L/U_\infty$, versus the ratio of the cavitation number and the angle of attack, $\sigma 2 \alpha$. A very good agreement can be seen between the numerical predictions and the experimental measurements in the simulated range.

Figure 12: Comparison for $\sigma = 1.65$ and $\sigma = 1.50$ of the time histories of the streamwise velocity (a) and of the instantaneous pressure (b) monitored at $x = 0.4 L$ on the NACA 0015 foil. $\alpha = 8^\circ$, $U_\infty = 10$ m/s, $L = 0.12$ m.
APPLICATION TO 3D HYDROFOIL

To test the capability of the developed multi-scale model to the simulation of sheet and cloud cavitation on 3D lifting surfaces we present below 3D simulation cases with different levels of complexity.

3D Computation of a NACA0015 Foil

3D computations on a hydrofoil having the same foil shape as the above 2D NACA0015, but with a span equal to one quarter of the chord length are considered. The foil uses the same computational domain and grid resolution as for the 2D computations except that 20 grid points are used here in the span-wise direction which is bounded by two side walls. The same boundary conditions as in previous sections are applied on the far field boundaries and slip boundary conditions are imposed on both side walls in the 3D simulations.

Figure 13: Comparison between the cases of $\sigma = 1.65$ and $\sigma = 1.50$ of the frequency spectra of the temporal variations of the streamwise velocity (a) and of the pressure (b) monitored at $x = 0.4 \, L$ at the solid wall on the suction side.

Figure 14: Comparison of the shedding frequencies of the sheet cavity on the NACA0015 for $\alpha = 8^\circ$, $U_\infty = 10 \, \text{m/s}$, $L = 0.12 \, \text{m}$ with the experimental results summarized in Berntsen et al. 2001.

In addition to the shedding frequency, the time-averaged cavity lengths deduced from the unsteady simulations are also compared to the experimental measurements. Figure 15 illustrates the time-averaged cavity shape for $\sigma = 1.65$ and $\sigma = 1.50$ by averaging the solution from the unsteady simulations after excluding the initial transient period. Figure 16 shows a comparison with the results summarized in Berntsen et al. 2001. The figure shows very good agreement between the present numerical predictions and the experimental measurements for the time-averaged cavity length for three different cavitation numbers, $\sigma = 1.50$, 1.65, and 1.80 and two angle of attacks, $\alpha = 8^\circ$ and $10^\circ$.

Figure 15: Comparison of the time-averaged cavity shapes between $\sigma = 1.65$ and $\sigma = 1.50$ for the NACA 0015 hydrofoil. $\alpha = 8^\circ$, $U_\infty = 10 \, \text{m/s}$, $L = 0.12 \, \text{m}$.

Figure 16: Comparison of the time-averaged cavity length on the NACA0015 for $\alpha = 8^\circ$, $U_\infty = 10 \, \text{m/s}$, $L = 0.12 \, \text{m}$ with the experimental results summarized in Berntsen et al. 2001.
A time sequence of sheet cavity development, oscillations, and bubble cloud shedding on the foil for $\sigma = 1.65$ and $\alpha = 8^\circ$ is shown in Figure 17 and Figure 18. It is seen that although the cloud shedding dynamics is similar to that observed in the 2D computations, i.e. the sheet cavity starts near the leading edge and gradually evolves until a reentrant jet breaks the cavity into parts and forms a detached bubble cloud, the 3D computations exhibit a more complicated breakup, i.e. the cavity breakup location varies in the spanwise direction due to the 3D nature of the unsteady flow.

![Figure 17](image1.png)

**Figure 17:** Time sequence of 3D sheet cavity dynamics on a NACA0015 hydrofoil for $U_\infty = 10\text{m/s}$, $L = 0.12\text{ m}$, $\sigma = 1.65$, $\alpha = 8^\circ$.

**Oscillating Finite-Span Hydrofoil**

**Case Setup**

In this section we consider an oscillating finite-span hydrofoil for which experimental data is available from Boulon (1996). This concerns an elliptic hydrofoil having a NACA16020 cross-section and a root chord length of $L = 6\text{ cm}$ with an aspect ratio of 1.5 (based on semi-span). In order to simulate the hydrofoil oscillations in an overall fixed fluid frame, a moving overset grid scheme is applied to a small region enclosing the foil. As illustrated in Figure 19 a 12-block grid is generated to encompass the finite-span foil in a domain bounded by boundaries located at about one chord length away from the foil surface. This 12-block grid is then overset over a fixed background grid which has domain boundaries located at five chord lengths away from the foil surface as shown in Figure 20. A total of 0.25 million grid points was generated for the overset grid with a first grid spacing of $1 \times 10^{-4}$ chord length in the normal direction to the foil surface.

![Figure 18](image2.png)

**Figure 18:** 2D projected side views of a time sequence of a 3D simulation of an oscillating sheet cavity on a NACA0015 hydrofoil for $\sigma = 1.65$, $\alpha = 8^\circ$, $U_\infty = 10\text{m/s}$, $L = 0.12\text{ m}$. 
Single-Phase Flow Computations

We present first the case of the hydrofoil in a uniform flow field at a velocity of 3.33 m/s oscillating with a time-dependent angle of attack, $\alpha(t)$, prescribed by the sinusoidal function:

$$\alpha(t) = \alpha_0 + \Delta \alpha \sin(2\pi ft) , \quad (18)$$

where the initial angle of attack $\alpha_0$ is 10°. The oscillations amplitude is $\Delta \alpha = 5^\circ$ and the oscillations frequency is $f = 23$ Hz. The single phase flow field was first simulated until the solution reached limit cycle oscillations. Figure 21 shows the time histories of the pressure coefficient monitored at three locations, $x = 0.04L$ (near leading edge), $x = 0.5L$ (middle chord), and $x = L$ (at the trailing edge). It is seen that the pressure has reached a repeated limit cycle oscillation at the leading edge and the mid chord locations. However, the pressure at the trailing edge location still shows irregular vortex pairing occurring upstream.

Figure 22 shows pressure contours and the iso-pressure surface $p=p_v$ for one oscillating cycle. It is seen that a low pressure region, where the flow separation occurs, starts to form near the leading edge as the angle of attack increases during the cycle. The low pressure region extends downstream and reaches a maximum length before the hydrofoil reaches the highest angle of attack. The length of the low pressure region is reduced but its height increased when the hydrofoil reaches the maximum angle of attack. Then, the low pressure region starts to shrink as the angle of attack is decreased from the highest value and the low pressure region reduces to its smallest size when the angle of attack reaches the lowest value. Strong well defined vortical regions are seen to propagate downstream from the leading edge.

Simulation of Sheet and Cloud Cavitation

Two-phase simulations to capture sheet and cloud cavitation were initiated once the single-phase flow solution reached limit cycle oscillation. Free nuclei were allowed to propagate in the liquid, while nucleation was activated at the foil boundary as in the 2D study shown earlier. Figure 23 shows a time sequence of the sheet cavitation evolving on the suction side of the finite-span hydrofoil for $\sigma = 0.9$.

A zoom on the root sheet cavity near the leading edge is also shown in Figure 24 for better illustration. It is seen that the sheet cavity starts to form from the leading edge near the tip region. It then extends toward the root as the angle of attack increases. However, unlike the low pressure region observed in the single-
phase flow solution, the sheet cavity does not reach its maximum extent before the hydrofoil reaches the highest angle of attack. On the contrary, the sheet cavity continues to grow beyond the time the foil reaches the highest angle of attack. After reaching a maximum extent, the sheet cavity loses volume without showing clear cloud shedding in this case. This could be due to the grid resolution used not being adequate. Simulations with much finer grids will be conducted for better prediction of the sheet cavity.

Figure 22: A time sequence of the pressure contours and an iso-pressure surface ($\rho = \rho_v$) during one oscillating cycle of the finite span NACA16020 elliptic hydrofoil after the unsteady solution reached limit cycle oscillation.

Figure 23: A time sequence of sheet cavity evolving on the suction side of an oscillating finite-span hydrofoil obtained for $\sigma = 0.9$.

Figure 25 shows a comparison of the time histories of the pressure coefficient monitored at $x = 0.04L$ (near the leading edge) and $x = 0.5L$ (mid-chord) between the single-phase and the two-phase flow computations. The modifications of the pressure due to the sheet cavity can be clearly seen at both locations. For the location near the leading edge, the pressure at the hydrofoil surface becomes equal to the vapor pressure ($C_p = -0.9$) once the sheet cavity reaches this location. Pressure spikes are also generated, as in the 2D computations, due to the local breakup of the sheet cavity. For the location at mid-chord, the pressure is also changed by the presence of the unsteady cavity. However, the pressure does not reach vapor pressure because the sheet cavity length does not reach this location.
Figure 24: Zoom views of the root sheet cavity near the leading edge obtained for $\sigma = 0.9$.

CONCLUSIONS

A multi-scale framework was developed for the simulation of unsteady sheet cavitation on hydrofoils by smoothly bridging a Level Set method for large size cavities and a Discrete Singularity Model for small bubbles. Starting with the micro-scale physics of nucleation from the solid surface and of dispersed nuclei in the bulk liquid, sheet cavitation along a NACA0015 hydrofoil is well captured by the model. The predicted sheet length and oscillation frequency under several cavitation numbers match experimental data very well. Parametric studies indicate that when the nucleation rate and the number of nucleation sites per area are not too small, the cavity characteristics show negligible dependency on those parameters.

Figure 25: Comparison of time histories of pressure coefficients monitored at $x = 0.04 \, L$ (near the leading edge) in (a) and at $x = 0.5 \, L$ (mid-chord) in (b) for single phase and two-phase flow computations.

The criteria to switch from singular microbubbles to large resolved cavities also show little dependency on the selected parameters as long as the limit size for transforming a nuclei bubble into a large cavity is not too small.

The developed multi-scale two-phase flow model was also applied to simulate sheet and cloud cavitation on a 3D lifting surface. This part of the work is still ongoing. Three-dimensional simulations for the 2D NACA0015 foil capture well sheet cavitation dynamics and cloud shedding with a more complicated breakup observed due to the 3D unsteadiness of the flow.

Sheet cavitation on an oscillating finite-span hydrofoil was also simulated using a moving overset grid scheme. Although the unsteady development of the sheet cavity was also captured, the results did not show clearly cloud shedding, probably due to inadequate grid resolution used in these simulations. This will be improved by refining the grids and conducting longer duration simulations.
ACKNOWLEDGMENTS

This work was supported by the Office of Naval Research under contract N00014-12-C-0382 monitored by Dr. Ki-Han Kim. This support is highly appreciated.

REFERENCES


