EFFECT OF VORTEX/VORTEX INTERACTION ON BUBBLE DYNAMICS AND CAVITATION NOISE

Chao-Tsung Hsiao
DYNAFLOW, INC.
ctsxung@dynaflow-inc.com

Geroges L. Chahine
DYNAFLOW, INC
glchahine@dynaflow-inc.com

ABSTRACT
The effect of vortex/vortex interaction on bubble dynamics and cavitation noise is investigated using a Eulerian-Lagrangian approach. For the liquid phase direct numerical simulation of the Navier-Stokes equations is used to simulate two unequal co-rotating vortices. A bubble dynamics model is employed to track the nuclei for the disperse phase flow and is coupled with the Navier-Stokes computation at each time step. Two cases of different relative vortex strengths are selected to study the interaction between the vortices. The unsteady flow structure due to the interaction is described in detail. The effect of unsteady flow filed on the bubble dynamics is shown to be important and could lead to a probability issue on the cavitation event.

INTRODUCTION
Prediction of cavitation inception on navy propulsors is a very challenging task that has preoccupied propulsor designers for many years. This is in fact true for predictions using scaled experimental tests as well as for predictions based on analytical/numerical modeling. Although novel sophisticated measurement techniques have allowed one to reveal detailed velocity field, the pressure field still needs to be deduced based on some assumptions. Furthermore, these techniques quantify mainly the space variations of the flow field using some time and space averaging. Therefore, the pressure field may not reflect the real instantaneous pressure field that the nuclei encounter as cavitation inception events occur.

Recent observations [1] of cavitation inception for ducted propellers have indicated that the interaction between tip-leakage vortex and trailing-edge vortex may cause early cavitation where the two vortices strongly interact. Several experimental studies [2,3] have also revealed that co-rotating vortex pairs strongly interact and merge into a single vortex with significantly different turbulence characteristics. The study of non-equal co-rotating vortex pairs [4]) shows an even much stronger three dimensional time-dependent flow resulting from vortex merger. The merger is preceded by a splitting of the weaker vortex into filaments that, depending on the relative strengths of the vortices, can occur in the radial direction, the axial direction, or a combination of the two.

It is important to study the effect of vortex/vortex interaction on bubble dynamics in order to predict cavitation inception under those circumstances. Although numerical simulations of vortex flows are able to provide the detailed pressure field for studying bubble dynamics, numerical studies of tip vortex flow based on Reynolds-Averaged Navier-Stokes (RANS) computations [5] have shown an over-diffusive and dissipative vortex due to grid resolution and turbulence model deficiencies. Recent numerical studies of tip-leakage vortex flow generated by a ducted propeller [6,7] also attributed under-prediction of the pressure in the vortex center to using the time-averaged method with a turbulence model.

Time-accurate simulations of the tip vortex flow have been difficult due to the geometric complexity, which often leads to highly skewed, large-aspect-ratio meshes and very small computational time step sizes. Using Direct Numerical Simulation (DNS) of the Navier-Stokes equations, Hsiao and Pauley [8] were able to qualitatively study the unsteadiness of the tip vortex due to its interaction with spanwise shedding vortices on a finite-span hydrofoil at a low Reynolds number (Re ~10^5). You et al. [9] were able to quantitatively simulate a tip-leakage vortex flow for a linear compressor cascade at a low Reynolds number using Large Eddy Simulation (LES), with an immersed boundary method to relax the constraint of geometry. However, tremendous CPU time (O(10^8) single processor hours) and memory (over 10GB) were required.

To enable the study of the effect of vortex/vortex interaction on bubble dynamics with a moderate requirement of computer resources, we have reduced geometric complexity by focusing only on the interaction and merger of two non-equal co-rotating vortices to capture the essential unsteady flow characteristics. A simple computational domain with rectangular grid allows us to DNS to simulate the evolution of two co-rotating vortices with moderate requirement of CPU time and memory.

To study bubble dynamics and cavitation inception noise small nuclei are released at the inlet of the domain. Their trajectories, volume variations, and acoustic signals are predicted using the Surface Averaged Pressure (SAP) spherical bubble dynamics model [10]. Unlike the previous studies [11,12] in which computations of bubble dynamics were conducted with a prescribed steady-state flow, in the current study the background flow through which the nuclei is convected is time-dependent. To accurately capture the effect of the instantaneous pressure field on the nuclei, the bubble dynamics model and the Navier-Stokes Solver are coupled at each time step. The statistical nuclei...
distribution model described in [12] is also used to study cavitation inception as probabilistic events.

**Numerical Method**

The current numerical method is based on an Euler-Lagrange coupled two-phase flow model in which the liquid phase flow is solved with an Euler solver while the bubble phase flow is solved with a Lagrangian scheme. Since the bubble phase is very disperse and the sizes of bubble considered here are very small (≤50 microns), the current scheme only considers the effect of liquid phase flow on the bubble phase flow and no interaction among bubbles.

**Liquid Phase Flow**

The liquid phase flow is obtained via direct numerical simulation of the Navier-Stokes Equations without turbulence modeling. The unsteady incompressible continuity and Navier-Stokes equations written in non-dimensional form and Cartesian tensor notations are given as

\[
\frac{\partial u_i}{\partial x_j} = 0, \quad \text{where } u_i = (u, v, w) \text{ are the Cartesian components of the velocity, } \frac{\partial u_i}{\partial x_j} = \frac{\partial p}{\partial x_j} + \frac{1}{\rho} \frac{\partial \tau_{ij}}{\partial x_j},
\]

where \( x_j = (x, y, z) \) are the Cartesian coordinates, \( p \) is the pressure, \( \rho \) is the density, \( \tau_{ij} \) is the effective stress tensor, \( \partial u_i/\partial x_j \) is its dynamic viscosity. The effective stress tensor \( \tau_{ij} \) is given by:

\[
\tau_{ij} = \left[ \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \frac{\partial u_l}{\partial x_l} \right].
\]

To solve Equation (1) and (2) numerically, a three-dimensional incompressible Navier-Stokes solver, DF Uncle, developed at Mississippi State University and Dynaflow, Inc. is applied. DF Uncle is based on the artificial-compressibility method [13] in which a time derivative of pressure is added to the continuity equation as

\[
\frac{1}{\beta} \frac{\partial \rho}{\partial t} + \frac{\partial u_i}{\partial x_i} = 0, \quad \text{where } \beta \text{ is the artificial compressibility factor. As a consequence, a hyperbolic system of equations is formed and can be solved using a time marching scheme. This method can be marched in pseudo-time to reach a steady-state solution. To obtain a time-dependent solution, a Newton iterative procedure needs to be performed in each physical time step in order to satisfy the continuity equation. In the present study the time-accurate solution was obtained when the maximum velocity divergence was less than 1.0x10^-3. Detailed descriptions of the numerical scheme can be seen in [14].}

**Disperse Bubble Phase Flow**

In the disperse phase flow, each bubble is tracked by a Lagrangian scheme. Here we assume that the bubble remains spherical and does not interact with each other although in other studies we are considering full bubble deformations [15,16]. Bubble transport can be modeled via the motion equation described by Johnson and Hsieh [17]:

\[
\frac{d u_b}{dt} = \frac{3}{\rho} \nabla p + \frac{3}{R} C_D (u_b - u_b) \left| u_b - u_b \right| + \frac{3}{R^2} (u_b - u_b) \dot{R}, \tag{5}
\]

where \( u_b \) is the bubble traveling velocity, \( C_D \) is the drag coefficient given by an empirical equation:

\[
C_D = \frac{24}{Re_p} (1 + 0.197 Re_p^{0.63} + 2.6 \times 10^{-4} Re_p^{1.38}). \tag{6}
\]

\( R \) is the time varying bubble radius with the bubble Reynolds number defined as

\[
Re_b = \frac{2 \rho R |u_b - u_b|}{\mu}. \tag{7}
\]

The last term on the right hand side of Equation (5) is the force due to the bubble volume variation which can be obtained by solving the Rayleigh-Plesset equation [18]. In our previous studies [10,12], a Surface Average Pressure (SAP) spherical bubble dynamics model based on the Rayleigh-Plesset equation was developed and successfully applied to study the bubble dynamics in a vortex flow. To better consider the energy loss due to acoustic emission, we have extended the SAP scheme to the Gilmore’s equation [19] and write the modified Rayleigh-Plesset equation as:

\[
(1 - \frac{\dot{R}}{c}) R \ddot{R} + \frac{3}{2}(1 - \frac{\dot{R}}{c}) R \dot{R} = \frac{3k}{4} \frac{p}{\rho} \left( \frac{R_0}{R} \right)^{3k} - \frac{p_{\text{encounter}}}{R} - \frac{2\gamma}{R} - \frac{4\mu}{R}, \tag{8}
\]

where \( c \) is the sound speed, \( R_0 \) is the initial bubble radius, \( \gamma \) is the surface tension parameter, \( p_0 \) is the vapor pressure, \( P_{\text{encounter}} \) is the initial gas pressure inside the bubble, \( p_{\text{encounter}} \) is the ambient pressure “felt” by the bubble during its travel, and \( k \) is the polytropic gas constant. \( k = 1.4 \) for adiabatic behavior is used in the present study. The first term on the right hand side is an additional pressure term resulting from SAP scheme due to the slip velocity between the liquid and bubble. In the SAP spherical model \( p_{\text{encounter}} \) is defined as the averaged liquid pressure over the bubble surface. This concept enabled more realistic modeling of the bubble dynamics in flow condition where strong pressure gradients exist.

To deduce the noise level at a distance \( l \) from the bubble center resulting from the bubble dynamics, we applied Fitzpatrick and Strasberg [20] expression to compute the acoustic pressure, \( p_a \):

\[
p_a(t') = \frac{R \rho}{l} \left[ R \ddot{R}(t') + 2 \dot{R}^2(t') \right], \quad t' = t - \frac{r - R}{c}. \tag{9}
\]
computations. Although only one-way coupling is considered in this study, a special attention to the time integration procedure is still needed due to the two different time scales involved. In the liquid phase flow the characteristic time is based on the vortex core size and the free stream velocity while the characteristic time is controlled by the bubble radius and bubble wall velocity in the bubble phase flow. For the current problem, the bubble size is much smaller than the vortex core size and the bubble wall velocity is much larger than the free stream velocity when cavitation occurs. Therefore, the time step size of the bubble phase flow is smaller than that of the liquid phase flow. In the current study, a non-dimensional time step size $\Delta t = 0.02$ is applied to the liquid phase flow while $\Delta t = 0.0004$ is required in the bubble phase flow for the smallest bubble size equal to 5 microns. This leads to 50 time-integration steps in the bubble phase flow for each physical time step in the liquid phase flow.

The numerical solution of the Euler solver, offers the solution only at the grid points. To obtain the values for any specified location $(x,y,z)$ on the bubble we need to interpolate from the background grid. The detailed numerical scheme for locating the interpolation stencil and coefficients can be seen in [12].

**Flow Configuration**

We consider two co-rotating vortices with different vortex core radii, $a_{c1}$ and $a_{c2}$, initially located apart at a distance, $d=6a_{c2}$, as shown in Figure 1. A rectangular computational domain is used to simulate the evolution of the two co-rotating vortices with the center of the bigger size vortex located at the origin of the $Y-Z$ plane. The computational domain is completed by extending a square cross plane to $80a_{c2}$ downstream. The size of each square cross plane is set to be $30a_{c2}$ away from the center such that the free stream condition can be applied at the far-field boundary. This size is also presumed to be large enough for neglecting the influence of the boundary condition on the interaction between two vortices.

![Figure 1. Sketch of flow configuration for two unequal co-rotating vortices.](image)

A rectangular grid with locally refined in the region of the two vortices is generated for this simulation. The grid has a total of 2 million grid points in which $201 \times 101 \times 101$ grid points were created in the streamwise direction and two cross plane directions respectively. This grid results in a uniform grid size of $0.4 \ a_{c2}$ in the streamwise direction and $0.2a_{c2}$ in both cross directions for the center fine grid region as shown in Figure 2.

![Figure 2. A close-up center view of the fine grid region.](image)

The initial condition for the unsteady computation of the liquid phase flow is specified with a uniform streamwise velocity plus circumferential velocity and pressure obtained by a superposition of two parallel Lamb-Oseen vortices, i.e. axisymmetric two-dimensional vortices with a Gaussian vorticity distribution. The method of characteristics for the boundary condition with all three components of velocities specified as the initial values is applied to the inlet boundary while a first-order extrapolation for all variables is applied at the outlet boundary.

For the bubble phase flow, bubbles start to be released at the inlet boundary when the unsteady computation of the liquid phase passes the startup transient stage. The amount of bubbles to be tracked at each time step and the initial bubble sizes are determined based a statistical nuclei distribution model. In this model, we consider a liquid with a known nuclei number density distribution, $N(R)$, where $R$ is the nuclei size and $N$ is the nuclei number density in $m^4$. This can be obtained from experimental measurements such as light scattering, cavitation susceptibility measurements [21], and can be expressed as discrete distribution of $M$ selected nuclei sizes. The total void fraction, $\alpha$, in the liquid can be written as

$$\alpha = \sum_{i=1}^{M} \frac{4\pi R_i^3}{3} n_i,$$

where $n_i$ is the number of nuclei of size $R_i$ in a unit volume of liquid. The discrete nuclei distribution $n_i(R_i)$ is used to determine the total number and size of the nuclei to be used in the computations. The position and timing of injection of the nuclei at the release plane are obtained using random distribution functions. The size of the release area and the total release time is determined such that the overall void fraction is equal to the ratio of the total bubble volume to a fictitious volume that is deduced by the size of the release area and a length scale equal to $V_\infty \Delta t$, where $V_\infty$ is the free stream velocity and $\Delta t$ is the total release time (or signal acquisition time).
RESULTS

Unsteady two-unequal co-rotating vortices

Two cases of two-unequal co-rotating vortices were simulated in the current study. The Reynolds number $Re=6.7 \times 10^4$ is based on the free stream velocity ($u^*=V_\infty^*$) and the small vortex radius ($L^*=a_{c1}^*$) for both cases. In both cases the separation distance, $d=6a_{c2}$, the ratio of vortex core size, $a_{c1}/a_{c2}=2$, and the non-dimensional circulation ($\Gamma=\Gamma^*/u^*/L^*$) of the bigger size vortex, $\Gamma_1=18.85$, remain the same while the non-dimensional circulation of the small size vortex, $\Gamma_2$, is specified to be 9.42 and 6.28 respectively. The results of both cases were obtained by integrating the solution until non-dimensional time $T=t^*L^*/u^*=200$. This time interval allows the flow to travel a distance more than twice of the domain length in the streamwise direction ($X_{max}=80$).

To illustrate the vortex structure for the simulated results, iso-pressure surfaces are plotted at different values because this is similar to visualize cavitating vortices at different cavitation numbers. Figures 3 and 4 shows the iso-pressure surfaces at different values at time step $T=150$. From the figures, it is seen that the pressure along the bigger size vortex ($a_{c1}$), the one located below, is lower than along the smaller size vortex ($a_{c2}$) in the $\Gamma_2=6.28$ case. The smaller size vortex seems to merge into the bigger size very rapidly in less than one rotation orbit. The merger leads to oscillations in the pressure along the center of the bigger size vortex. On the contrary, in the $\Gamma_2=9.42$ case the smaller size vortex has a lower pressure distribution before the merger and the merging procedure is much more complicated. It is seen that both vortices seem to break into filaments before they completely merge and result in a much stronger interaction than in the previous case.

The time sequence of vortex structure variations for the $\Gamma_2=9.42$ case is shown in Figure 5. The iso-surface of $Cp=-2.8$ and pressure contours at different streamwise locations are plotted at three different time steps, $T=100, 150, 200$, to illustrate unsteady interaction between the two vortices. From the figure, highly unsteady variations are observed for vortex structure behind $X=24$ due to the vortex merger. Another way to illustrate the pressure distribution due to the interaction is to track the local minimum pressure coefficient, $Cp$, along the streamwise direction. Figures 6 and 7 show the local minimum $Cp$ curves at three time steps for both cases. It is seen that the pressure reaches a minimum value at about $X=24$ for the $\Gamma_2=9.42$ case and at about $X=8$ for the $\Gamma_2=6.28$ case. In both cases the minimum pressure seems to occur before these two vortices merge completely. However, it is interesting to note that a second prominent pressure drop seems to occur after merger in the $\Gamma_2=9.42$ case. Although the second pressure drop is not lower than the first drop in the current simulation, it is possible that different setups of flow conditions to be considered in the future could lead to a stronger pressure drop after merger.

It is also important to examine the flow field near the location where the pressure reaches the minimum value in order to understand its mechanism. Figure 8 shows contours of pressure, streamwise vorticity and velocity at four different streamwise locations near $X=24$ for the $\Gamma_2=9.42$ case. It is seen that the pressure reaches its minimum when the vorticity of weaker vortex is spread and sucked into the stronger vortex at $X<24$. This interaction also results in an acceleration of the flow and lead to a maximum streamwise velocity in the vortex center at the same streamwise location.

For the bubble phase flow, the bubble transport and dynamics equations are solved in dimensional forms. To enable the coupling, all flow parameters obtained from the liquid phase flow are dimensionalized based on a characteristic length and a characteristic velocity. In this study a vortex core radius of 0.0067m and a free stream velocity of 10m/s are specified for the characteristic length and velocity. All water properties used in the dimensional equations are defined at 20°C.
Figure 6. Local minimum pressure coefficients for $\Gamma_z=9.42$.

Figure 7. Local minimum pressure coefficients for $\Gamma_z=6.28$.

Figure 8. Contours of pressure, streamwise vorticity and velocity (from top to bottom) at four different streamwise locations $X=20$, 22, 24 and 26 coefficients for $\Gamma_z=9.42$.

From previous studies [11,12], we know that only nuclei that enter a “window of opportunity” are actually captured by the vortex and generate strong acoustic signals. From a single line vortex flow study, the window was found to be a circular-shape area with its center coinciding with the vortex center at the inlet boundary. Since there are two distinguished vortices at the inlet boundary in this study, the window shape, size and location could be very different from the single line vortex flow.

From the analysis of two-unequal co-rotating vortices as discussed in the previous section we have found that the minimum pressure could appear at different locations depending on the relative strength of the two vortices. To establish the “window of opportunity” for both cases, a rectangular release area was specified at the inlet boundary with 375 nuclei of a given size released from a 15×25 grid points. The cavitation number was specified high enough ($\sigma=6$ for this study) such that the maximum growth size of nucleus was less than 10%. Every nucleus was tracked and the minimum pressure coefficient encountered by each nucleus during its travel was recorded at its release grid point. This enables us to plot a contour of the minimum encounter pressure coefficient at the release grid points and to illustrate the “window of opportunity” for each case.

Since the flow is unsteady, the actual “window of opportunity” is time-dependent and could not be determined easily. Here the “window of opportunity” was determined by selecting the solution at a certain time step and assuming a steady...
flow during the computation. Figure 9 shows contours of minimum encounter pressure coefficient for four different nuclei sizes, \( R_0 = 5, 10, 20, 50 \) microns, which were released at \( T = 100 \) for the \( \Gamma_2 = 9.42 \) case. From the figure it is seen that the shape of the “window of opportunity” is like a two-arm spiral. The release area for the nuclei to encounter lower pressure shrinks to a region close to the center of the smaller size vortex as the nuclei size is reduced. It is important to note that there is a region near the center of the bigger size vortex from which even larger size nuclei do not encounter the global minimum if released. This is because the nuclei released from this region are captured by the bigger size vortex and miss the global minimum pressure which appears in the smaller size vortex before merger.

To illustrate the effect of the unsteady flow on the “window of opportunity”, similar computations were also conducted using the \( T = 150 \) and \( 200 \) flow fields for the \( \Gamma_2 = 9.42 \) case. Comparison of the “window of opportunity” between different time steps for the nuclei size \( R_0 = 50 \) microns is shown in Figure 10. It is seen that the nuclei miss encountering the lowest \( Cp_{\text{min}} \) level \((Cp_{\text{min}} = -5.2)\) if released at \( T = 150 \), while a very large window led the nuclei to encounter the \( Cp_{\text{min}} \leq -5.0 \) for the \( T = 100 \) flow field.

The contours of minimum encounter pressure coefficient for the \( \Gamma_2 = 6.28 \) case with different nuclei size \( R_0 = 5 \) and 50 microns are shown in Figure 11. By comparing the result with Figure 9, one can see that the “window of opportunity” for the \( \Gamma_2 = 6.28 \) case is very different from that of \( \Gamma_2 = 9.42 \). The release region that led the nuclei to encounter the minimum pressure is around the center of the bigger size vortex instead of the smaller size vortex.

**Figure 9.** Contours of encounter \( Cp_{\text{min}} \) for nuclei with \( R_0 = 5, 10, 20, 50 \) microns, released at \( T = 100 \) for \( \Gamma_2 = 9.42 \).

**Figure 10.** Contours of encounter \( Cp_{\text{min}} \) for nuclei with \( R_0 = 50 \) microns released at \( T = 150 \) and \( 200 \) for \( \Gamma_2 = 9.42 \).

**Figure 11.** Contours of encounter \( Cp_{\text{min}} \) for nuclei with \( R_0 = 5 \) and 50 microns released at \( T = 100 \) for \( \Gamma_2 = 6.28 \).

**Bubble Trajectories and Cavitation Noise**

The unsteady computation of the liquid phase flow alone was conducted until \( T = 100 \). Afterward, the coupling of the two-phase flow computation was activated and the nuclei started to be released at the inlet boundary. The coupling computation was conducted until \( T = 200 \) and gave a dimensional time interval of 0.067 seconds for recording acoustical signals. In the present study we consider a nuclei size distribution ranging from 5 to 50 microns with a void fraction \( \alpha = 3.4 \times 10^{-8} \). A 0.08m\( \times 0.12 \)m rectangular area on the inlet boundary was used to release the nuclei. This resulted in a total of 568 nuclei as shown in Figure 12. The acoustic pressure emitted from all nuclei was computed and superposed at each time step.

**Figure 12.** The number of nuclei released versus nuclei size.
To ensure having enough cavitation events during the time interval of unsteady coupling, a series of computations with different cavitation numbers were conducted for both cases at $T=100$ by assuming a steady flow during the time interval. Although the steady computation is expected to get a different signal from the unsteady computation, it offers a good estimation of cavitation number to be studied in the unsteady computation. As a result, $\sigma=5.15$ and 4.30 were chosen for the $\Gamma_2=9.42$ and 6.28 cases respectively. Figure 13 shows the acoustic signals obtained with the cavitation number $\sigma=5.15$ for the $\Gamma_2=9.42$ case in which 6 cavitation events are observed during the time interval. Unsteady effects on the acoustic signal can be seen by comparing the signals of the unsteady computation as shown in Figure 14 to those of the steady computation at $T=100$. The number of cavitation events and the time for the event to occur are different. A same steady computation was conducted at $T=150$ with $\sigma=5.15$ for $\Gamma_2=9.42$ but no cavitation event was found. Figure 14 also shows the acoustic signals obtained from unsteady coupling for $\Gamma_2=6.28$. As expected, the $\Gamma_2=9.42$ case has a much higher cavitation inception number because the time-average $-C_{p_{\min}}$ is equal to 5.21 and 4.52 for the $\Gamma_2=9.42$ and 6.28 cases respectively. Although the $\Gamma_2=9.42$ case has smaller deviation between the cavitation number and time-average $-C_{p_{\min}}$, it still has more cavitation events than the $\Gamma_2=6.28$ case. This is because the minimum pressure of the $\Gamma_2=9.42$ case appears further downstream which allows more nuclei to be captured and encounter the minimum pressure.

Two different ways are used to illustrate bubble trajectories for the current simulations. Figure 15 shows instantaneous bubble locations related to the two vortices which is illustrated by iso-$C_p$ surfaces at two values ($C_p=-2.8$ and $-5.0$) at four different time steps for the $\Gamma_2=9.42$ case. The bubble size is amplified 5 times to enhance the visualization of small nuclei. From the figure it can be clearly seen that a cavitation event is occurring near the location of the minimum pressure at time = 0.05 and 0.067 seconds. From the acoustic signal we have identified there are seven cavitation events for the $\Gamma_2=9.42$ case. Bubble trajectories for these seven nuclei are plotted in Figure 16. This figure again reinforces the discussion in the previous section that only nuclei released near the smaller size vortex encounter the minimum pressure and cavitate for the $\Gamma_2=9.42$ case.

CONCLUSIONS

A numerical method based on an Euler-Lagrange coupled two-phase flow model has been developed to study the effect of vortex/vortex interaction on bubble dynamics and cavitation noise. The liquid phase flow was solved by direct numerical simulation of the Navier-Stokes equations and was coupled with the SAP spherical bubble dynamics model to solve to the bubble phase flow at each time step. From the study of two-unequal co-rotating vortices, it was found that the minimum pressure could appear at different locations depending on the relative strength of the two vortices. For both cases studied, the minimum pressure appeared before the two vortices completely merged. The pressure reached its minimum when the vorticity of the weaker vortex was spread and sucked into the stronger vortex. This also resulted in an acceleration of the flow and led to a maximum streamwise velocity in the vortex center. A stronger interaction between the two vortices was also observed when the strengths of two vortices were closer.

The study of the “window of opportunity” showed that the shape, size and location of the window were highly dependent on the relative strength of the two vortices besides the nuclei sizes. A large size of “window of opportunity” was found for the stronger interaction case.

The effect of unsteady flow on the window was also shown to be important and could lead to a probability issue on the cavitation event. This unsteady effect was also found to cause a probability issue in cavitation event when comparing the acoustic signals between the steady and the unsteady computations.

ACKNOWLEDGMENTS

This work was conducted at Dynaflow, INC. (www.dynaflow-inc.com) and was partially supported by the the Office of Naval Research. The support of Dr. Ki-Han Kim, ONR, is greatly appreciated.
Figure 15. instantaneous bubble locations related to the two vortices which is illustrated by iso-$C_p$ surfaces at two values ($C_p=-2.8$ and $-5.0$) at four different time steps for $\Gamma_2=9.42$.

Figure 16. Bubble trajectories for nuclei captured by vortex and cavitate for $\Gamma_2=9.42$.

REFERENCES


