

Modelling Cavitating Flows using an Eulerian-Lagrangian Approach and a Nucleation Model

This content has been downloaded from IOPscience. Please scroll down to see the full text.

2015 J. Phys.: Conf. Ser. 656 012160

(<http://iopscience.iop.org/1742-6596/656/1/012160>)

View [the table of contents for this issue](#), or go to the [journal homepage](#) for more

Download details:

IP Address: 96.83.66.233

This content was downloaded on 22/12/2015 at 18:20

Please note that [terms and conditions apply](#).

Modelling Cavitating Flows using an Eulerian-Lagrangian Approach and a Nucleation Model

Jingsen Ma¹, Chao-Tsung Hsiao, and Georges L. Chahine
DYNAFLOW, INC., 10621-J Iron Bridge Road, Jessup, MD 20794, USA
E-mail: jingsen@dynaflow-inc.com

Abstract. An Eulerian/Lagrangian multi-scale two-phase flow model is developed to simulate the various types of cavitation including bubble, sheet, and tip vortex cavitation. Sheet cavitation inception, unsteady breakup, and cloud shedding on a hydrofoil are used as an example here. No assumptions are needed on mass transfer between phases; instead, the method tracks bubble nuclei, which are in the bulk of the liquid and those generated by nucleation from solid boundaries and this is sufficient to accurately capture the sheet 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. Nuclei are treated as flow singularities until they grow into large bubbles, which eventually merge to form a large scale discretised sheet cavity. The sheet performs large scale oscillations with a periodic reentrant jet forming under the sheet cavity, traveling upstream, and breaking the cavity. This results in bubble cloud formation and in high pressure peaks as the broken pockets shrink and collapse while travelling downstream. The results for a NACA0015 foil are in good agreement with the experimental data.

1. Introduction

Sheet cavitation on a lifting surface starts near the leading edge, effectuates large amplitude oscillations, and breaks up periodically releasing clouds of bubbles, which collapse violently generating high local pressures on the blade. Modelling of this process is challenging as it involves a wide range of characteristic lengths and phenomena including nucleation, large deforming free surfaces, and breakup into very fine bubble cloud, etc. Observations have indicated weak dependence of the sheet cavity dynamics on the nuclei distribution in the test facility, which has led some modellers to believe that sheet cavitation is not connected to bubble dynamics. Actually, over the years, several researchers (e.g. [1]) have observed that the presence of free field nuclei affected significantly the repeatability of the experimental results. When nuclei count was 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 these observations is that nucleation from the solid surface of the blades could play a more important role than the free field nuclei, and this has overlooked in previous studies of sheet cavitation even though bubble nucleation has been extensively investigated [2,3]. Motivated by the need for a physics-based model of sheet initiation, the present model aims at simulating cavitation starting from the dynamics of nuclei without using empirical

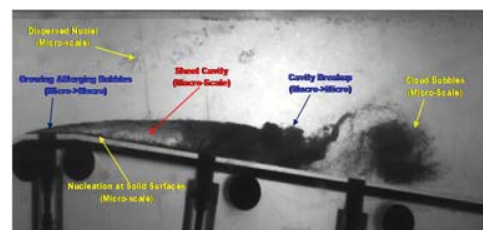


Figure 1. Illustration of various scales involved in sheet inception and dynamics.

¹ To whom any correspondence should be addressed.

fits for mass exchange as in conventional approaches. The model illustrated in Figure 1 addresses a macro-scale associated with the foil length and the sheet cavity, a micro-scale for the bubble nuclei, an intermediary scale for the bubble clouds, and transition and coupling procedures between these scales.

2. Multiscale Two-phase Model

2.1. Eulerian Continuum Model for Viscous Mixture Flow

The two-phase continuum medium satisfies the following continuity and momentum equations:

$$\frac{\partial \rho_m}{\partial t} + \nabla \cdot (\rho_m \mathbf{u}) = 0, \quad \rho_m \frac{D\mathbf{u}}{Dt} = -\nabla p + \mu_m \nabla^2 \mathbf{u} + \rho_m \mathbf{g}, \quad (1)$$

where \mathbf{u} is the mixture velocity, p the pressure and \mathbf{g} the acceleration due to gravity. The mixture density ρ_m and the mixture viscosity, μ_m , are related to the void fraction, α , and the liquid and gas properties through $\rho_m = (1 - \alpha)\rho_l + \alpha\rho_g$, $\mu_m = (1 - \alpha)\mu_l + \alpha\mu_g$. α is deduced from the location and size information of the bubbles using a Gaussian distribution scheme which smoothly “spreads” each bubble’s volume over its surroundings while conserving the total bubble volumes [4].

2.2. Lagrangian Discrete Bubble Model for Microbubbles

The bubble model uses a Surface Average Pressure approach [5] to average fluid quantities along the bubble surface and model its dynamics using a Rayleigh-Plesset-Keller-Herring equation:

$$\left(1 - \frac{\dot{R}}{c_m}\right) R \ddot{R} + \frac{3}{2} \left(1 - \frac{\dot{R}}{3c_m}\right) \dot{R} = \frac{\mathbf{u}_s^2}{4} + \frac{1}{\rho_m} \left(1 + \frac{\dot{R}}{c_m} + \frac{R}{c_m} \frac{d}{dt}\right) \left[p_v + p_g - p_{enc} - \frac{2\gamma}{R} - 4\mu_m \frac{\dot{R}}{R} \right], \quad (2)$$

where R is the bubble radius, p_v is the liquid vapor pressure, p_g is the bubble gas pressure, γ is the surface tension, $\mathbf{u}_s = \mathbf{u}_{enc} - \mathbf{u}_b$, \mathbf{u}_b is the bubble travel velocity, while \mathbf{u}_{enc} and p_{enc} are respectively the liquid velocity and the ambient pressure “seen” by the bubble during its travel. c_m is the local sound velocity in the two phase medium. The bubble trajectory is obtained from the following bubble motion equation:

$$\frac{d\mathbf{u}_b}{dt} = \left(\frac{\rho_l}{\rho_b} \right) \left[\frac{3}{8R} C_D |\mathbf{u}_s| \mathbf{u}_s + \frac{1}{2} \left(\frac{d\mathbf{u}_{enc}}{dt} - \frac{d\mathbf{u}_b}{dt} \right) + \frac{3\dot{R}}{2R} \mathbf{u}_s - \frac{\nabla p}{\rho_l} + \frac{(\rho_b - \rho_l)}{\rho_l} \mathbf{g} + \frac{3C_L \sqrt{v}}{4\pi R} \frac{\mathbf{u}_s \times \boldsymbol{\Omega}}{\sqrt{|\boldsymbol{\Omega}|}} \right], \quad (3)$$

where ρ_b is the bubble density, C_D , C_L are the drag and lift coefficients and $\boldsymbol{\Omega}$ is the deformation tensor.

2.3. Level-Set Method for Macro Cavities

In order to simulate large free surface deformations including folding and breakup as in a sheet cavity, a Level-Set method is used. A smooth function $\varphi(x, y, z, t)$, whose zero level coincides with 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 $\varphi(x, y, z, 0) = d(x, y, z)$. This function is enforced to be a material surface evolving with velocity field \mathbf{u} at each time step:

$$\frac{d\varphi}{dt} = \frac{\partial \varphi}{\partial t} + \mathbf{u} \nabla \varphi = 0. \quad (4)$$

2.4. Transition Model between Micro- and Macro-Scale

The transformation between micro- to macro-scale and vice-versa includes two scenarios. In the first scenario, since the bubbles are tracked using the Lagrangian scheme, the bubble sizes and locations are known at every time step. A criterion based on bubble size is set to “activate” the bubbles for computation of a local distance function for neighbouring fluid cells. For each cell i , the distance function is defined by $\varphi_i = \min(\varphi_{LS0}, \varphi_{b,j})$; $j = 1, N_i$, where

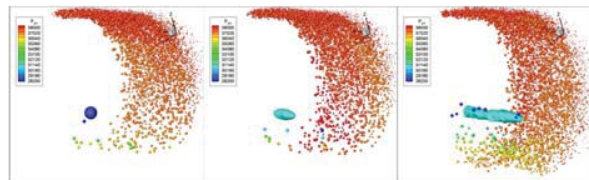


Figure 2. Time sequence of bubble capture in a line vortex cavitation core. The bubbles are colored by their pressure. The aqua blue iso-surface is the gas/vapor-liquid interface resulting from discrete large free surface formation.

φ_{LS0} is the original distance cell i from level set 0, $\varphi_{b,j}$ is the local distance between the center of cell i and the surface of bubble j , and N_i is the number of relevant bubbles around cell i . This scheme allows bubbles to merge together into a large cavity, as well as single bubbles to be absorbed by a large cavity as illustrated in Figure 2 for a sequence of bubble capture in a cavitating line vortex. Once a bubble grows explosively, its interface becomes part of the level set 0. Bubbles can coalesce and join to form a sheet or a tubular vortex cavity. The second scenario is applied when the volume of a large cavity is decreasing in time. As the large cavity collapses, the lost volume is replaced with a distribution of micro-bubbles of the same volume if the mixture normal speed is larger than the interface normal speed [6,7].

2.5. Nucleation Model

A very important aspect of our approach is to account for nucleation from the rigid boundaries [4] in addition to bubble nuclei dispersed in the liquid. The wall nucleation model involves the following parameters, which are functions of the wall material properties and the local flow conditions: a) P_{thr} : a nucleation pressure threshold, b) N_s : a number density of nucleation sites per unit area, c) f_n : a nucleation rate, and d) R_0 : the initial nuclei size(s). At each time step the pressure at a cell is compared with P_{thr} to determine if nucleation should occur. Once the pressure drops below the threshold, the cell releases a total of N nuclei, $N = N_s f_n \Delta t \Delta A$, where ΔA is the surface area of the grid cell and Δt the integration time step. All above parameters are strong functions of surface roughness and temperature. In the results below, P_{thr} is selected to be the vapour pressure. The initial size of the nuclei emitted is selected to be $10 \mu\text{m}$ 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 $f_n = 22\text{kHz}$. These parameters are selected based on the study in [8], where the results became independent of N_s and f_n when larger than these values.

3. Simulation of Sheet Cavitation on Naca0015 Hydrofoil

As illustrated in Figure 3, the present model successfully captures sheet cavity initiation, dynamics, and breakup on a well-tested foil. Bubbles nucleated from the blade surface and free nuclei in the liquid were the basis of the simulations and no empirical mass exchange models were needed. Cavitation initiates on the hydrofoil leading edge when the local pressure drops below the vapour pressure. The nuclei grow, aggregate, and coalesce into a large cavity well-captured with the Level Set method. As the sheet cavity develops, its trailing edge curves into a reentrant jet, which moves upstream over the foil and into the cavity, then breaks it into a smaller cavity and a vortical large bubble cloud, which collapses generating high pressures. This periodic shedding phenomenon can be further quantified by analysing the time histories of the instantaneous pressures and stream-wise velocities in the flow. For example Figure 5 shows the pressures and velocities monitored at the location $x=0.4 L$ downstream of the leading edge on the suction side of the foil. This is close to the maximal extent of the trailing edge of the fully developed sheet cavity. This location alternatively experiences negative (reentrant jet development) and positive (no reentrant jet) stream-wise velocities with magnitudes as high as the incoming flow U_∞ . The time interval between consecutive negative velocity maxima is repeatable and has a value close to $1.3 L/U_\infty$. The corresponding oscillation frequency is about $0.78 U_\infty/L$, which matches well with the experimentally measured shedding frequency for the same cavitation number [9]. Comparison of the numerical results with experimental observations for a large range of σ is very satisfactory, as seen in Figure 4. Both the shedding frequency and the time-averaged cavity length compare well with the experimental measurements.

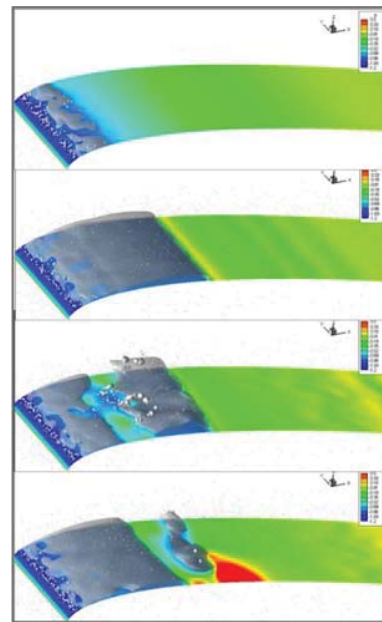


Figure 3. Time sequence of sheet-to-cloud cavitation modeling on

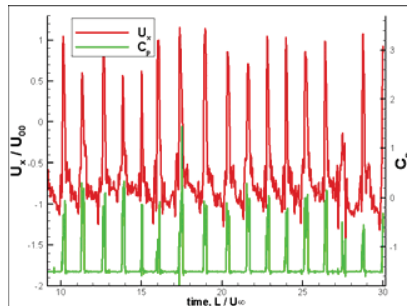


Figure 5. Pressures and streamwise velocity at $x = 0.4L$ on the solid wall near the cavity trailing edge. $\sigma = 1.5$

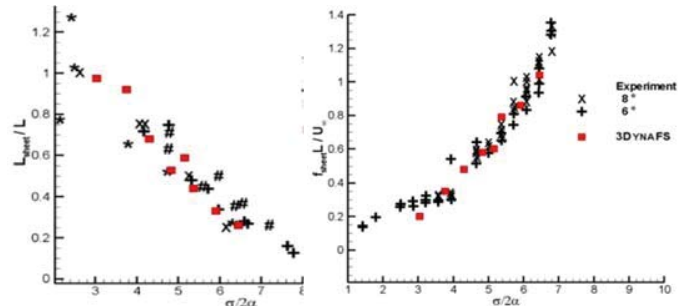


Figure 4. Comparison of predicted sheet cavity averaged length and shedding frequencies with data in [14].

4. Summary

A multi-scale framework was developed for the simulation of various forms of cavitation. The example of sheet cavitation on a hydrofoil shows how the model bridges smoothly 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 match experimental data very well for a large range of cavitation numbers.

5. 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.

6. References

- [1] Kuiper G 1985 Reflection on cavitation inception *ASME Cavitation and Multiphase Flow Forum* (Albuquerque, NM)
- [2] Yount D E 1979 Skin of varying permeability: A stabilization mechanism for gas cavitation nuclei *J. Acoust. Soc. Am.* **65** 1429–39
- [3] Mørch K A 2009 Cavitation Nuclei: Experiments and Theory. *Journal of Hydrodynamics J. Hydrodyn. Ser. B* **21** 176–89
- [4] Ma J, Chahine G L and Hsiao C-T 2015 Spherical bubble dynamics in a bubbly medium using an Euler–Lagrange model *Chemical Engineering Science* vol 128 pp 64–81
- [5] Hsiao C-T, Chahine G L and Liu H-L 2003 Scaling effect on prediction of cavitation inception in a line vortex flow *J. Fluids Eng.* **125** 53–60
- [6] Ma J, Oberai A A, Drew D A, Lahey R T and Hyman M C 2011 A comprehensive sub-grid air entrainment model for RANS modeling of free-surface bubbly flows *J. Comput. Multiph. Flows* **3** 41–56
- [7] Hsiao C-T, Wu X, Ma J and Chahine G L 2013 Numerical and Experimental Study of Bubble Entrainment Due to a Horizontal Plunging Jet *Int. Shipbuild. Prog.* **60** 435–69
- [8] Hsiao C T, Ma J and Chahine G L 2014 Multi-scale two-phase flow modeling of sheet and cloud cavitation *30th Symposium on Naval Hydrodynamics* (Hobart, Tasmania, Australia)
- [9] Berntsen G S, Kjeldsen M and Arndt R E 2001 Numerical modeling of sheet and tip vortex cavitation with FLUENT 5 *Fourth International Symposium on Cavitation*. (Pasadena, CA)