NUMERICAL STUDY OF CAVITATING FLOWS AROUND A HYDROFOIL

DEZHI DAI* and ALBERT Y. TONG

Department of Mechanical and Aerospace Engineering, University of Texas at Arlington, Arlington, TX 76019, USA, dezhi.dai@mavs.uta.edu, tong@uta.edu

Keywords: *cavitation, hydrofoil, polygonal mesh, VOF, PLIC*

1 Introduction

Cavitation appears when local static pressure drops below the vapor pressure of water and usually causes significant impacts on the performance of many hydraulic devices, especially marine propeller blades. For efficiency reasons, the propeller usually operates in cavitating conditions whereas the cavitation may cause blade surface erosion, noise, vibration and performance breakdown [1]. Accurate prediction of the cavitating flows around a hydrofoil is essential in the design of modern marine propellers.

The Transport Equation based Model (TEM) has been extensively employed in the numerical study of cavitating flows around a hydrofoil [1]. In the TEM model, the interface between water and its vapor is captured by the Volume of Fluid (VOF) method and a source term regarding the mass transfer is added to the standard VOF equation. Four important factors should be considered for the TEM model: an appropriate mass transfer rate evaluation method, turbulent effect of cavitating flows, computational domain discretization and a numerical algorithm to solve the VOF equation.

The mass transfer rate between the liquid and gas phases is evaluated by cavitation models. A detailed developing history of the cavitation models can be found in [2]. Schnerr and Sauer [3] presented the first model without any empirical constants and it is employed in the present study.

Most of the applications of cavitation are based on Reynolds-Averaged NavierStokes (RANS) equations [3–7]. In the present study, the Spalart-Allmaras (SA) one-equation model is employed for the sake of computational efficiency and several studies [5, 7] have already confirmed that the SA turbulence model can ensure the accuracy for the cavitating flow simulations.

The applications with structured meshes are restrained to a very simple domain. Polygonal unstructured meshes with superior flexibility for complex geometries are employed in the present study for the sake of computation efficiency and gradient evaluation accuracy [8].

The discontinuity property of the VOF function near the interface makes it unable to be solved like other flow variables by using standard advection schemes. In OpenFOAM, the Multidimensional Universal Limiter with Explicit Solution (MULES) scheme [9] is employed to capture the interface. However, MULES scheme suffers from numerical diffusion at the interface cells [10]. The Piecewise Linear Interface Calculation (PLIC) method [11] can keep the interface sharp while maintaining mass conservation at the expense of an extra reconstruction step and few papers [12] have employed this method.

The present study focuses on verification of the PLIC-VOF method on a polygonal unstructured mesh with RANS flow solver, SA turbulent and SchnerrSauer cavitation models in cavitating flow simulations and the influence of cavitation on the dynamics of the two-dimensional hydrofoil used in [13].

2 Methodology

The RANS equations with phase-change are given by:

$$\nabla \cdot \vec{U} = \left(\frac{1}{\rho_1} - \frac{1}{\rho_2}\right)\dot{m}, \tag{1a}$$

$$\frac{\partial}{\partial t}\left(\rho\vec{U}\right) + \nabla \cdot \left(\rho\vec{U}\vec{U}\right) = -\nabla p + \nabla \cdot \left(\left(\mu + \mu_t\right)\left(\nabla\vec{U} + \left(\nabla\vec{U}\right)^T - \frac{2}{3}\left(\nabla \cdot \vec{U}\right)\bar{\bar{I}}\right)\right), \tag{1b}$$

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot \left(\alpha\vec{U}\right) = \frac{\dot{m}}{\rho_1}, \tag{1c}$$

where ρ_1 and ρ_2 are the density of the liquid and vapor phases, respectively, $\dot{m}$ the mass transfer rate due to cavitation, μ_t the turbulent eddy viscosity, $\bar{\bar{I}}$ the unit tensor and α the VOF function. Both the liquid and vapor phases are considered incompressible and share the same mixture velocity field $\vec{U}$. Also, the turbulent eddy viscosity μ_t and the mass transfer rate $\dot{m}$ are solved by the SA turbulence and SchnerrSauer cavitation models built in OpenFOAM, respectively.

The RANS equations are solved by a modified cavitating flow solver based on `interPhaseChangeFoam` (details can be found in [14]) which is a standard for two incompressible, isothermal immiscible fluids with phase-change. The MULES-VOF scheme is replaced by a PLIC-VOF scheme developed in the present study and the source code will be released once the full paper is published.

On an unstructured mesh, Eq.(1c) is discretized as

$$\left(\alpha^{n+1}-\alpha^{n}\right)+\frac{1}{\varOmega}\sum_{f=1}^{NF}\left(\phi_f^n\int_t^{t+\Delta t}\alpha_f dt\right)=\frac{\dot{m}}{\rho_1}\Delta t, \tag{2}$$

where t is the time, Δt the time step, $\varOmega$ the cell volume, NF the number of cell faces, ϕ_f the volumetric flux through cell face f and superscripts $n+1$ and n represent $t+\Delta t$ and t, respectively. The liquid fraction flux $L_f=\left(\phi_f^n\int_t^{t+\Delta t}\alpha_f dt\right)$ is calculated by using the PLIC-VOF method. As shown in Figure 1, the reconstructed interface is given by:

$$\vec{n}\cdot\vec{X}+D_0=0, \tag{3}$$

where $\vec{n}\left(=-\frac{\nabla\alpha}{\|\nabla\alpha\|}\right)$ is the unit outward normal vector of the interface, $\vec{X}$ the position vector of the interface and D_0 the signed distance from the origin. D_0 is calculated by an analytical algorithm developed by the authors recently. The interface moved from D_0^n to a new position D_0^{n+1} in the time interval $[t,t+\Delta t]$ with interface normal velocity U_0 and $D_0^{n+1}=D_0^n-U_0\Delta t$. In the PLIC-VOF method, the liquid fraction flux L_f is evaluated by using the trapezoidal rule, i.e.

$$L_f=\phi_f^n\int_t^{t+\Delta t}\alpha_f dt=\frac{\phi_f^n}{A_f}\int_t^{t+\Delta t}A_{l,f}dt=\frac{\phi_f^n\Delta t\left(A_{l,f}^n+A_{l,f}^{n+1}\right)}{2A_f}, \tag{4}$$

where A_f and $A_{l,f}$ are the area of face f and the area below the interface, respectively.

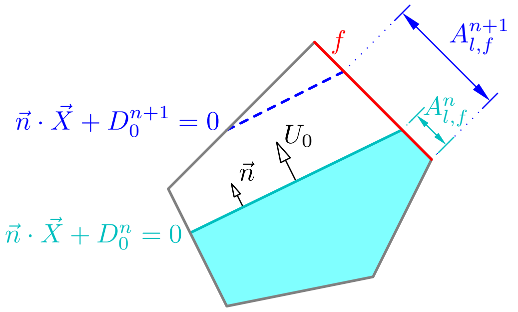
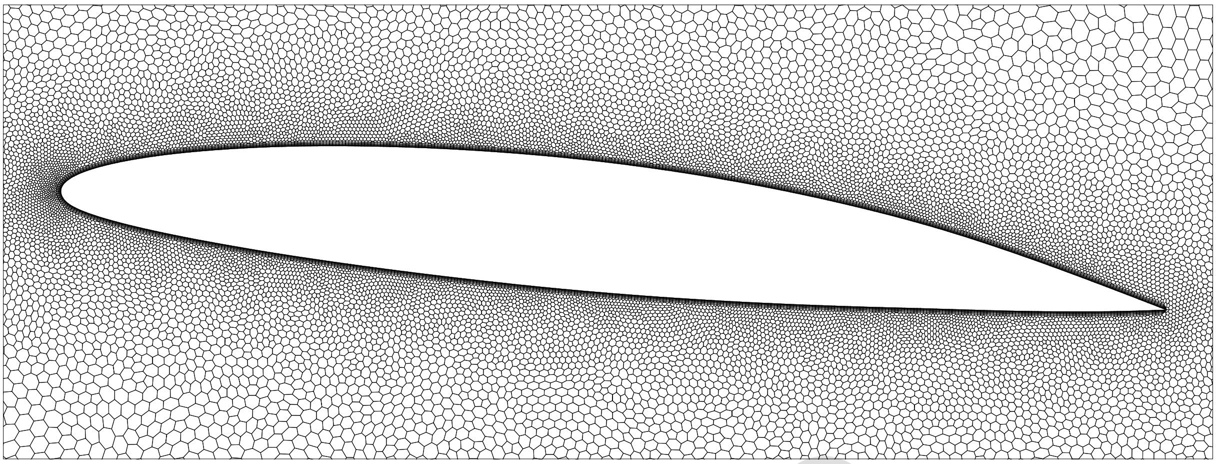


Figure 1: Illustration of the interface line in a mixed cell.

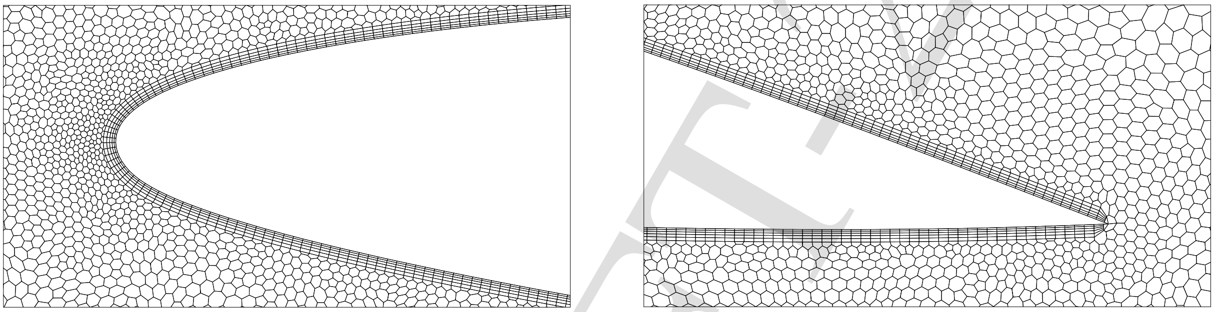
3 Preliminary Results

The numerical simulations are performed on a polygonal unstructured grid with a body-fitted boundary layer mesh as shown in Figure 2. The numerical models, that is the combination of the PLIC-VOF method, SchnerrSauer cavitation model, RANS solver and SA turbulent model, are verified by comparing the numerical results in cavitating conditions with the experimental data [13] and other numerical results available in the literature [15]. All of the simulations are performed at $AOA=6°$ and $Re=7.5\times10^5$ with different cavitation numbers. The time-averaged c_p distribution on the suction side of the hydrofoil and water volume fraction contours at $\sigma=1.622$, 1.541 and 1.495 are shown in Figures 3 - 5. The agreement between the present numerical results and measured c_p values is very good. Compared with the numerical results in [15], the c_p distributions in the present study are closer to the experimental data, especially near the cavity closure region. This suggests that the numerical models employed in the present study could adequately simulate the fluid dynamics of cavitating flows around a hydrofoil.

The influence of various parameters on the dynamics of the hydrofoil is currently being studied. The results will be reported in the conference.



(a) Mesh around the hydrofoil



(b) Close-up view of mesh near the leading edge

(c) Close-up view of mesh near the trailing edge

Figure 2: Employed polygonal mesh with 22359 cells at $AOA = 6°$.

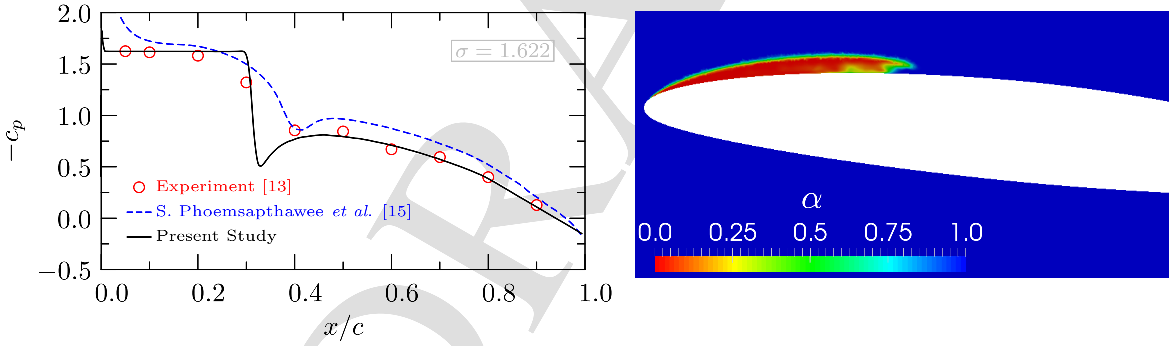


Figure 3: Time-averaged c_p distribution and water volume fraction contours at $\sigma = 1.622$.

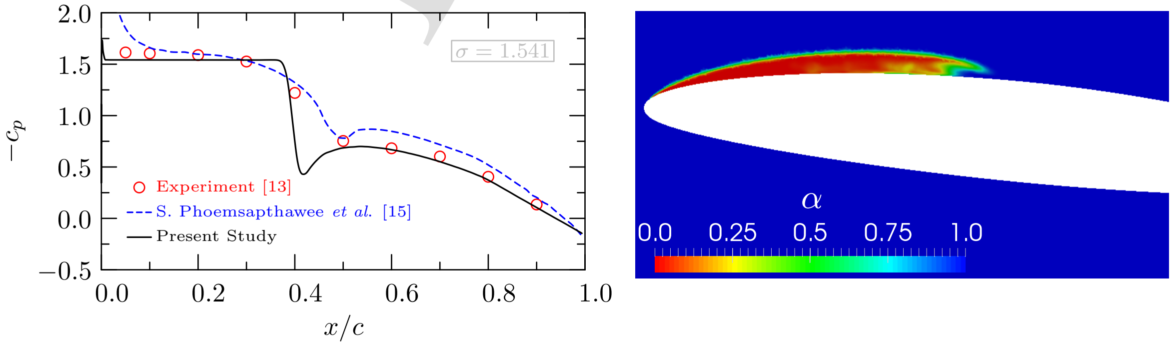


Figure 4: Time-averaged c_p distribution and water volume fraction contours at $\sigma = 1.544$.

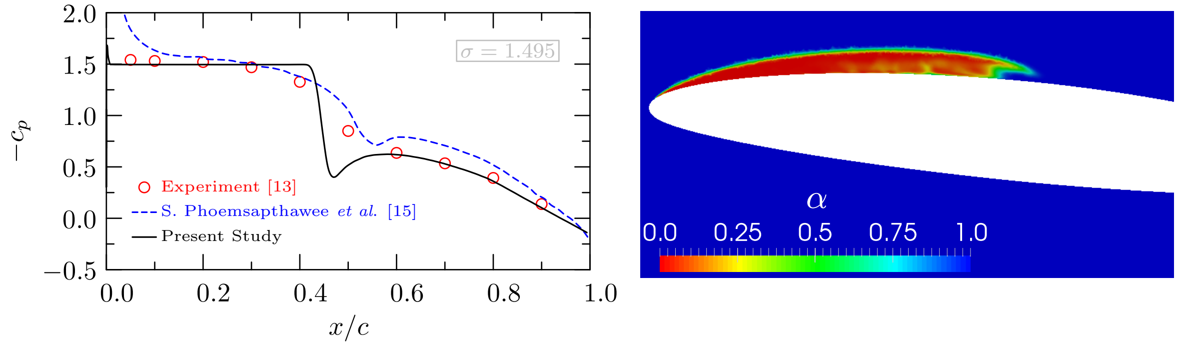


Figure 5: Time-averaged c_p distribution and water volume fraction contours at $\sigma = 1.495$.

Acknowledgments

The authors are thankful to all those involved in the organization of OFW13, all the contributors that will enrich this event and The Texas Advanced Computing Center (TACC) for providing the necessary computational resources.

References

[1] E. Roohi, A. P. Zahiri, and M. Passandideh-Fard, “Numerical simulation of cavitation around a two-dimensional hydrofoil using vof method and les turbulence model,” *Applied Mathematical Modelling*, vol. 37, no. 9, pp. 6469–6488, 2013.

[2] A. Niedźwiedzka, G. H. Schnerr, and W. Sobieski, “Review of numerical models of cavitating flows with the use of the homogeneous approach,” *Archives of Thermodynamics*, vol. 37, no. 2, pp. 71–88, 2016.

[3] G. H. Schnerr and J. Sauer, “Physical and numerical modeling of unsteady cavitation dynamics,” in *Fourth international conference on multiphase flow, New Orleans, USA*, vol. 1, 2001.

[4] S. H. Rhee, T. Kawamura, and H. Li, “Propeller cavitation study using an unstructured grid based navier-stoker solver,” *Journal of Fluids Engineering*, vol. 127, no. 5, pp. 986–994, 2005.

[5] J. Seo and S. Lele, “Numerical investigation of cloud cavitation and cavitation noise on a hydrofoil section,” 2009.

[6] M. Morgut, E. Nobile, and I. Biluš, “Comparison of mass transfer models for the numerical prediction of sheet cavitation around a hydrofoil,” *International Journal of Multiphase Flow*, vol. 37, no. 6, pp. 620–626, 2011.

[7] S. J. Ahn and O. J. Kwon, “Numerical investigation of cavitating flows for marine propulsors using an unstructured mesh technique,” *International Journal of Heat and Fluid Flow*, vol. 43, pp. 259–267, 2013.

[8] M. Peric and S. Ferguson, “The advantage of polyhedral meshes,” *Dynamics*, vol. 24, p. 45, 2012.

[9] S. S. Deshpande, L. Anumolu, and M. F. Trujillo, “Evaluating the performance of the two-phase flow solver interfoam,” *Computational science & discovery*, vol. 5, no. 1, p. 014016, 2012.

[10] J. Roenby, H. Bredmose, and H. Jasak, “A computational method for sharp interface advection,” *Open Science*, vol. 3, no. 11, p. 160405, 2016.

[11] D. L. Youngs, “Time-dependent multi-material flow with large fluid distortion,” *Numerical methods for fluid dynamics*, vol. 24, no. 2, pp. 273–285, 1982.

[12] M. Passandideh-Fard and E. Roohi, “Transient simulations of cavitating flows using a modified volume-of-fluid (vof) technique,” *International Journal of Computational Fluid Dynamics*, vol. 22, no. 1-2, pp. 97–114, 2008.

[13] J.-B. Leroux, J. A. Astolfi, and J. Y. Billard, “An experimental study of unsteady partial cavitation,” *Journal of fluids engineering*, vol. 126, no. 1, pp. 94–101, 2004.

[14] M. Andersen, E. Yasari, C. Järpner, and A. Zarmehri, “A interphasechangefoam tutorial,” *MSc/PhD course in CFD with OpenSource software*, 2011.

[15] S. Phoemsapthawee, J.-B. Leroux, J.-M. Laurens, ENSIETA, F. Deniset, and E. Navale, “A transpiration velocities based sheet cavitation model,” *Ship Technology Research*, vol. 56, no. 4, pp. 161–176, 2009.