A NUMERICAL STUDY OF CAVITATING FLOWS AROUND A HYDROFOIL USING OVERSET MESHES

DEZHI DAI[1] and ALBERT Y. TONG[2]

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

[2]Department of Mechanical and Aerospace Engineering, University of Texas at Arlington, Arlington, TX 76019, USA, tong@uta.edu

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

1 Introduction

Cavitation appears when local static pressure drops below the water vapor pressure and causes significant impacts on marine propeller blades. For efficiency reasons, marine propellers usually operate under cavitating conditions and may suffer blade surface erosion, noise, vibration and performance breakdown [1]. As hydrofoil is the radial section of propeller blades, an accurate prediction of cavitating flows around a hydrofoil is essential in the design of marine propellers.

The Transport Equation-based Model (TEM) has been extensively employed in the numerical study of cavitating flows [1]. In the TEM model, the interface between liquid and vapor is captured by the Volume of Fluid (VOF) method and a source term regarding the mass transfer is added to the standard homogeneous VOF equation. Four important elements are considered in the TEM model: computational domain discretization, numerical algorithm for the VOF equation, mass transfer rate evaluation for phase change and the turbulent modeling of cavitating flows.

The use of stationary meshes is restrained to unmoving parts in the computational domain. Also, a common difficulty in simulating complex fluid flows is that some geometries cannot be well-represented by using a single mesh [2]. Representing distinct geometries by different mesh parts is a better choice in general. It is also complicated and time-consuming to prepare a single stationary mesh with complex geometries. Dynamic overset mesh [3] can be especially useful in applications involving component motion and is supported in the latest version of OpenFOAM. However, overset mesh cannot be used in the standard cavitation solver, *interPhaseChangeFoam*, in OpenFOAM.

Furthermore, *interPhaseChangeFoam* employs an algebraic method, the Multidimensional Universal Limiter with Explicit Solution (MULES) scheme [4], to solve the VOF equation which suffers from numerical diffusion at interface cells [5]. Piecewise Linear Interface Calculation (PLIC) method [6] can keep the interface sharp while maintaining mass conservation at the expense of an extra reconstruction step and has been employed in the study of cavitation [7].

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 [8]. Schnerr and Sauer [9] presented the first model without any empirical constants and is employed in the present study.

Most of the applications on cavitation are based on the Reynolds-Averaged Navier-Stokes (RANS) equations [9–13]. In the present study, the Spalart-Allmaras (SA) one-equation model is employed for the sake of computational efficiency. Several studies [11, 13] have already confirmed that the SA turbulence model can ensure accuracy for the cavitating flow simulations.

The cavitation solver *overInterPlicPhaseChangeDyMFoam*, recently developed by the authors, implements the dynamic overset mesh technique in conjunction with an analytical PLIC interface reconstruction algorithm to perform interface flow simulations. The present study focuses on the applications of the solver on an overset mesh along with the RANS equation, SA turbulent and SchnerrSauer cavitation models in cavitating flow simulations over a hydrofoil.

2 Methodology

The VOF equation phase-change is given by:

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

where ρ1 and ρ2 are the density of the liquid and vapor phases, respectively, $\dot{m}$ the mass transfer rate due to phase change 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 mass transfer rate $\dot{m}$ is solved by the SchnerrSauer cavitation models built in the OpenFOAM.

On a general unstructured mesh, Eq.(1) is discretized as

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

where t is the time, Δt the time step, Ω 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 $t+\Delta t$, 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

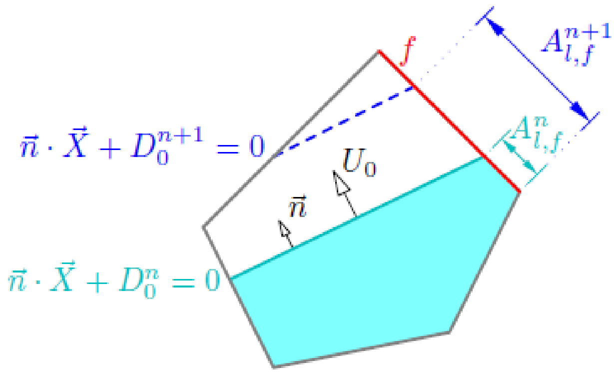


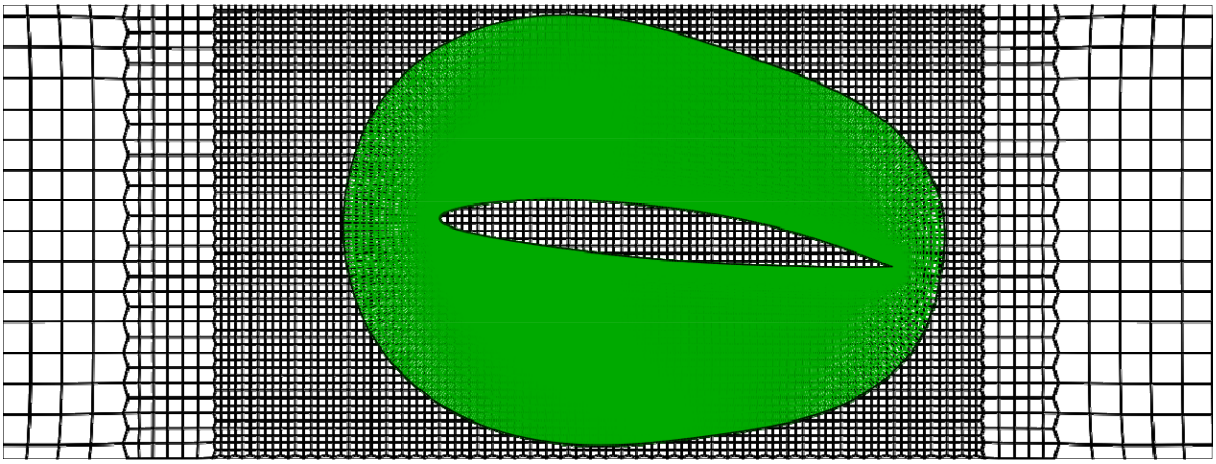
Figure 1: Illustration of the interface line in a mixed cell

The overset interpolation of the VOF function α as well as the other variables are implemented by using the inverse distance scheme.

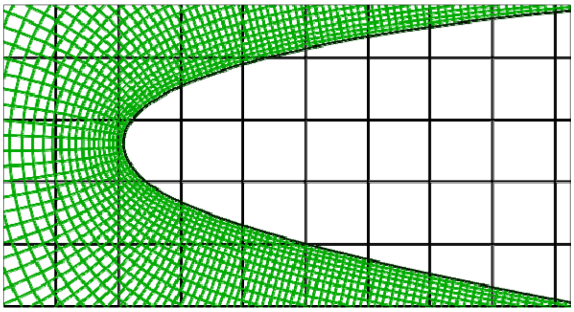
3 Preliminary Results

The numerical simulations are performed on an overset mesh as shown in Figure 2. The numerical models, based on the combination of the PLIC-VOF method in overset mesh, SchnerrSauer cavitation model, RANS solver and SA turbulent model, are verified by comparing the numerical results in cavitating conditions with the experimental data [14] and other numerical results available in the literature [15]. All of the simulations are performed at AOA = 6◦ and Re = 7.5 105 with different cavitation numbers. The time-averaged cp distribution on the suction side of the hydrofoil and water volume fraction contours at σ = 1.622, 1.541 and 1.495 are shown in Figures 3 - 5. The present numerical results for cp are in good agreement with the experimental values. Compared with the numerical results in [15], the cp distributions in the present study are closer to the experimental data, especially near the cavity closure region. This suggests that the overset mesh and numerical models employed in the present study could adequately simulate the fluid dynamics of cavitating flows around a hydrofoil.

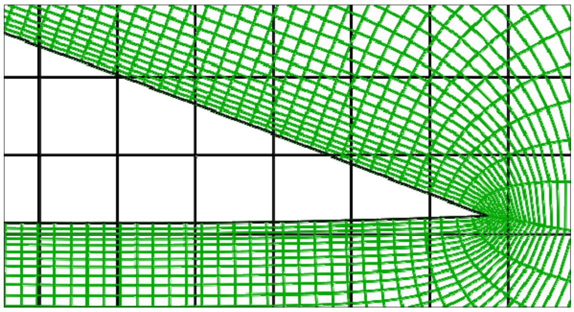
The influence of various parameters on the dynamics of the hydrofoil is currently being studied. The results will be reported in the conference.



(a) Overset 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 overset mesh with 30665 cells at $AOA = 6^{\circ}$.

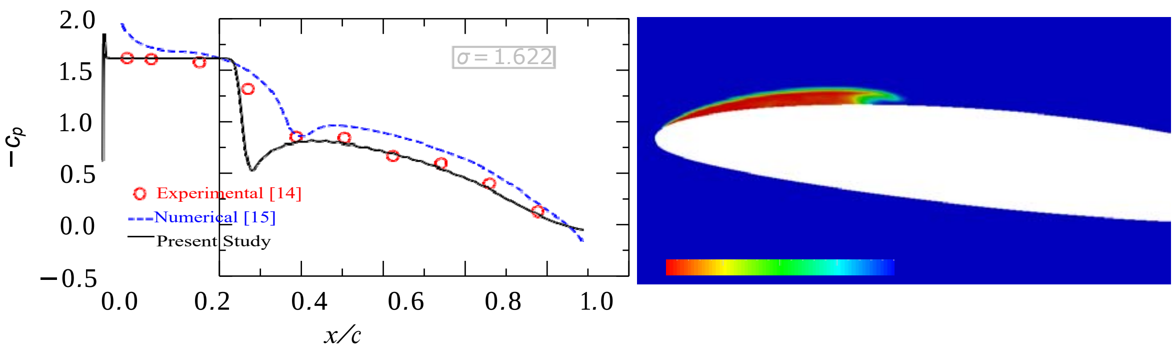


Figure 3: Time-averaged c_p distribution and water volume fraction contours at $\sigma = 1.622$.

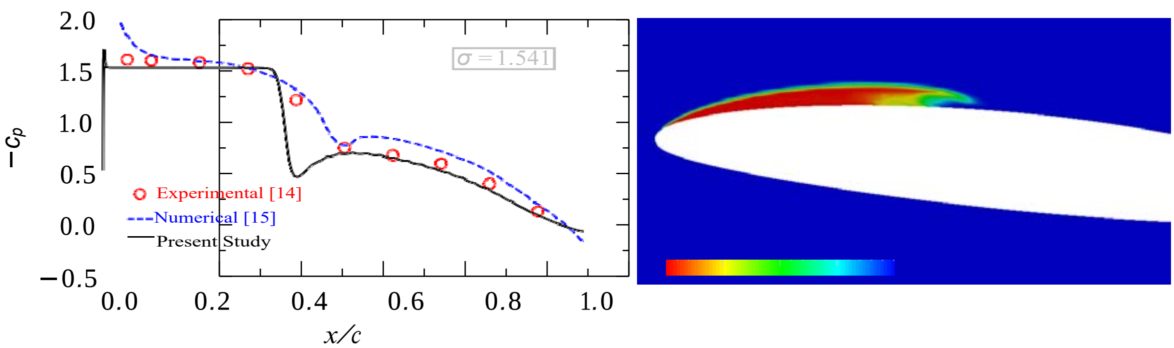


Figure 4: Time-averaged c_p distribution and water volume fraction contours at $\sigma = 1.544$.

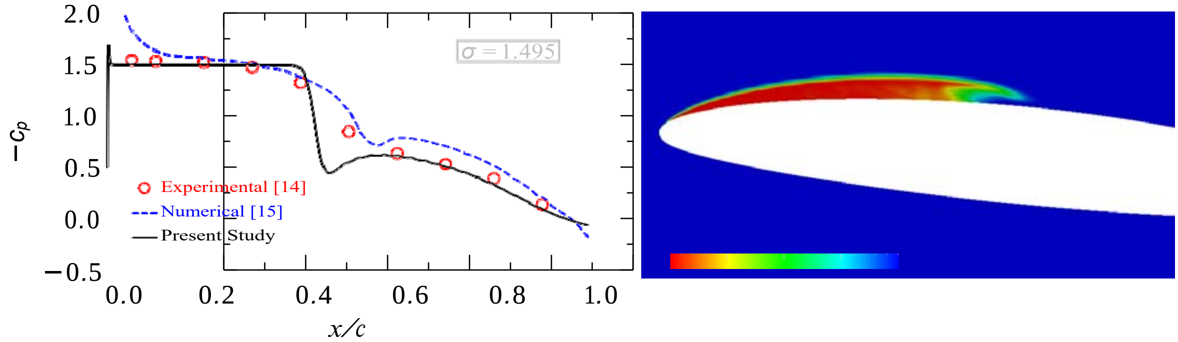


Figure 5: Time-averaged c_p distribution and water volume fraction contours at $\sigma = 1.495$.

Acknowledgments

The authors are thankful to The University of Texas at Arlington (UTA) for the financial support to attend the workshop 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. Fluent, "Fluent 12. theory guide," 2017.

[3] Z. Shen, D. Wan, and P. M. Carrica, "Rans simulations of free maneuvers with moving rudders and propellers using overset grids in openfoam," in *SIMMAN workshop on Verification and Validation of Ship Maneuvering Simulation Methods. Presented at the SIMMAN workshop on Verification and Validation of Ship Maneuvering Simulation Methods, Lyngby, Denmark*, 2014.

[4] 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.

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

[6] 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.

[7] 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.

[8] A. Nied źwiedzka, 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.

[9] 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.

[10] 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.

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

[12] M. Morgut, E. Nobile, and I. Bilu sˇ, "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.

[13] 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.

[14] 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.

[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.