esi OpenFOAM CONFERENCE in Germany | OCT 15-17, 2019

# Near-surface Cavitation Modelling in OpenFOAM

N. Lex Mulcahy, Peter D. M. Brady, Damian G. McGuckin

*Pacific Engineering Systems International, 277-279 Broadway, Glebe NSW 2037, Australia*
*ph: +61 2 8571 0847, lexm@esi.com.au, peterb@esi.com.au, damianm@esi.com.au*

Cavitation is a flow phenomenon that sees the formation of vapour bubbles in the liquid in regions in which the flow pressure is less than the vapour pressure. There have been many studies on cavitation but only a small proportion of these have considered cavitation near a free liquid surface. Hydrofoils used as lifting devices for high-speed vessels [1] are near the free surface by necessity, and cavitation may be unavoidable in such situations. As well, the presence of the hydrofoil affects the shape of the free surface. Propellers of surface vessels may be near the water surface.

The present work uses OpenFOAM v1706 [2]. Specifically, a solver for a three-phase mixture with cavitation was developed based on the interPhaseChangeFoam solver along with some parts of the multiphaseInterFoam solver. The approach solves the Reynolds-Averaged Navier-Stokes (RANS) flow equations. It involves a multiphase analysis using a homogeneous mixture model with the velocities of the different phases (liquid water, water vapour and air) assumed to be equal at any given point. Mass transfer due to cavitation can only occur between the liquid water and water vapour phases. Transport equations are solved for the volume fractions of the air and water vapour phases with the volume fraction of liquid water being the remainder. This contrasts to the (two-phase) interPhaseChangeFoam which solves a transport equation for the liquid fraction with the vapour fraction as the remainder. In the three-phase model, solving for liquid fraction often resulted in spurious vapour fractions at the liquid-air interface that affected subsequent calculations.

Various cavitation models may be used to predict phase changes. Modified versions of the Schnerr-Sauer model [3,6], Singhal's Full Cavitation Model [4,6] and the Zwart-Gerber-Belamri model (hereafter referred to as the Zwart model) [5,6] are incorporated in the code. Typically, the two-phase (liquid, vapour) mass transfer expressions were used but multiplied by a filter to enforce the condition that mass transfer does not take place between either air and the liquid or air and vapour. The filter was $(1-\alpha_a)$ where $\alpha_a$ is the air volume fraction.

A simulation of a test by Duncan [7] was performed as a check on the behaviour of the model with a free surface but without cavitation. Duncan's tests were for a two-dimensional submerged hydrofoil moving horizontally at constant speed in a shallow tank 24 m long, 0.61 m deep and 0.61 m wide. The foil had a NACA 0012 section, 0.203 m chord length (*c*) and was located at 0.175 m above the tank floor. The simulation is for a test with foil speed of 0.8 m/s, 5° angle of attack and depth of submergence of 0.193 m at the foil mid-chord. The water fraction at the end of the simulation is shown in Figure 1, and the generated water surface profile is compared to the experimental results of Duncan and the simulation results of Yang and Stern [8] and Wu and Chen [9] in Figure 2. Singhal's cavitation model was active but no cavitation occurred. Results for water surface profile are reasonable.

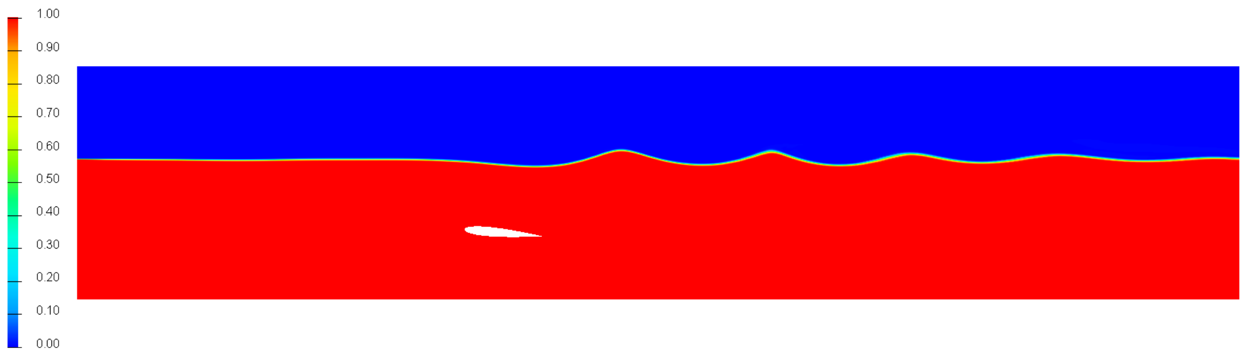


*Figure 1: NACA 0012 foil with free surface – water volume fraction*

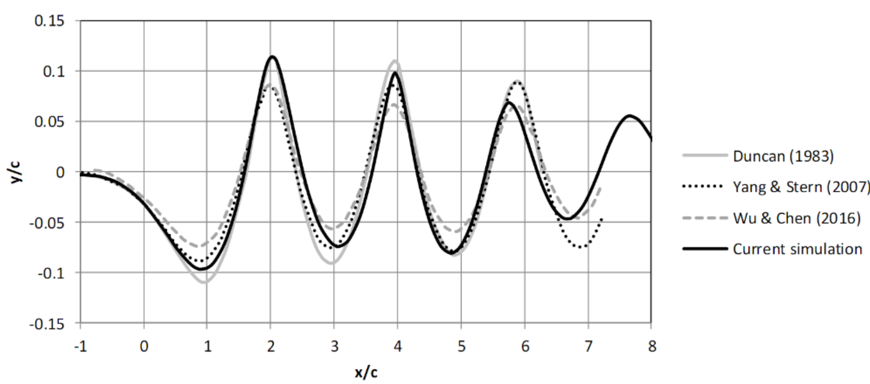


*Figure 2: NACA 0012 foil with free surface – water surface profile*

Xu et al. [10] reported a water tank experiment and simulations for cavitating flow around a blunt cylindrical body near the free water surface. In the experiment a slender polished stainless-steel cylindrical projectile of 37 mm diameter was launched horizontally into a water tank, attaining a speed of 18.5 m/s in 50 $\mu$s. For the present example, the top of the cylinder was 15 mm below the undisturbed water surface. The finite length of the projectile was neglected and instead it extended to the outflow boundary. Some results for a simulation with Zwart's cavitation model are presented in Figure 3. The cavitation bubble lengths are on the centreline of the cylinder (above and below). The comparison to the test results is reasonable until about 0.014s, after which there is a lot of bubble break-up.

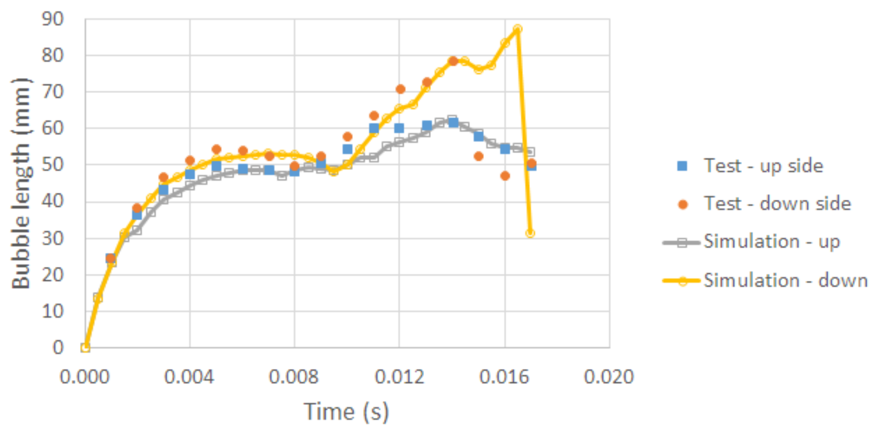


*Figure 3: Blunt cylinder with free surface – bubble lengths*

**References**


1. Acosta AJ. Hydrofoils and hydrofoil craft. Ann. Rev. Fluid Mech. 1973;5:161–184.
2. OpenFOAM The Open Source CFD Toolbox User Guide, Version v1706, 28th June 2017.
3. Schnerr GH, Sauer J. Physical and numerical modeling of unsteady cavitation dynamics. In: Proc. 4th Int. Conf. Multiphase Flow (ICMF'01), New Orleans 2001.
4. Singhal AK, Athavale MM, Li H, Jiang Y. Mathematical basis and validation of the full cavitation model, Journal of Fluids Engineering, September 2002, Vol. 124, 617-624.
5. Zwart PJ, Gerber G, Belamri T. A two-phase flow model for prediction cavitation dynamics. In: Proc. 5th Int. Conf. Multiphase Flow (ICMF 2004), Yokohama 2004.
6. Niedźwiedzka A, Schnerr GH, Sobieski W. Review of numerical models of cavitating flows with the use of the homogeneous approach, Archives of Thermodynamics, Vol. 37(2016), No. 2, 71–88.
7. Duncan, JH. The breaking and non-breaking wave resistance of a two-dimensional hydrofoil. J. Fluid Mech. 126 (1983) 507–520.
8. Yang J, Stern F. Large-eddy simulation of breaking waves using embedded-boundary/level-set method, in: Proceedings of the 45th AIAA Aerospace Sciences Meeting and Exhibit, Reno, NV, 2007.
9. Wu PC, Chen JH. Numerical study on cavitating flow due to a hydrofoil near a free surface, Journal of Ocean Engineering and Science 1 (2016) 238–245.
10. Xu C, Wang Y, Huang C, Huang J, Yu C. The effect of free surface on cloud cavitating flow around a blunt body, Journal of Hydrodynamics, 2017, 29(6):979-986.