Compressible Turbulent Flow Numerical Simulations of Tip Vortex Cavitation

F. Khatami, E. van der Weide and H. Hoeijmakers

1 Mechanical Engineering Department, University of Twente
P.O. box 217, 7500 AE Enschede, The Netherlands
web page: http://www.utwente.nl/ctw/ts/
E-mail: f.khatami@utwente.nl, h.w.m.hoeijmakers@utwente.nl

Abstract.
For an elliptic Arndt’s hydrofoil numerical simulations of vortex cavitation are presented. An equilibrium cavitation model is employed. This single-fluid model assumes local thermodynamic and mechanical equilibrium in the mixture region of the flow, is employed. Furthermore, for characterizing the thermodynamic state of the system, precomputed multiphase thermodynamic tables containing data for the appropriate equations of state for each of the phases are used and a fast, accurate, and efficient look-up approach is employed for interpolating the data. The numerical simulations are carried out using the Unsteady Reynolds-Averaged Navier-Stokes (URANS) equations for compressible flow. The URANS equations of motion are discretized using a finite volume method for unstructured grids. The numerical simulations clearly show the formation of the tip vortex cavitation in the flow about the elliptic hydrofoil.

1. Introduction
In most practical situations, cavitation can and typically does occur when the local static pressure of a liquid drops below the saturation pressure. During an unsteady process, vapor cavities appear in the low pressure zones and subsequently disappear in the areas with higher pressure. Cavitation may cause a number of problems in hydraulic systems, including noise, surface erosion and loss of turbomachinery efficiency. However, besides these undesirable effects, cavitation is used advantageously in medical applications such as e.g. ultrasound cavitation, and lithotripsy as well as in some industrial processes for cleaning of surfaces and the dispersion of particles in a liquid. To control effects of cavitation in these applications, it is essential to understand the structure and mechanisms of the cavitation phenomenon. Following Saurel et al. [1], Schnerr et al. [2], and Schmidt et al. [3] we employ an equilibrium cavitation model, a compressible single-fluid flow model which does not depend on empirical constants. A computational method is presented to numerically simulate an unsteady cavitating flow, see also [5]. The method is applied to predict the unsteady tip vortex cavitation occurring in the flow about Arndt’s elliptic hydrofoil. The numerical simulations are performed by using the compressible flow URANS equations (and RANS equations for the case of fully wetted flow) which are closed with the thermodynamic state relations based on the employed equilibrium cavitation model.
2. Physical model

The Unsteady Reynolds Averaged Navier Stokes (URANS) equations are employed for the modeling of turbulence, which in integral conservation form are given by

$$\frac{\partial}{\partial t} \int_\Omega \mathbf{U} \, d\Omega + \int_{\Gamma=\partial\Omega} \mathbf{F}_j(\mathbf{U}) \, n_j \, d\Gamma = \int_{\Gamma=\partial\Omega} \mathbf{F}^\nu_j \, n_j \, d\Gamma. \quad (1)$$

Here, it is assumed that \(\Omega\) is a bounded stationary polygon domain in \(R^3\) with boundary \(\partial\Omega\), \(\mathbf{U} = [\rho, \rho u_i, \rho E]\) denotes the vector of conservative variables, with \(\rho\) the density, \(u_i\) a component of the velocity vector, and \(E\) the specific total energy. The \(\mathbf{F}_j(\mathbf{U}) \, n_j\) and \(\mathbf{F}^\nu_j \, n_j\) are the normal components of the inviscid and viscous flux vectors, respectively, which in Cartesian coordinates can be written as:

$$\mathbf{F}_j(\mathbf{U}) \, n_j = \begin{bmatrix} \rho \hat{u} \\ \rho \hat{u} n_i + p n_i \\ \rho \hat{u} H \end{bmatrix}, \quad \mathbf{F}^\nu_j \, n_j = \begin{bmatrix} 0 \\ \hat{\tau}_{ij} n_j \\ \hat{\tau}_{jk} u_k n_j - \hat{q}_j n_j \end{bmatrix}, \quad (2)$$

where \(\hat{u} = u_j n_j\) is the velocity normal to the surface \(\Gamma = \partial\Omega\), \(\hat{\tau}_{ij}\) and \(\hat{q}_j\) are the components of the viscous stress tensor and the heat flux which can be written as \(\hat{\tau}_{ij} = \mu_{\text{tot}} \left( \frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right) - \frac{2}{3} \delta_{ij} \frac{\partial u_k}{\partial x_k} \), and \(\hat{q}_j = -\mu_{\text{tot}}^* C_p \frac{\partial T}{\partial x_j}\). Moreover, \(\mu_{\text{tot}}\) is the total viscosity defined by \(\mu_{\text{tot}} = \mu + \mu_t\), with \(\mu\) and \(\mu_t\) the dynamic and turbulent viscosities, respectively, \(\mu_{\text{tot}}^* = \mu_{\text{tot}} / Pr + \hat{\mu} / Pr\), with \(Pr\) and \(\hat{Pr}\) the dynamic and turbulent Prandtl numbers, respectively, and \(C_p\) is the specific heat at constant pressure.

It is assumed that the transport properties are constant for both the liquid and vapor phases. The mixture transport properties are obtained via a blending of the properties of the liquid and vapor phase. The blending function used to compute the mixture values of \(\mu\) and \(Pr\) is the vapor void fraction \(\alpha\), while the mass fraction of the vapor \(f_v = \frac{\rho_v}{\rho}\) is used to compute the mixture values of \(C_p\). For example, the mixture \(\mu\) can be computed from \(\mu_{\text{mix}} = \alpha \mu_v + (1 - \alpha) \mu_l\). The turbulent viscosity \(\hat{\mu}\) is computed from a turbulence closure model. In this paper we employ two-equation Menter’s Shear Stress Transport (SST) model.

The URANS equations require additional thermodynamic relations in order to close the system of equations. The equilibrium cavitation model [1], [2], [3] is employed, which in the two-phase flow region assumes local thermodynamic and mechanical equilibrium. Note that in this model the phase transition does not depend on empirical constants, and a cavitation threshold at \(p = p_{\text{sat}}\) is assumed. The computational method assumes Tait’s equation of state for the liquid phase, perfect gas for the vapor phase, and an equilibrium model for the mixture phase (see e.g. Schmidt et al. [3] and Koop [4]). Khatami et al. [5] showed that these thermodynamic equations are highly computational-intensive. They replaced this approach by using precomputed multiphase thermodynamic tables containing the same thermodynamic information, and they developed a fast, accurate, and efficient method (POITC) [5] for the computational use of these tables. This computational approach is utilized in this paper.

3. Results

The computational method uses a cell-centered unstructured finite-volume method applied on a structured grid with the MUSCL-type reconstruction scheme (Van Leer [6]), and the AUSM+up for all speeds (Liou [7]) numerical flux scheme. To ensure the monotonicity of the solution, the Venkatakrishnan limiter [8] is employed. The temporal discretization is carried out using a third-order accurate three-stage TVD Runge-Kutta scheme (Shu [9]). The time step used in
Table 1: Conditions for the cavitating flow around Arndt’s hydrofoil at 7 deg AOA.

| $U_\infty$ [m s$^{-1}$] | $p_\infty$ [10$^5$ Pa] | $T_\infty$ [K] | $\rho_\infty$ [kg m$^{-3}$] | $c_\infty$ [m s$^{-1}$] | $\sigma$ [-] |
|-------------------------|------------------------|-------------|------------------------|------------------------|-----------|
| 50                      | 12.5                   | 293         | 998.7                  | 1540.0                 | 1.0       |

the numerical simulations must be taken very small. This is because the equilibrium cavitation model takes into account the physics of both the incompressible (liquid) and compressible (vapor and mixture) flows, which are solved using the compressible flow approach. For such a flow the Mach number ranges from low values in the liquid (low-Mach number flow), with a large value of speed of sound $c$ (e.g. $c \approx 1500$ m s$^{-1}$), to relatively high values in the mixture, i.e. with low values of speed of sound (e.g. with $c < 1$ m s$^{-1}$). Therefore, in order to resolve the dynamics of the pressure waves, which are directly related to the quantity $\rho c$ (see e.g. [3]), and to satisfy the appropriate CFL condition for numerical stability, in particular for low-Mach number flow, the time steps that are needed are extremely small.

3.1. Tip vortex cavitation on elliptic hydrofoil

The numerical simulation of vortex cavitation in the flow about the elliptic Arndt’s hydrofoil (NACA662 − 415) [11], [12] at 7 deg AOA are presented. A cavitation number of $\sigma = 1$ is used, where $\sigma$ is defined by $\sigma = \frac{p_\infty - p_v}{\frac{1}{2} \rho_\infty U_\infty^2}$ with $p_v$ the vapor pressure, and $p_\infty$, $\rho_\infty$, and $U_\infty$ denoting the free-stream pressure, density, and velocity, respectively. The free-stream velocity and pressure are chosen at values higher than the standard conditions (keeping the cavitation number constant), in order to accelerate the cavitation cycles [4]. The initial solution for the unsteady compressible cavitating flow is the steady fully-converged fully-wetted flow solution. Prior to running the main unsteady simulations, it was checked whether or not the flow solution is indeed unsteady. Therefore, the numerical simulations were carried out using an approach for steady flow. It was observed that, when using this approach the simulations did not converge and the system showed an entirely unsteady-like behavior. Hence, an approach for time-accurate unsteady flow is pursued.

Results are presented in Figure 1, which are obtained using the grid shown in Figure 1a. For this grid, the value of $y^+$ at the tip of the hydrofoil has everywhere on the surface a value close to unity. However, it has a maximum value of $y^+ \approx 50$. Furthermore, Figure 1b (top) depicts isosurface of the negative of the pressure coefficient $C_p$, with $C_p \equiv \frac{p - p_\infty}{\frac{1}{2} \rho_\infty U_\infty^2}$, together with the streamlines on the suction side. Experimental studies [12] for similar flow conditions show that the vortex cavitates up to the farfield of the domain. However, from the plots in Figure 1, in particular from Figure 1c (top), it is observed that after a short distance downstream of the hydrofoil, the vorticity magnitude is decreased at a rather high rate. This results in a less compact vortex core with a static pressure that does not reach the low values required for cavitation to occur. The reason for this is mainly due to artificial dissipation that is introduced by the employed numerical approach, in particular the MUSCL reconstruction scheme. In order to improve the results one may use a finer grid, which is in particular refined (far enough from the hydrofoil) downstream of the hydrofoil in the areas in which the formation of the vortex core takes place and cavitation is expected to occur.

4. Conclusions

Results of compressible flow numerical simulations of the tip vortex cavitation in the flow about Arndt’s elliptic hydrofoil have been presented. The URANS equations for compressible flow have been used which are closed with an equilibrium cavitation model. Furthermore, the
thermodynamic state of the system has been characterized by using the precomputed multiphase thermodynamic tables containing data from the appropriate equations of state in the liquid, mixture, and vapor phases and the fast, accurate, and efficient look-up approach of POITC [5] has been adopted. The equations have been discretized using a cell-centered unstructured finite volume method on a structured grid. The numerical results clearly show the formation of tip vortex cavitation for the elliptic hydrofoil at $\sigma = 1$ and 7 deg AOA.

Acknowledgment
This research has been carried out under the sponsorship of AgentschapNL in the framework of the Maritime Innovation Program, MIP-IOP IMA10007.

References
[1] Saurel R, Cocchi J and Butler J 1999 J. Propulsion and power 15(4) 513-22
[2] Schnerr G, Schmidt S, Sezal I and Thalhamer M 2006 Proceedings CAV2006 Sixth International Symposium on Cavitation (Wageningen: The Netherlands)
[3] Schmidt S, Sezal I and Schnerr G 2006 ECCOMAS CFD (Egmond aan Zee: The Netherlands)
[4] Koop A 2009 Numerical Simulation of Unsteady Three-Dimensional Sheet Cavitation (PhD thesis, University of Twente: The Netherlands)
[5] Khatami F, van der Weide E and Hoeijmakers H 2015 Heat Transfer Engineering (special topic issue), 36(12) 1065-83
[6] Van Leer B 1979 J. Comput. Phys., 32 101-36
[7] Liou M 2006 J. Comput. Phys. 214 137-70
[8] Venkatakrishnan V 1995 J. Comput. Phys. 118 120-30
[9] Shu C-W 1988 SIAM J. Sci. and Stat. Comput. 9 1073-84
[10] Roe P L 1981 J. Comput. Phys. 43 357-72
[11] Maines B and Arndt R 1997 J. Fluids Engineering 119(2) 413-19
[12] Maines B 1995 Tip Vortex Formation and Cavitation (PhD thesis, The University of Minnesota: USA)