Numerical investigation of shedding partial cavities over a sharp wedge

B Budich¹, S Neuner¹, S J Schmidt¹ and N A Adams¹

¹ Institute of Aerodynamics and Fluid Mechanics, Department of Mechanical Engineering, Technische Universität München, Boltzmannstr. 15, D-85748 Garching b. München

E-mail: bernd.budich@tum.de

Abstract. In this contribution, we examine transient dynamics and cavitation patterns of periodically shedding partial cavities by numerical simulations. The investigation reproduces reference experiments of the cavitating flow over a sharp wedge. Utilizing a homogeneous mixture model, full compressibility of the two-phase flow of water and water vapor is taken into account by the numerical method. We focus on inertia-dominated mechanisms, thus modeling the flow as inviscid. Based on the assumptions of thermodynamic equilibrium and barotropic flow, the thermodynamic properties are computed from closed-form analytical relations. Emphasis is put on a validation of the employed numerical approach. We demonstrate that computed shedding dynamics are in agreement with the references. Complex flow features observed in the experiments, including cavitating hairpin and horse-shoe vortices, are also predicted by the simulations. Furthermore, a condensation discontinuity occurring during the collapse phase at the trailing portion of the partial cavity is equally obtained.

Introduction

The understanding and assessment of transient cavity dynamics play a key role for a broad range of applications, such as hydraulic machinery or naval propulsors. Cavitation decreases the efficiency of fluid machinery, causes pressure oscillations in conduits and leads to noise and structural vibrations. Moreover, the occurrence of transient cavitating structures is frequently connected to material erosion, thereby limiting the lifespan of the affected components. Thus, an accurate prediction of transient cavitation phenomena by numerical methods is desirable in order to complement experimental work in the design process.

For an investigation of the cavitating flow over a sharp wedge, Ganesh [1] recently used X-ray densitometry in order to acquire a spanwise-averaged, time-resolved void fraction field. For intermittently shedding and periodic shedding partial cavities, experimental evidence for an upstream-propagating condensation discontinuity is obtained.

With the present numerical study, we reproduce the experiments by Ganesh [1] at an upstream cavitation number $\sigma_0 = 1.94$, which exhibits periodic shedding. In a previous numerical investigation of a NACA0015 hydrofoil by Schmidt et al. [2], a homogeneous mixture approach was applied. Utilizing an inviscid flow model, it has been shown by the authors that the condensation discontinuity can be predicted numerically. Therefore, the same strategy is pursued for the current contribution. Focus is put on the validation of the employed numerical model by comparing cavitation patterns and flow dynamics.
Numerical Method
A density-based, finite volume method in combination with a homogeneous mixture model [2, 3] is applied to the simulation of the two-phase flow of liquid water and water vapor. It is assumed that the flow is predominantly inertia-driven, i.e. viscous effects are neglected. Furthermore, local mechanical and thermodynamic equilibrium is assumed. Solved and non-condensable gas content is neglected and the flow is considered as barotropic. Phase change in two-phase regions, treated as saturated mixtures, is modeled as isentropic.

The governing equations are the unsteady, compressible Euler equations, spatially discretized on structured, body-fitted grids. Time integration is performed by a 4-stage Runge-Kutta method. A modified Tait-equation [4] is utilized for computing the pressure \( p(\rho) \) in the pure liquid. For water-vapor mixtures, the pressure is obtained by integrating the equilibrium speed of sound \( c^2 = \frac{\partial p}{\partial \rho}|_{s} \). The void fraction \( \alpha \) is computed using the mixture density \( \rho \) and the densities for the saturated liquid \( \rho_{l,sat} \) and vapor \( \rho_{v,sat} \) as \( \alpha = (\rho - \rho_{l,sat})/(\rho_{v,sat} - \rho_{l,sat}) \). The values of \( \rho_{l,sat} \) and \( \rho_{v,sat} \) are evaluated at a constant reference temperature \( T_{ref} = 20^\circ C \).

Numerical Setup
The experimental facilities are described in detail by Ganesh [1]. A sketch of the test section geometry is depicted in Fig. 1. The computational domain encompasses the double contraction leading to the reduced test section with a 76.2 × 76.2 mm cross section and a sharp step in the downstream portion. The wedge, with a contraction angle \( \varphi_1 = 22.1^\circ \), diffuser angle \( \varphi_2 = 8.13^\circ \) and a height \( h = 25.4 \) mm, is mounted on the lower wall. The full domain is shown in Fig. 2. The inlet is located at 1.5 m upstream of the wedge apex. A homogeneous inflow velocity is prescribed such that the experimentally measured velocity at position \( \mathbf{1} \), \( v_1 = 8 \) m/s, is matched. A large volume at the downstream boundary reduces wave reflections to the interior. An asymptotic outlet pressure boundary condition is imposed, such that the pressure at position \( \mathbf{2} \), \( p_2 = 52 \) kPa, agrees with the experiments. Remaining mesh boundaries are modeled as slip-walls.

A grid study involving four grid levels is conducted, with a total of \( 45 \cdot 10^6 \) cells on the finest level. In order to reduce the computational cost, a grid sequencing method is employed. In the vicinity of the wedge, as depicted in Fig. 3, the flow field is discretized on the finest level with an average minimum cell length of 0.3 mm in wall-normal and 0.07 mm in wall-tangential direction.

Figure 1. Test section geometry.

Figure 2. Overview of the computational domain, with upstream ducting and a double contraction, the test section with wedge, and the downstream diffuser.

Figure 3. Numerical grid in the vicinity of the wedge (finest grid level, every 4th grid line shown).
Results and Discussion

Onset of cavitation is observed at the sharp apex of the wedge. At the chosen operating conditions, a partial cavity develops and experiences periodic shedding. A characteristic Strouhal-number of $S_t = 0.28$ is obtained, which is in close agreement with experimentally reported values [1]. The shedding process is visualized in Figs. 4(a-e) by means of $\alpha = 0.1$ iso-surfaces, with a time interval of $\Delta t \approx 7.5 \cdot 10^{-3}$ s between selected frames. The maximum extent of the attached cavity sheet at $t = t_0$ is displayed in Fig. 4(a). While fragmented vapor structures from the previous cycle are convected further downstream, the attached part collapses towards the wedge apex. At $t = t_0 + 2\Delta t$, Fig. 4(c), the collapse front reaches the apex and the cavity completely detaches, leading to the generation of a new separated cloud. At $t = t_0 + 4\Delta t$, Fig. 4(e), a new partial cavity develops at the apex.

The simulation exhibits complex flow patterns that are also observed experimentally. Highlighted in Fig. 4, this includes cavitating horse-shoe vortices, Fig. 4(b,e), cavitation of streamwise-oriented hairpin vortices, Figs. 4(c,e), and crescent-shaped regions, Fig. 4(a,d,e). The latter are propagating shock wave structures occurring for localized collapses of vapor clouds [5]. In Fig. 5, vortical structures, visualized by iso-surfaces of vorticity magnitude $|\omega| = 4000 \text{ s}^{-1}$ (colored by axial velocity), are compared to iso-surfaces of 10% vapor volume fraction. Cavitating structures are closely connected to regions of increased vorticity.

In addition to a shedding mechanism involving a re-entrant flow, Ganesh [1] observes a condensation discontinuity propagating upstream towards the wedge apex. This mechanism is also predicted by the simulations. Figure 6 shows the spanwise-averaged instantaneous void fraction $\langle \alpha \rangle$ (left) and flow velocity tangential to the wedge surface $\langle u || \rangle$ (right), with vectors of spanwise-averaged local flow velocity $\langle u \rangle$.
fraction $\langle \alpha \rangle$ and the flow velocity parallel to the wedge surface $\langle u_{||} \rangle$. A discontinuity in $\langle \alpha \rangle$ spans across the complete height of the cavity, separating the attached part from a detached cloud. Downstream of this discontinuity, the flow is directed towards the leading edge, with a maximum velocity of approximately $-4$ m/s. Behind the discontinuity, $\langle \alpha \rangle$ does not fully vanish. This can be accredited to the averaging procedure, as a noticeable variation in the location of the discontinuity in the spanwise direction is observed. On individual slices in the $x-y$ plane (omitted here for brevity), complete condensation to $\alpha = 0$ is found.

Figure 7 shows the $s-t$-diagram of $\langle \alpha \rangle$, recorded on a plane at distance $d = 2$ mm parallel to the wedge surface. Cavity growth and collapse phases can easily be identified, as indicated by lines of positive and negative slope, respectively. For the cavity growth, a characteristic speed of $\approx 7$ m/s is recorded for the observed shedding cycles. In close agreement with the experiments, the condensation discontinuity travels towards the apex with an average bulk velocity of $-3.5$ m/s, accelerating during the late collapse phase. The void fraction reaches maximum instantaneous values of $\langle \alpha \rangle \approx 0.95$, which is consistent with the experimental observations.

In summary, very good agreement between the experiments and the simulations is found. Experimentally observed flow and cavitation patterns as well as cavity dynamics are reproduced numerically, including the condensation discontinuity. It is concluded, that the chosen homogeneous mixture model is well suited for the simulation of shedding partial cavities.

**Acknowledgments**

Support for this research was provided by the U.S. Office of Naval Research (ONR) and ONR Global through the NICOP project entitled “Numerical Investigation of Ship-Propeller Cavitation with Full Description of Shock-Wave Dynamics” (Contract No. N62909-12-1-7034). The authors are grateful for Dr. Ki-Han Kim (ONR HQs) and Dr. Woei-Min Lin (ONR Global) for their support and guidance throughout this project. The authors further greatly acknowledge Prof. Steven Ceccio and Dr. Harish Ganesh for sharing data from the experiments and for the fruitful discussions on the subject. Computing time was provided by the Gauss Centre for Supercomputing e.V. (www.gauss-centre.eu) on the GCS Supercomputer SuperMUC at the Leibniz Supercomputing Centre (LRZ, www.lrz.de).

**References**

[1] Ganesh H 2015 *Bubbly Shock Propagation as a Cause of Sheet to Cloud Transition of Partial Cavitation and Stationary Cavitation Bubbles Forming on a Delta Wing Vortex* Ph.D. thesis University of Michigan

[2] Schmidt S J, Thalhammer M and Schnerr G H 2009 7th International Symposium on Cavitation, CAV 2009 (Ann Arbor, Michigan, USA)

[3] Schmidt S J, Sezal I H, Schnerr G H and Thalhammer M 2008 46th AIAA Aerospace Sciences Meeting and Exhibit, ASME 2008

[4] Sezal I H 2009 *Compressible Dynamics of Cavitating 3-D Multi-Phase Flows* Ph.D. thesis Technische Universität München

[5] Reisman G E and Wang Y C 1998 *Journal of Fluid Mechanics* 355 255–283