NUMERICAL SMULATION OF LEADING EDGE CAVITATION OVER 2D HYDROFOIL

Youcef AIT BOUZIAD
Mohamed FARHAT
François AVELLAN,

Swiss Federal Institute of Technology
Laboratory For Hydraulic Machines
Lausanne, Switzerland

ABSTRACT

in the present paper, we compare two different computation methods for the simulation of leading edge cavitation. In the first one, known as interface tracking method, the cavity interface is taken as a free surface boundary in the computation domain and the calculation is performed in a single phase flow. The cavity shape is then determined apart from the flow calculation using an iterative procedure. We have used the *Neptune* code which derive the initial cavity shape from the envelope of a travelling bubble along the hydrofoil suction side. The second method is based on the so-called interface-capturing scheme. The TASCflow industrial code that we have used assumes a constant enthalpy (CEV) during the vaporization and condensation processes. Both methods are tested in the case of an isolated 2-D hydrofoil (NACA0009) having 100 mm chord length and 150 mm span. Both models gave good prediction of the cavity length and pressure distribution. Nevertheless, the CEV model exhibits significant instabilities when the cavity extends beyond the hydrofoil mid-chord. Moreover, results obtained with the *Neptune* predict well the pressure distribution near the cavity detachment. Either methods do not allow a good prediction of the drag coefficient nor the drop of the lift due to cavitation.

RÉSUMÉ

Le but de la présente étude est de comparer deux différentes méthodes de simulation de la cavitation de bord d'attaque. Dans la première méthode, connue sous le nom de suivi d'interface, l'interface de la cavité est considérée comme une surface libre et le calcul est effectué en écoulement monophasique. Nous avons utilisé le logiciel *Neptune* dans lequel la forme initiale de la poche est estimée par l'enveloppe d'une bulle évoluant le long du profil. Pour la deuxième méthode, dite, à capture d'interface, le logiciel industriel TASCflow, qui est basé sur un modèle de vaporisation à enthalpie constante dans le processus de vaporisation-condensation (CEV) a été utilisé. Les deux méthodes sont testées dans le cas d'un profil isolé bidimensionnel (NACA0009) de 100 mm de corde et de 150 mm d'envergure. Une bonne prédiction de la longueur de la poche et de la distribution de pression a été obtenue par les deux méthodes. Toutefois, le modèle CEV présente de fortes instabilités de calcul lorsque la longueur de la poche dépasse la moitié du profil. En outre, le code Neptune permet une bonne prédiction de la zone de pression négative à l'amont du point de détachement. Par ailleurs, aucune des deux méthodes ne permet le calcul du coefficient de traînée ni de la chute du coefficient de portance.

NOMENCLATURE

Term	Symbol	Definition
Reference velocity	C	[m/s]
Lift coefficient	C_L	[-]
Incidence angle	α	[°]
Chord	L	[m]
Cavity thickness	e	[m]
Enthalpy	h	[m²/s²]
Pressure coefficient	c_p	$c_p = \frac{p - p_{ref}}{\frac{1}{2}\rho C^2}$

Term	Symbol	Definition
Static pressure	p	[Pa]
Drag coefficient	C_D	[-]
Reference pressure	p_{ref}	[Pa]
Cavity length	l	[m]
Density	ρ	[Kg/m³]
Liquid mass fraction	y_l	[-]
Cavitation number	σ	$\sigma = \frac{p_{ref} - p_V}{\frac{1}{2}\rho C^2}$

INTRODUCTION

The leading edge cavitation, which may develop in hydraulic machines, is often responsible of erosion, noise and vibration as well as deterioration of hydrodynamic performances. Although the numerical modelling of such a cavitation has received a great deal of attention, it is still not possible to predict such complex unstationnary and two-phase flows with an acceptable accuracy. Early studies in cavitation modelling were based on the potential flow theory and are still used in various engineering applications. In the last few years, studies were more focused on the single-fluid Navier-Stokes equations. An extensive overview of existing cavitation models is provided by Senocak et. al. (Ref. 11). Two different approaches have been mainly proposed.

In the first approach, called interface tracking or fitting model, the cavity interface is considered as a free surface boundary of the computation domain. The cavity is and deformed in an iterative way to reach the vapour pressure at its border. Obviously, the initial shape of the cavity has to be provided. Chen et. al. (Ref. 2) used a simple criteria ($p<p_v$). Hirschi et. al. (Ref. 6) proposed an approximation of the initial cavity shape by the envelope of a travelling bubble and obtained promising results in estimating the main cavity dimensions for different geometries such as isolated hydrofoils and hydraulic machines.

The second approach is based on an interface-capturing model where the vapour-liquid interface is directly derived from the flow calculation. In this approach, a pseudo-density function of the liquid-vapour mixture is used to close the equations system. Kubota et. al. (Ref. 7) introduced the pseudo density calculation with the help of Rayleigh-Plesset model. Delannoy et. al. (Ref. 3) proposed a simple barotropic law. Merkle et. al. (Ref. 9) and Kunz et. al. (Ref. 8) have introduced an additional equation for the void fraction in order to obtain the local mixture density according to actual phase composition.

In the present study, we have tested both approaches. The interface tracking method that we have used is called *Neptune* (Ref. 6) and has been developed at EPFL. The interface-capturing method, which has been tested, is part of the *CFX-TASCflow* commercial (Ref. 1) code where a constant enthalpy is assumed for both vaporisation and condensation processes.

THE CASE STUDY

The case study is a symmetric 2D NACA0009-7.38 45/1.95 hydrofoil having 100 mm chord length and 150 mm span. The hydrofoil is placed in the test section of the EPFL high-speed cavitation tunnel, which is 150 mm x 150 mm x 750 mm.

THE CAVITY INTERFACE FITTING MODEL: NEPTUNE

Initial Cavity shape

The initial shape of the vapour cavity is estimated by the envelope of a travelling bubble along the suction side of the hydrofoil. The Rayleigh-Plesset model is used to calculate the evolution of a nucleus placed in infinite water volume. The driving pressure field is derived from the cavitation free calculation along a mesh line. It should be noticed that only half of the bubble diameter is considered for cavity thickness. This is in accordance with former experimental observations (Farhat et al., Ref. 4, Ref. 5).

The use of the envelope of a travelling bubble for initial cavity estimation is justified by the physics of leading edge cavitation. In fact, we have already shown how attached cavity may originate from a smooth and continuous transition from bubble to sheet cavitation (Ref. 4). We have presented in Fig. 1 an illustration of this transition process in the case of a cavitating flow over a 2-D hydrofoil. The flow velocity is 15 m/s and the sigma value is 0.9. As the incidence angle is increased from 0.5° to 2°, the bubble cavitation turns into an attached cavity in a continuous way. Moreover, the length of the attached cavity is found to be very close to the length of the bubbles envelope.

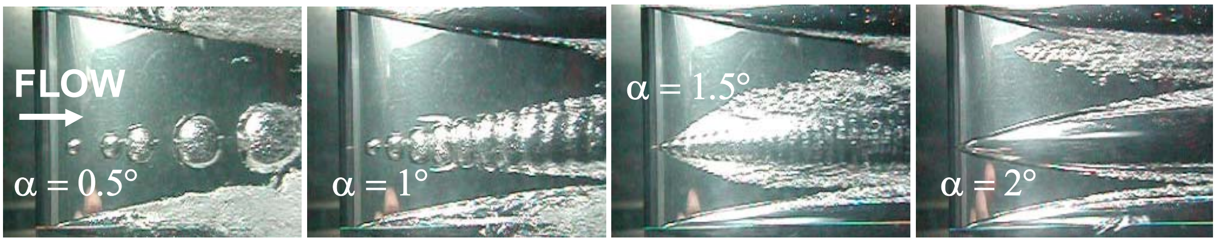


Fig. 1 Bubble to sheet cavitation transition on a 2D hydrofoil tested in the LMH Tunnel

Deformation Algorithm

Once the initial cavity shape is thus estimated, the liquid domain is re-meshed and the flow is calculated in the newly defined domain. The cavity interface is then deformed in an iterative way until the vapour pressure is reached in the cavity boundary. The deformation procedure is performed according to the pressure distribution on the blade obtained from the liquid flow computation. For a given cavitation number σ, the modified cavity thickness $\vec{e}$ at time step $t+1$ corresponding to the abscissa ξ along the streamline η is given by:

$$\vec{e}(\xi,\eta,t+1) = \vec{e}(\xi,\eta,t) + \lambda C_1 \, [c_p(\xi,\eta,t) + \sigma] \cdot \vec{n}(\xi,\eta,t)$$

where $\vec{n}$ is the normal vector to the cavity interface at the point (ξ,η,t) and λ is a function of the flow confining. C_1 is a factor depending on the relaxation coefficient given by the term $(c_p(\xi, \eta, t)+\sigma)$ and the local curvature $\Re$, which allows to avoid oscillations in high thickness gradients.

Cavity closure law

With the deformation algorithm presented above, it is not possible to change the cavity closure location. To overcome this difficulty, we assume that the cavity may be approximated from its maximum thickness to its closure by the envelope of a collapsing bubble. The initial radius of this bubble is taken equal to the maximum thickness of the cavity and the Rayleigh-Plesset equation is once again used.

THE TASCFLOW CAPTURING INTERFACE MODEL : CEV

The cavitation module is part of the *CFX-TASCflow* CFD code. Here, the cavitation phenomenon is assumed to follow a constant enthalpy vaporization-condensation process, beginning in the sub-cooled liquid region and expanding into the two-phase zone (Ref. 1).

For a given local temperature and with the local pressure p calculated by the resolution of full Navier-Stokes equations, saturation enthalpy values corresponding to the gas ($h_g(p)$) and the liquid ($h_l(p)$) are obtained from the liquid-phase diagram and the pseudo-density as well as the mass fraction of the liquid (y_l) are derived as follows:

$$\rho_e = \frac{1}{\frac{1}{\rho_e} + y_l(\frac{1}{\rho_l} - \frac{1}{\rho_g})} \qquad y_l = \frac{h_g(p) - h}{h_g(p) - h_l(p)}$$

with ρ_l and ρ_g denoting the liquid and the vapour densities.

RESULTS

CAVITY SHAPE

The cavity length *l* normalized with respect to the chord length *L*, is plotted in Fig. 2 versus the cavitation number σ. Numerical results computed with *Neptune* and *CEV* models are compared to the experimental measurements obtained by Pereira (Ref. 10). Reference velocity is 20 m/s for the numerical computations. In *CEV* computation, the cavity boundary is taken as the 0.9 iso-fraction of the liquid mass.

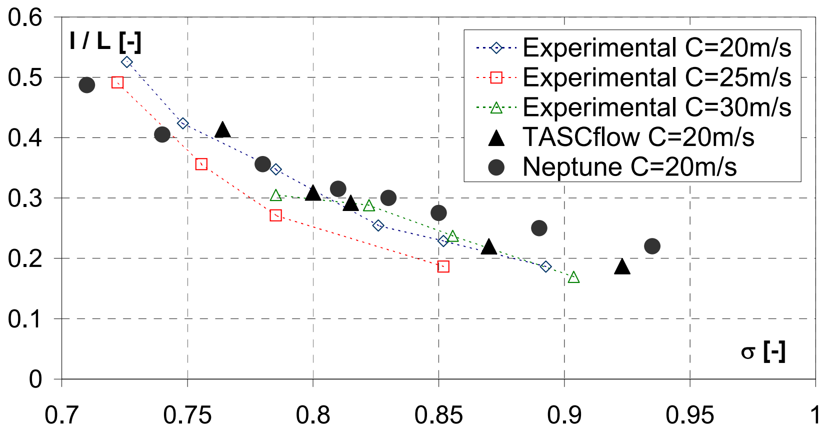


Fig. 2 Dimensionless cavity length l/L versus the cavitation number σ ; α=2.5

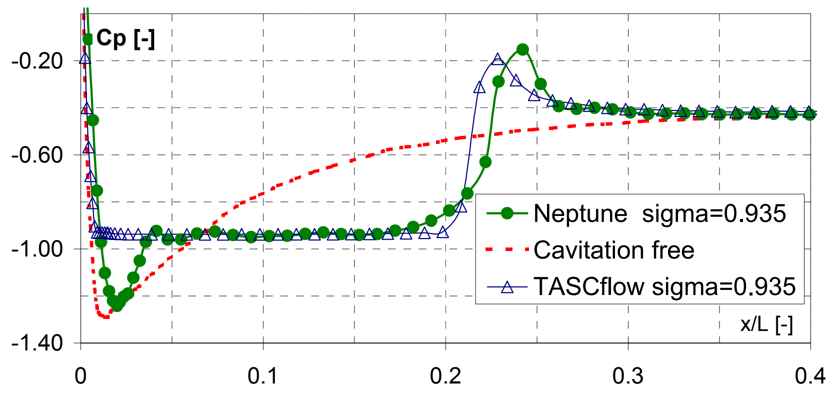
First, a significant influence of the flow velocity on the measured cavity length may be observed, specially for short cavities. This is due to the complex process of cavitation inception, which depends on many parameters such as Reynolds number, the surface roughness, flow instabilities and nuclei content. It should be noticed that none of these parameters is taken into account in both cavitation models addressed in the present paper.

The CEV model gives good predictions of cavity length compared to the experiments. The maximum dimensionless cavity length computed is 0.41, which corresponds to σ = 0.77. For longer cavities, the CEV model exhibits very unstable behaviour and requires much higher computation time to converge.

The curve *l/L*(σ) predicted by *Neptune* model gives a good agreement with the measurements for well developed cavities while a slight discrepancy is observed for small cavities.

PRESSURE DISTRIBUTION AND FLOW PATTERN

Computed pressure coefficients along the suction side of the hydrofoil corresponding to the non-cavitating flow and two different cavitating flows (σ = 0.94 and σ = 0.83) are reported in Fig. 3 as a function of the dimensionless chord (x/L).



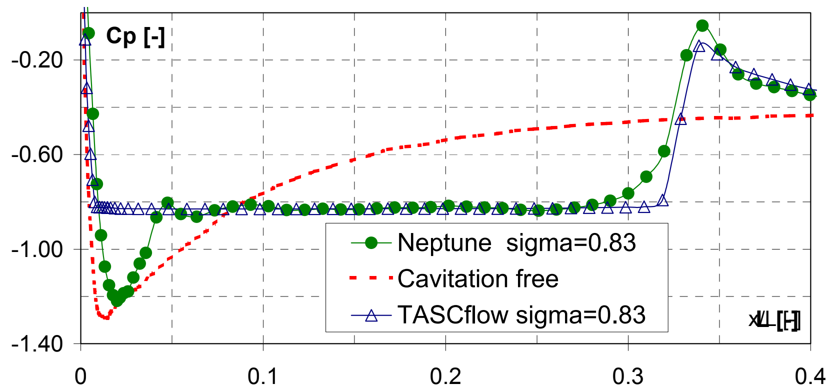


Fig. 3 Pressure distribution versus dimensionless abscissa for α=2.5; C=20 m/s

Cavity detachment

In the CEV model, since the transition to the vapour state is only function of the static pressure, the cavity detaches instantaneously as soon as the pressure equals the vapour pressure (Fig. 3).

In *Neptune* model the location of the cavity detachment is based on the explosion of a single nucleus. Here, the dynamic response of the nucleus to the pressure field is taken into account, which leads to the existence of a negative pressure zone located just upstream to the cavity detachment. This is in accordance with recent studies that have clearly demonstrated that the liquid close to the cavity detachment may withstand a significant tension (Ref. 4). Moreover, one may also observe that the cavity produces a slight increase of the minimum pressure value. This result is also in accordance with the experimental studies in Ref. 4. Nevertheless, the slight move of the minimum pressure location to the downstream direction is not yet well understood.

Cavity closure

The cavity closure is known to be strongly unsteady. The generation of transient cavities as well as the re-entrant jet phenomena are very difficult to compute by both numerical models. The two methods have different approaches to model the cavity closure.

In *Neptune* code, the closure region of the cavity is estimated by the envelope of a collapsing bubble. For this purpose, the Rayleigh-Plesset model is adopted. This approach used with mono-fluid assumption gives good agreements with experiments for cavity length and thickness in steady state computation. In the case of CEV model, the cavity closure is a region of fluid-vapour mixture. For high values of the cavity length ($l/L \geq 0.4$), the mixture area is too thick and the cavity length vary significantly upon the threshold of the liquid mass fraction that defines the cavity border.

HYDRODYNAMIC FORCES

The lift (C_L) and drag coefficients (C_D) have been calculated by both Neptune and CEV models, and compared to experimental results (Fig. 4) .It should be noticed that the incidence angle is fixed to 2.5 ° and the theoretical cavitation index corresponding to the cavitation inception is 1.29.

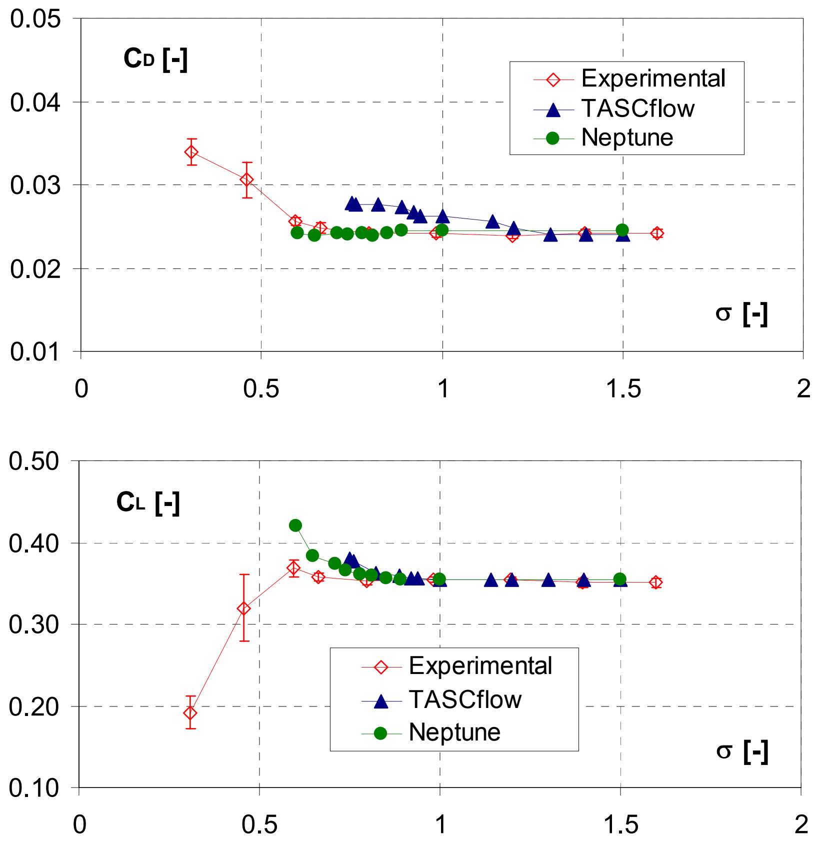


Fig. 4 Drag and lift coefficient versus σ . C=20 m/s, α=2.5°

For both *Neptune* and *CEV* models, the computation of the lift coefficient gives a good prediction for high cavitation numbers as far as the cavity length does not alter the hydrodynamic performances ($\sigma \geq 0.75$). The implemented models predict the threshold of the increase of the lift coefficient, but estimated values are higher that measured ones.

Since the cavitation occurs on the hydrofoil, the CEV model estimates a high increase of the drag coefficient, which is not in concordance with the experimental measurements, whereas Neptune gives a good prediction of the drag coefficient evolution for a large range of the cavitation number. It can be notified that the drag computation is more difficult since it is more sensitive to the flow modelling parameters (turbulence model, wall function).

Both models do not predict the drop of hydrodynamic performances. The main reason is that such a drop occurs when the vapour cavity reaches the trailing edge of the hydrofoil and both *Neptune* and *CEV* models are not suitable for predicting supercavitating flows. Moreover, it is well known that as the cavity length increases, strong instabilities take place leading to large cavity fluctuations. The steady state computation assumed in both models is no more valid.

CONCLUSION

Two methods allowing the numerical modelling of leading edge cavitation have been presented. Computation analysis for cavity shape, flow pattern and hydrodynamic forces have been performed and compared to experimental results. The two methods have been discussed with regards to the physics of leading edge cavitation phenomenon. The cavity length is well predicted by both models as far as the cavity does not extend beyond the hydrofoil mid-chord. Furthermore, the *Neptune* model allows the prediction of a negative pressure upstream to the cavity detachment as well as the increase of the minimum pressure as already reported in experimental works. Nevertheless, the prediction of the drop of hydrodynamic performances is not yet satisfactory by both models.

AKNOWLEDGMENT

Authors would like to thank Mitsubishi Heavy Industry for its technical and financial support.

REFERENCES

Ref. 1 CFX-TASCflow, V 2.10: Primer Documentation. AEA Technology, Canada, 2000.

Ref. 2 Chen Y., Heister S.D., 1994, “A numerical treatment for attached cavitation”, Journal of Fluids Engineering, 1994, vol.116, pp 613-618.

Ref. 3 Delannoy Y., Kueny J. L., 1990, “Two phase flow approach in unsteady cavitation modelling”, ASME Cavitation and Multi-phase Flow Forum, vol. 109, pp 153-159.

Ref. 4 Farhat M., Avellan F. “On the Detachment of a leading edge Cavitation”. Proc. of the 4th international Symposium on Cavitation, June 20-23, 2001, Pasadena, Ca, USA.

Ref. 5 Farhat M., Guennoun F., Avellan F., “The leading edge cavitation dynamics”. Proc. of ASME FEDSM 2002, Fluids Eng. Division Meeting Montreal, Canada.

Ref. 6 Hirschi R., Dupont F., Avellan F., 1998, “Centrifugal Pump Performance Drop Due to Leading Edge Cavitation: Numerical Predictions Compared With Model Tests”, Journal of Fluids Engineering, December 1998, vol.120, pp 705-711.

Ref. 7 Kubota A., Kato H., Yamaguchi H., 1992, “A new modelling of cavitating flows: a numerical study of unsteady cavitation on a hydrofoil section”, Journal of Fluid Mechanics, vol. 240, pp.59-96.

Ref. 8 Kunz R.F., Boger D.A, Stinebring D.R., Chyczewski T.S., Gibeling H.J., Govindan T.R., 1999, “Multi-phase CFD analysis of natural and ventilated cavitation about submerged bodies”, 3rd ASME/JSME Joint Fluids Engineering Conf., 1999, USA.

Ref. 9 Merkle C., Feng L., Buelow, 1998, “Computational Modeling of the Dynamics of Sheet Cavitation”, 3rd International Symp. on Cavitation, April 1998, Grenoble.

Ref. 10 Pereira F., 1997, “Prédiction de l’érosion de cavitation: Approche énergétique”, Ph.D. Thesis N° 1592, EPFL, Lausanne, Switzerland.

Ref. 11 Senocak I., Shyy W., 2002, “A Pressure-Based Method for turbulent Cavitating Flow Computations”, Journal of computational Physics, vol.176, (pp363-383).