PHYSICAL MODELLING AND SIMULATION OF LEADING EDGE CAVITATION, APPLICATION TO AN INDUSTRIAL INDUCER

Youcef AIT BOUZIAD, Mohamed FARHAT, Faïcal GUENNOUN, Jean-Louis KUENY, François AVELLAN
Laboratory for Hydraulic Machines, EPFL, Lausanne, Switzerland
E-mail : youcef.aitbouziad@epfl.ch

Kazuyoshi MIYAGAWA
Mitsubishi Heavy Industry, Takasago R&D center, Japan

ABSTRACT

In the present study, two CFD methods for cavitation modeling have been investigated and compared to experimental results in the case of 3-bladed industrial inducer. The first model is an interface tracking method and the second is a VOF model. It was found that both models allow a good prediction of the cavitation inception as well as the main cavity dimensions. The threshold corresponding to the head drop is also well predicted by both models. It was also found that the cavitation induced head drop is mainly due to an increase of energy losses and a decrease of the supplied energy. Furthermore, a meridian analysis of the rothalpy evolution clearly demonstrates that the energy losses are mainly located in the channel downstream to the cavity closure. The hydrodynamic mechanism of head drop is explained through a global and local analysis of the flow field. In fact, the leading edge cavitation causes a flow blockage as well as a flow imbalance as it reaches the throat. The pressure is significantly reduced at the pressure side of the leading edge, which allows cavitation occurrence and consequently influences the energy transfer.

INTRODUCTION

Cavitation plays an important role in design of hydraulic machines; it is often responsible of erosion, noise and vibration as well as deterioration of hydrodynamic performances. The so called “Sheet cavitation” or “leading edge cavitation” is commonly observed cavitation when a hydraulic machine operates under off design conditions. Although the numerical modeling of such a cavitation has received a great deal of attention, it is still very difficult and challenging task to predict such complex unsteady two-phase flows with an acceptable accuracy. Early studies in cavitation modeling 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. Three different approaches have been mainly proposed for leading edge cavitation simulation in hydraulic machinery. In the first approach, called ‘interface tracking’ or ‘Interface fitting’ model, the cavity interface is considered as a free surface boundary of the computation domain, and the computational grid includes only the liquid phase. The cavity is then deformed at every time step in order to reach the vapor pressure at its border assuming that no mass flux is allowed across the interface. Obviously, in this case, the initial shape of the cavity and a closure region models have to be provided. Hirschi et. al. [1] proposed an approximation of the initial cavity shape by the envelope of a bubble traveling close to the blade. Rayleigh-Plesset equation is used to compute bubble radius as it crosses the pressure field over the blade. This approach was successfully used to predict mean dimensions of the cavity in the case of different centrifugal pump configurations. The second approach is based on a ‘state-law’ model where the vapor-liquid interface is directly derived from the flow calculation. In this approach, a pseudo-density function of the liquid-vapor mixture is used to close the equations system. Delannoy and Kueny [2] have proposed a barotropic law relating pressure to density. This approach was tested in the case of centrifugal pumps and inducers [3]. Recently, a new cavitation modeling through a multiphase mixture model has been introduced. Merkle et. al. [4] introduced a ‘dual-species’ model having an additional equation for the volume fraction including source terms for vaporization and condensation processes. Several researchers [5-7] have used this physical model for several cavitation case studies such as hydrofoils and pumps.

In the present study, we have tested two approaches. The first one is an interface tracking method Neptune [1]. The second is a VOF transport-based model; the mass source term in the volume fraction transport equation is provided from a simple form of the RP equation to model the liquid/vapor mass transfer and is a part of the TASCflow [8] commercial software. A comparative study between the 2 models for different case studies with a comparison to experimental data is carried out. A brief theoretical formulation of the physical models is given and is tested in the case of a 3 bladed industrial inducer, where cavitation development at different cavitation number with the both models is compared to the experimental data, and a statement is made regarding to the ability of each model to predict the cavitation development and performances alteration. Analysis is provided concerning the head drop phenomena.
NOMENCLATURE

<table>
<thead>
<tr>
<th>Definition</th>
<th>Symbol</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density</td>
<td>( \rho )</td>
<td>[kg m(^{-3})]</td>
</tr>
<tr>
<td>Volume fraction</td>
<td>( \alpha )</td>
<td>[-]</td>
</tr>
<tr>
<td>Mass fraction</td>
<td>( y )</td>
<td>[-]</td>
</tr>
<tr>
<td>Mass source term</td>
<td>( m )</td>
<td>[kg s(^{-1})]</td>
</tr>
<tr>
<td>Initial Bubble radius</td>
<td>( R_0 )</td>
<td>[m]</td>
</tr>
<tr>
<td>Number of bubbles per volume unit</td>
<td>( N )</td>
<td>[m(^3)]</td>
</tr>
<tr>
<td>Condensation mass source factor</td>
<td>( F^c )</td>
<td>[-]</td>
</tr>
<tr>
<td>Vaporization mass source factor</td>
<td>( F^v )</td>
<td>[-]</td>
</tr>
<tr>
<td>Non dimensional meridian distance</td>
<td>( \sqrt{dr^2 + dz^2/r} )</td>
<td>[-]</td>
</tr>
<tr>
<td>Cylindrical coordinate components</td>
<td>((r, \theta, z))</td>
<td>[m, rad, m]</td>
</tr>
<tr>
<td>Impeller radius</td>
<td>( R )</td>
<td>[m]</td>
</tr>
<tr>
<td>Volume flow rate</td>
<td>( Q )</td>
<td>[m(^3) s(^{-1})]</td>
</tr>
<tr>
<td>Angular rotation speed</td>
<td>( \Omega, \omega )</td>
<td>[rad s(^{-1})]</td>
</tr>
<tr>
<td>Peripherical velocity</td>
<td>( U = \omega R )</td>
<td>[m s(^{-1})]</td>
</tr>
<tr>
<td>Meridian absolute velocity</td>
<td>( Cm )</td>
<td>[m s(^{-1})]</td>
</tr>
<tr>
<td>Tangential absolute velocity</td>
<td>( Cu )</td>
<td>[m s(^{-1})]</td>
</tr>
<tr>
<td>Gravitational acceleration</td>
<td>( g )</td>
<td>[m s(^{-2})]</td>
</tr>
<tr>
<td>Net pump head</td>
<td>( H )</td>
<td>[m]</td>
</tr>
<tr>
<td>Energy</td>
<td>( E = gh )</td>
<td>[j kg(^{-1})]</td>
</tr>
<tr>
<td>Torque</td>
<td>( T )</td>
<td>[N m]</td>
</tr>
<tr>
<td>Net positive suction Energy</td>
<td>( NPSE )</td>
<td>[j kg(^{-1})]</td>
</tr>
</tbody>
</table>

THEORETICAL FORMULATION AND MODELING

Cavity Interface Fitting Model, Neptune

The model is a mono-fluid interface tracking model where the cavity interface is considered as a free surface boundary with a vapor pressure value. Here, a short description is given for the model. Further information is given in Hirshi et al. [1].

Initial Cavity shape

The initial shape of the vapor cavity is estimated by the envelope of a traveling 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, and it should be noticed that only half of the bubble diameter is considered for cavity thickness. The use of the envelope of a traveling bubble for initial cavity estimation is justified by the physics of leading edge cavitation as already reported by Farhat et. al. [11].

Deformation Algorithm

Once the initial shape of the cavity is estimated, the cavity interface is then deformed in an iterative way until the vapor pressure is reached in the cavity boundary. At each time step, the liquid domain is re-meshed and the flow is calculated in the newly defined domain. The deformation procedure is performed according to the pressure distribution on the blade obtained from the liquid flow computation. For a given cavitation number \( \sigma \), the modified cavity thickness \( \tilde{e} \) at time step \( t+1 \) corresponding to the abscissa \( \xi \) along the streamline \( \eta \) is given by:

\[
\tilde{e}(\xi, \eta, t+1) = \tilde{e}(\xi, \eta, t) + \lambda C_1 \left[ C_p (\xi, \eta, t) + \sigma \right] \cdot \tilde{n}(\xi, \eta, t)
\]

Where \( \tilde{n} \) is the normal vector to the cavity interface at the point \((\xi, \eta, t)\) and \( \lambda \) 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.

To overcome the problem of the unsteady character of the closure region, 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

VOF transport-based model

In this model, which is implemented in CFX-TASCflow, a truncated form of the Rayleigh-Plesset (RP) equation is used and relies on the assumption that thermal and mechanical equilibrium exists between the liquid and vapor phases. The RP equation provides the basis for the rate equation controlling vapor generation/ destruction, and is implemented through a volume fraction equation with source a term using a volume-of-fluid (VOF) method.

The modeling of free surface flows is a common VOF application and much effort is spent in minimizing numerical diffusion at the interface. In the case of cavitation, no attempt is made to model a distinct liquid/vapor interface; the volume fraction field may vary continuously from 0 to 1 in the cavitation zone covering many grid elements.

Volume-Fraction Scalar Equations

The governing equations describe the cavitation process involving two-phase three-component system, where we assume no-slip between the phases, and thermal equilibrium between all phases. The three components are: vapor (v), water (w), and non-condensable gas in the form of micro-bubbles (g). The relative quantity of each of the components is described by a volume fraction scalar \( \alpha \), with:

\[
(\alpha_v + \alpha_w) + \alpha_g = 1
\]
Volume fractions are related to the mass fractions, \( y \), for each component \( i \) through the relations:

\[
y_i = \frac{\alpha_i \rho}{\rho} \quad 	ext{and: } \sum y_i = 1
\]

Based on the above description of a multiphase system, only two volume-fraction equations need be solved, since the distribution of the third phase can be determined from the additional constraint equation. In many cavitation problems the non-condensable gas phase is assumed to be well mixed in the liquid phase with a constant mass fraction \( y_g \). On this basis the mass fractions \( y_l \) and \( y_g \) can be combined and treated as one. This is the approach taken here and the volume scalar \( \alpha \) is introduced from this where \( \alpha = \alpha_w + \alpha_g \) and the density associated with \( \alpha \) becomes:

\[
\rho = \frac{1}{(1-y_g)/\rho_w + (y_g/\rho_g)}
\]

Choosing the scalar \( \alpha \) to solve, and remembering the need to consider compressibility effects, the governing volume-fraction equation for the primary liquid phase with non-condensable gas becomes:

\[
\frac{\partial}{\partial t} (\rho \alpha) + \frac{\partial}{\partial x_j} (\rho u_j \alpha) = \dot{m}_l = \dot{m}_l^V + \dot{m}_l^C
\]

where: \( \alpha = 1-\alpha_g \quad \text{and: } \dot{m}_l = -\dot{m}_v \)

The source terms \( \dot{m}_l \) and \( \dot{m}_v \) have units of kg/s and account for mass exchange between the vapor and liquid during cavitation. The form of the source term \( \dot{m}_v \) for the standard model has been derived by considering the Rayleigh-Plesset equation for bubble dynamics.

**Cavitation Source Term: Rayleigh-Plesset Model**

Using the same idea as Singhal [7], the cavitation model has been implemented based on the use of the Rayleigh-Plesset equation to estimate the rate of vapor production; the formulation listed here is based on the bubble number density development instead of the volume fraction. For a vapor bubble nucleated in a surrounding liquid the dynamic growth of the bubble can be described by the RP equation as follows:

\[
\dot{R} = \frac{3}{2} \frac{R}{R^3} = 2 \frac{p_v - p}{\rho_l}
\]

Where \( R \) is the radius of the bubble, \( p_v \) the vapor pressure in the bubble, \( p \) the pressure in the surrounding liquid and \( \rho_l \) the liquid density. For the present model we assume that \( p_v \) is at the vapor pressure corresponding to the temperature of the liquid. The above nonlinear ordinary differential equation is difficult to implement within an Eulerian-Eulerian framework for multiphase flows, therefore a first order approximation is used where:

\[
\dot{R} = \sqrt{\frac{2}{3}} \frac{|p_v - p|}{\rho_l} \quad \text{and: } \dot{m}_l = N \rho_l 4\pi R_l^2 \dot{R}
\]

The growth or collapse of a bubble follows the RP equation, neglecting higher order terms and bubble interactions. The number of bubbles per unit volume of the mixture, \( N \), available as nucleation sites is given by:

\[
\frac{\dot{N}^V}{4\pi R_0^4} = \frac{3\alpha_g}{4\pi R_0^4} \alpha_g + \alpha_l - \alpha_l
\]

And during condensation:

\[
\frac{\dot{N}^C}{4\pi R_0^4} = \frac{3\alpha_g}{4\pi R_0^4} \alpha_g + \alpha_l
\]

In practice, the vaporization and condensation processes have different time scales. Empirical constants, \( F^V \) and \( F^C \), are introduced to allow for these constraints. The definition of \( N \) changes depending on the direction of the phase change.

\[
\begin{align*}
\dot{m}_l^V &= F^V 4\pi \rho_l R_l^2 \frac{2}{3} \text{Max} \left( \frac{p_v - p}{\rho_l} \right) \\
\dot{m}_l^C &= F^C 4\pi \rho_l R_l^2 \frac{2}{3} \text{Max} \left( \frac{p_l - p}{\rho_l} \right)
\end{align*}
\]

The non-condensable gas, assumed as spherical bubbles, provide nucleation sites for the cavitation process. The default value for \( y_g \) is taken equal to \( 10^{-5} \). A typical initial radius for the nuclei is \( R_0 = 10^{-2} \) m.

Since no reliable data is available for values for source terms, we have used a simple and well documented 2D hydrofoil case study to derive optimized values, which allow the best prediction of cavity dimensions (see Ait Bouziad et al. [10]). We have obtained \( F^V = 50 \), for vaporization \((p_v - p > 0)\), and \( F^C = 0.015 \) for condensation \((p_l - p < 0)\).

**INDUCER CASE STUDY**

**Experimental setup**

![Inducer test loop facility](image)

Fig. 1 Inducer test loop facility

The present case study refers to a 3-bladed industrial inducer. Measurements of hydraulic performances as well as cavitation visualization have been achieved in the test rig of Mitsubishi Heavy Industry R&D center (Figure 1). In this closed loop, a large number of hydraulic conditions have been tested. In the present study, the condition \( \varphi = \varphi_{design} = 50 \) is selected.
**Numerical setup**

Both models (Neptune and VOF) have been used to compute hydraulic performances of the inducer for different values of $\psi_c$. We have used a finite volume RANS calculation in a rotating frame of reference with k-ε RNG turbulence model [8]. Steady state solutions were obtained for different regimes by setting the flow rate at the inducer inlet and the average static pressure at its outlet for the boundary conditions.

A multiblock structured mesh has been generated with the help of *TurboGrid* mesh generator for hydraulic machines. The mesh is made of 9 blocks and 263450 nodes for a single passage (1/3 of the machine). The computing domain is made of several O-blocks and H-blocks as well as two layers of cells in the meridian and blade to blade directions to ensure good grid orthogonality and cells evolution factor (Figure 2).

**RESULTS**

Cavitation figures corresponding to three values of $\psi_c$ are reported in Figure 3. Photographs of the flow taken through the transparent acrylic cover of the test rig are compared to predictions of both models (a value of $\alpha=0.1$ is taken for the VOF cavity interface).

The cavitation inception is observed experimentally at $\psi_c=0.6$, which corresponds to the numerical prediction. Nevertheless, while the experiment shows a simultaneous inception of cavitation in the tip clearance region and in the leading edge, both models predict cavitation inception only in the tip clearance region at $\psi_c=0.6$. The occurrence of the leading edge cavitation is predicted at $\psi_c<0.4$ by both models (Figures 3-a and 3-b)

For values of $\psi_c<0.15$, cavitation extends to the channel region and reaches the throat leading to a flow blockage (Figure 3-c). The corresponding flow visualization clearly shows strong unsteadiness of the cavity dynamics.

<table>
<thead>
<tr>
<th>Experiments</th>
<th>VOF ($\alpha=0.1$)</th>
<th>Neptune</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image1" alt="Image" /></td>
<td><img src="image2" alt="Image" /></td>
<td><img src="image3" alt="Image" /></td>
</tr>
<tr>
<td>a. $\psi_c=0.400$</td>
<td><img src="image4" alt="Image" /></td>
<td><img src="image5" alt="Image" /></td>
</tr>
<tr>
<td>b. $\psi_c=0.200$</td>
<td><img src="image6" alt="Image" /></td>
<td><img src="image7" alt="Image" /></td>
</tr>
<tr>
<td>c. $\psi_c=0.066$</td>
<td><img src="image8" alt="Image" /></td>
<td><img src="image9" alt="Image" /></td>
</tr>
</tbody>
</table>

Fig. 3 Cavitation development for different $\psi_c$ (comparison between experiments and computation)

We have plotted in Figure 4, the specific energy coefficient as a function of the cavitation number. Experimental results are compared to the predicted ones and show a good concordance concerning the threshold of performances alteration at $\psi_c \approx 0.12$. It should be noticed that at this operating condition, the suction side cavitation develops beyond the throat and an attached cavitation is observed in the pressure side.
The predicted breakdown value agrees well with experimental results when it comes to the beginning of performances drop. For lower values of cavitation numbers, the predicted drop is underestimated. For these flow regimes, the unsteady character of the flow is not taken into account by both models.

In the following sections, we will focus on VOF model to provide detailed analysis of the flow.

**BLADE-FLUID TRANSFER**

The energy supplied by the inducer is defined by:

\[
E_s = \frac{\bar{T}_s \cdot \bar{\Omega}}{\rho Q}
\]

where \(\bar{T}_s\) stands for the torque and may be derived from integration of the pressure and the viscous forces on the blades and the hub. The energy \(E\) transferred to the fluid is defined by the energy balance between inlet and outlet of the domain. We have reported in the same graph (Figure 5) the evolution of the specific energy supplied by the machine \(\psi_s\) as well as the specific energy transferred to the fluid \(\psi_t\) as a function of cavitation number \(\psi_c\). The scales are taken the same to allow qualitative analysis.

According to Figure 5, the drop in the water head at \(\psi_c \sim 0.12\) corresponds to a drop of both supplied and transferred energies. This observation let us believe that the flow field is deeply changed as the cavitation extends beyond the throat region, leading to lower value of the torque. Nevertheless, the transferred energy decreases faster than the supplied energy, which demonstrates that the hydrodynamic losses due to cavitation development play a major role in the head drop phenomenon. It should be noticed that this result is different from the one obtained by Pouffary et al. [12] in the case of a centrifugal pump, where the head drop is not due to an increase of hydrodynamic losses.

In order to eliminate the fraction of pressure, which does not contribute to the torque, we may define the following coefficients for any given location \(M\) on the blade surface:

\[
C_{p_{s,T}}(M) = \frac{\bar{U}_s}{\bar{U}} \cdot \bar{n} \cdot C_{p_s}(M)
\]

\[
C_{p_{t,T}}(M) = \left[ \frac{\bar{U}_s}{\bar{U}} \right] \cdot \bar{n} \cdot C_{p_t}(M)_\text{pressure side} + \left[ \frac{\bar{U}_s}{\bar{U}} \right] \cdot \bar{n} \cdot C_{p_t}(M)_\text{suction side}
\]

where \(\bar{n}\) is the normal vector to the blade surface at location \(M\).

We have plotted in Figure 6 the distribution of the coefficient \(C_{p_{s,T}}\) for cavitation free regime and cavitating regime \(\psi_c = 0.054\). We have also presented on Figure 7 the coefficient \(C_{p_{t,T}}\) evolution along the chord for two different values of the span (0.1 and 0.9).

As the cavitation parameter is reduced, one may easily observe a decrease of \(C_{p_{s,T}}\) coefficient in the leading edge area followed by an increase of the same coefficient at the cavity closure zone. Since no alteration of the torque is observed for \(\psi_c > 0.12\), the loss and gain in \(C_{p_{s,T}}\) coefficient are almost balanced. As soon as the cavity reaches the throat, the resulting drop of the torque illustrates that the loss in \(C_{p_{s,T}}\) coefficient is no more balanced by the gain at the cavity closure. Furthermore, one may easily observe in Figure 7 that the occurrence of leading edge cavitation on the pressure side is due to a significant decrease of pressure coefficient in this area. This illustrates once again the modification of the flow field in the channel as soon as the main cavitation reaches the throat area. The cavitation in pressure side could be the reason of the torque loss.
Several observations may be made through the evolution of the energy and energy transfer in the machine due to the cavitation development:

- **Sections 1-5 (upstream to the blade leading edge):** Obviously, since no energy is yet transferred to the fluid, total energy in sections 1 through 5 remains almost constant. Furthermore, no cavitation effect on energy evolution could be observed in this area.

- **Section 6 (throat, cavity closure):** The leading edge cavitation causes a decrease of the potential energy and an increase of the kinetic energy. This may be explained by the fact that the flow is accelerated in the non-cavitating region to ensure the mass conservation.

- **ψ<sub>c1</sub>:** slight increase of the total energy
- **ψ<sub>c2</sub>:** significant decrease of the total energy

- **Sections 6-8 (first part of the channel):** The excess of the kinetic energy due to cavitation at section N°6 is totally dissipated at section N°8, but partially transformed to a potential energy, since the potential and total energy losses are reduced but not totally recovered.

- **Sections 8-10 (second part of the channel):** The lack of total energy observed in the mid-channel (section N°8) is no more recovered in the rest of the channel.

- **Sections 11-15 (blade wake, inducer outlet):** In this area, the kinetic energy is drastically reduced while the potential energy remains constant. Furthermore, the cavitation has no significant effect on energy distribution and the lack of energy observed in previous sections is almost maintained.
where $I_{\text{ref}}$ denotes a reference rothalpy averaged over the section N°1.

We have presented in Figure 10 the evolution of the energy losses in the meridian channel for two cavitating conditions ($\psi_c=0.077$ and $\psi_c=0.054$). We may observe again that energy losses increase when cavitation number is lowered. Between sections 1 and 5, no significant change in energy losses is observed illustrating that the flow upstream the blades is not affected by the cavitation. The negative value of relative losses obtained in section N°6 for both cavitation numbers shows a better energy transfer between the blade and the fluid in the space between the leading edge and the channel inlet. On the contrary, from the channel inlet up to the trailing edge (sections 6 to 10), the flow exhibits an increasing energy loss due to cavitation. No further relative losses are observed in up to the inducer outlet.

**Fig. 10** Evolution of losses due to the cavitation in the meridian channel

### LOCAL FLOW ANALYSIS

We have presented in Figure 12 local values of relative losses in a blade to blade view for two different span values. The cavitation number is set to $\psi_c=0.054$ corresponding to a leading cavity, which extends up to the channel inlet.

We have presented in Figure 11 the distribution of the relative normal speed, relative losses and relative kinetic energy in two plans located near the throat and the mid channel. We have superposed to all these graphs the distribution of net tangential velocity vectors due to cavitation. It should be noticed that the plans are constructed normal to the main flow direction (we use $U_G$ as normal to the plan).

The distribution of relative energy losses shows clearly that maximum losses are mainly concentrated in the vicinity of the cavity closure near the shroud as well as above the cavity interface prior to channel inlet.

The normal velocity distribution in the throat clearly shows that the leading edge cavity causes a deep change in the flow repartition. A significant decrease of the normal velocity in the cavity wake is balanced by an increase of the normal
velocity in the hub region. At mid channel, although the flow imbalance is mitigated, the flow does not recover completely. The superposed vector field of the tangential velocity illustrates the secondary flow reorganization to achieve the flow recovery. As already reported in the Meridian Analysis, the inducer fails in recovering the kinetic energy up to its outlet.

The distribution of energy losses is confirmed by the observations made on figure 12. The cavitation causes a substantial increase of the energy losses in its wake. One may easily observe that in this area, an increase of kinetic energy is well visible in Figure 11. Obviously, this kinetic energy is mainly related to secondary flow motion.

CONCLUSION

In the present study, two CFD methods for cavitation modeling have been investigated in the case of 3-bladed industrial inducer. The first model is an interface tracking method and the second is a VOF multiphase model. Computation analysis and experimental comparison allowing the validation of the models have been done.

The main conclusions may summarize as follows:

- Both models allow a good prediction of the cavitation inception as well as the main cavity dimensions. The threshold corresponding to the head drop is also well predicted by both models. Nevertheless, we have focused on the VOF model to further analyse the flow field since it is closer to the physics of the cavitation phenomenon.

- The cavitation induced head drop is mainly due to an increase of energy losses. A secondary reason of the head phenomenon is a decrease of the supplied energy resulting from a torque reduction.

- A meridian analysis of the flow energy shows that the kinetic energy is increased at the channel inlet while the potential energy is decrease. Part of the gained kinetic energy is then transformed into potential energy in the channel while the other part is definitively lost. Furthermore, the rothalpy evolution analysis clearly demonstrates that the main energy losses are located in the channel downstream to the cavity closure.

- The hydrodynamic mechanism of head drop is explained through a global and local analysis of the flow field. As soon as the leading edge cavitation reaches the throat, it causes a flow blockage. The lack of the velocity in the flow direction in the cavity wake is balanced by an increase of the velocity in the hub region. Although this flow imbalance is mitigated further in the channel, the flow does not recover completely. The pressure distribution in the pressure side is significantly reduced at
the leading edge, which allows cavitation occurrence and thereby influences the energy transfer.

Although this analysis provides an explanation of the hydrodynamic origin of the head drop; it should be noticed that absolute value of head drop is not well predicted. In fact, the actual solution of the flow is steady-state and the flow rate is fixed in the computation which is far from the realistic pumping systems where the cavitation may induce significant flow unsteadiness and large fluctuation of the cavity length as well as the flow rate.

ACKNOWLEDGMENTS

Authors would like to thank their colleagues of EPFL and Mitsubishi Heavy Industry R&D centre for their valuable help. They are also grateful to the Swiss Science Foundation as well as Mitsubishi Heavy industry R&D centre for their financial support.

REFERENCES


