

High head pump-turbine: Pumping mode numerical simulations with a cavitation model for off-design conditions

U Jese¹, R Fortes-Patella¹ and S Antheaume²

¹ Univ. Grenoble Alpes, LEGI, F-38000 Grenoble, France

² Hydraulic Department Alstom Global Technology Center, Alstom, Grenoble, France

E-mail: uros.jese@legi.grenoble-inp.fr

Abstract. Flexibility and energy storage are one of the main challenges of the energy industry at the present time. Pumped Storage Power Plants (PSP), using reversible pump-turbines, are among the most cost-efficient solutions to answer these needs. To provide a rapid adjustment to the electricity grid, pump-turbines are subject of quick switching between pumping and generating modes and to extended operation under off-design conditions. In particular, at part load, instabilities in pump characteristics can occur. It can lead to unsteadiness and even to a shift of the operating point with significant modification of discharge and drop of efficiency. This unstable area is often exposed to the cavitation phenomenon, which can lead to vibrations, loss of performance and sometimes erosion. The paper focuses on the numerical analysis of the pumping mode regime, especially on the part load off-design instabilities, observed as a saddle shaped pump-turbine head curve and the presence and development of the cavitation in the part load area. The investigations were made on the reduce-scaled model of a high head pump-turbine design. Numerical calculations were performed using commercial code with implemented barotropic cavitation model. Some of the numerical results were compared to the experimental data. Flow analysis was stressed on the cavitation influence on the flow behavior and the performance of the machine. The analysis was made for various flow rates and a wide range of NPSH values. The importance of specific parts of the numerical domain for obtained results was investigated and evaluated.

1. Introduction

Growing environmental awareness leads to massive energy production increase from variable renewable sources such as wind and solar. However, the development of intermittent renewable energies power plant makes energy production highly dependent on meteorological conditions. Moreover, the daily energy consumption varies significantly. Therefore, the pumped storage power plants with installed reversible pump-turbines are reliable systems to ensure energy storage and enable flexible and rapid adjustment of whole electrical grid. Therefore, the pump turbine switches several times a day between pumping and generating mode, which means passing through off-design regime. Unstable areas of the unit characteristic performance curve are not welcome and have to be better understood since they may cause fatigue stress on the runner.

The main features of unstable behavior areas can be divided to generating and pumping modes. On one hand, the S-shaped turbine characteristic in generating mode occurs at low load off-design operating points close to runaway conditions. The phenomenon has been investigated numerically as well as experimentally by several authors [(1), (2), (3), (4), (5)], so far the most extensive study on the generating mode instabilities was made by (6). The study took place in Lausanne within the cooperation project HYDRODYNA between two universities and industrial partners including ALSTOM.



On the other hand, the issue of pump-turbines operation under off-design conditions in pumping mode appears as a head drop as the flow is reduced, the shape of curve is so called saddle-typed or humped. The phenomenon was investigated by Braun (7) during the mentioned HYDRODYNA project. Basic description of instability criterion for a turbomachinery was given by Gentner (1).

When discussing instabilities, unsteady operating points and off-design regimes, one cannot overlook the phenomenon of cavitation and its important influence on the design, the performance and the maintenance of the pump-turbines and turbomachinery in general. In the present work, the numerical investigations in the pumping mode under off-design conditions are performed by using commercial code with implemented barotropic cavitation model. Some of the numerical results were compared to experimental data. Moreover, the additional flow analysis at various flow rates and NPSH values are investigated and analyzed.

2. Hydraulic characteristics and instabilities

Switching between pumping and generating mode several times daily causes large operating range for reversible pump-turbine, including highly off-design operating points. Typical four quadrant characteristic chart (figure 1) represents discharge factor $Q11$ ($Q11 = Q / D^2 \cdot H^{0.5} - m^3/s$) dependence on speed factor $n11$ ($n11 = n \cdot D / H^{0.5} - rpm$) for three different guide vanes openings. The efficiency as well as reliance of pump-turbines depends on both, pumping and generating mode. Moreover, off-design operating areas, like start-up, play key role in designing and optimizing the reversible machine.

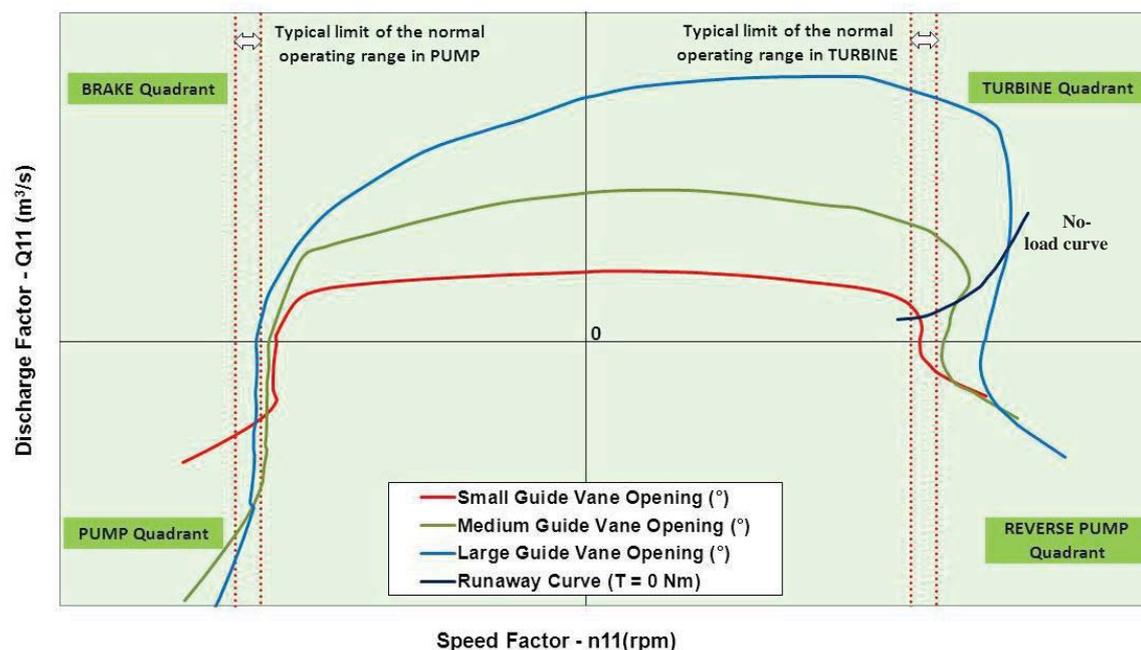


Figure 1. Typical four quadrant characteristics of a high head reversible Pump-Turbine (3)

Problematic S-shaped area close to runaway curve is clearly seen in the generating mode for all three different guide vane openings at the figure 1; meanwhile, in the pumping mode the S-shape is not completely obvious. Another possibility to present and describe the system instabilities is net head characteristic curve. The instabilities are especially related to the slopes of the curves (figure 2). As

explained by (3), (1) and (8), the slope gives a necessary but not sufficient condition for the instability of the system. Additional conditions are elasticity, inertia and the amount of energy dissipated in the system. Once the transferred energy is larger than the dissipated one, the system becomes self-excited.

At pumping off-design operating mode the criterion for instability is defined by: $\frac{dH}{dQ} > 0$;

On contrary for generating mode the instability definition can be: $\frac{dH}{dQ} < 0$.

Negative slope in the generating mode means that energy is transferred to the fluid and can cause instabilities in the system. On the other side, positive slope in the pumping mode means that energy, transferred to the fluid does not contribute to the head rise, but dissipates and potentially causes rise of unsteady flow patterns such as separated flow, recirculation zones and vortices.

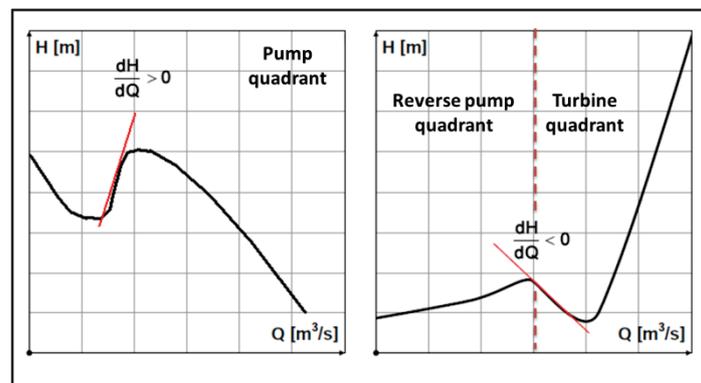


Figure 2 : Saddle type head curve characteristic in pumping and generating mode for constant guide vane opening (8)

3. Numerical setup

As mentioned, all CFD simulations were carried out in LEGI, using commercial code. The applied code solves 3D Reynolds-averaged Navier-Stokes (RANS) equations of a homogeneous fluid with a cavitation model [(9), (10)]. In the present study, the two-equational k- ϵ model with extended wall functions was used to describe turbulent behavior (13). All calculations were made for steady flow conditions.

The barotropic state law, initially proposed by Delannoy and Kueny (14), was used for mass transfer modeling in cavitating flows. The liquid-vapor mixture is defined as a mixture density ρ given as a function of a void ratio α , by the relation $\rho = \alpha\rho_v + (1 - \alpha)\rho_l$, where ρ in the flow field varies between the vapor and the liquid density as presented on the figure 3. The barotropic state law depends on the minimum speed of sound in the mixture C_{\min} , which is in our case taken as 1.0 m/s for cold water and on the density ratio value ρ_v / ρ_l , which was chosen to be 0.01. The details regarding numerical models are available at (11), (12).

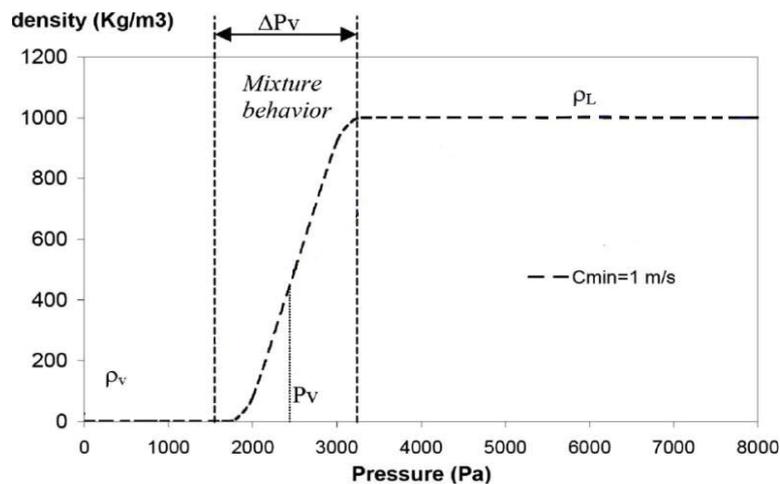


Figure 3. Barotropic state law $\rho=\rho(P)$, influence of speed of sound for water at 20°C (9)

The pump turbine under study is a design of low specific speed with 9 blades and 20 guide vanes. The simulations have been carried out for a single guide vane opening.

The computational domain corresponds to a periodical reduced-scale pump-turbine model which extends from the draft tube cone to the outlet of the stay vanes (figure 4). At the outlet of the domain, an extension with diffuser was used in order to reduce or eliminate potential back flow at high pressure side outlet, which can cause some convergence problems during the calculations (7). Some calculations on a similar geometry, but in generating mode, were performed by Zobeiri (4), who concluded, that for the rotor-stator instabilities, including the stay vanes into domain is essential for simulations and the spiral case is not.

Meshing is an important part of each CFD simulation. A commercial meshing software was used to create the mesh, which consist of around 900.000 cells and satisfies the Y^+ criteria $Y^+ < 50$ for applied $k-\epsilon$ (Extended wall function) turbulent model (figure 4). The other important mesh parameters are minimum skewness 27° and maximum expansion ratio 1.8.

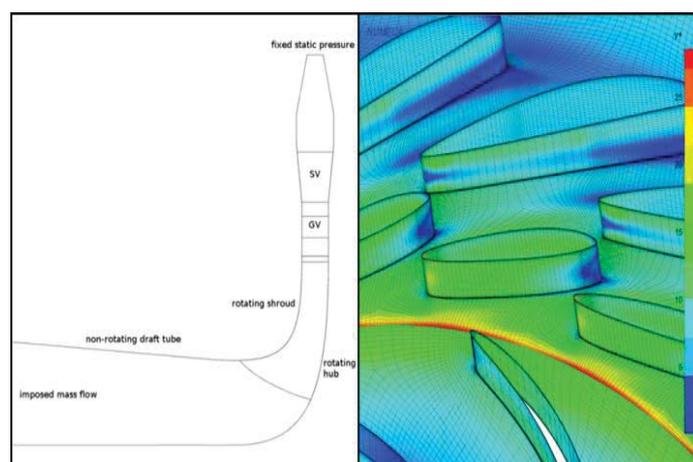


Figure 4. Boundary conditions, zoom of Y^+ value on distributor mesh,

Mass flow is imposed at the inlet as a boundary condition and static pressure is a boundary condition at the outlet (figure 4). The option of a periodicity was applied between the blades (figure 4). The periodicity

option reduces the number of mesh cells and allows calculating wide range of various operating points. On contrary, non-symmetrical instabilities, such as flow separation in the distributor in the form of rotating stall and some forms of cavitation cannot be detected correctly for several operating points due to periodicity condition between the blades. A non-cavitating solution was always used as an initial solution to start a calculation, which includes cavitating model.

Mesh density plays important role in providing a reliable results, especially when calculating the pump-turbine off-design operating points, such as a hump area and presence of the cavitation.

However, when studying cavitation phenomena on the macro scale (at turbomachinery), using coarse mesh can be an advantage. Coarse mesh enables calculating large cavitation forms, which influence the performance of the pump (changing velocity triangles (figure 10), reducing the active channel area between the blades, causing a head drop for the low NPSH values on the performance curve(10), (15)...). On the other hand, we do not calculate smaller, unsteady cavitation forms, such as separate bubbles and smaller cavitation clouds. Due to their unsteadiness, they could have negative influence on the steady simulation convergence criteria, consequently, on the quality of the calculated results. In our case, the best compromise seems to be a fine mesh in the distributor and a coarser mesh at the inlet of the impeller.

4. Global results

The comparison presented in the figure 5 can be analyzed qualitatively. A source of difference between experimental and numerical net head values is due to different total pressure measurement spots between the experimental and the numerical domain. The spiral case and draft tube are indeed not represented in the calculations. As shown on the figure 5, the position of the hump zone can be detected numerically by the accuracy of 2% on flow rate value; also the hump amplitude matches quite well, in both cases reach around 0.5m. The chart serves as a qualitative representation of a numerical accuracy and the results were obtained for non-cavitating regime.

However, it is worth mentioning the hysteresis effect. During several similar experimental measurements (16) it was shown, that the position of the hump area can vary, depending on the procedure of increasing and decreasing the flow rate during the measurements. The possibility, that hysteresis effect can influence the experimental data, consequently, the hump area, must be taken into account.

The qualitative comparison between the experimental and the numerical performance curves shows good match for all operating points at normal operating regime as well as around area, where hump occurs ($Q/Q_{opt}=0.76$ and higher). Q_{opt} represents the discharge of the best efficiency point (BEP) for considered guide vane opening. Meanwhile, $loss_{OUT}$ represents the numerically calculated losses in the distributor. The performance decrease, known as hump, occurs due to flow separation in the distributor and vaneless gap area, which increases the distributor losses and decreases the total head in the region lower than $Q/Q_{opt}=0.87$ (figure 5). At the beginning, the flow separation is present more or less symmetrically around the distributor. However, with additional lowering of the flow rate (lower than $Q/Q_{opt}=0.76$), the phenomenon is not anymore symmetrical as it was detailed described by Braun (7).

In our case, it means, that for flow rates lower than $Q/Q_{opt}=0.76$, the periodicity as boundary condition, might not be sufficient to simulate the flow behavior and distributor head loss. The head losses seems to be too low around $Q/Q_{opt}=0.7$ and too high around $Q/Q_{opt}=0.6$ due to different, most likely not accurate, forms of separated flow inside the distributor.

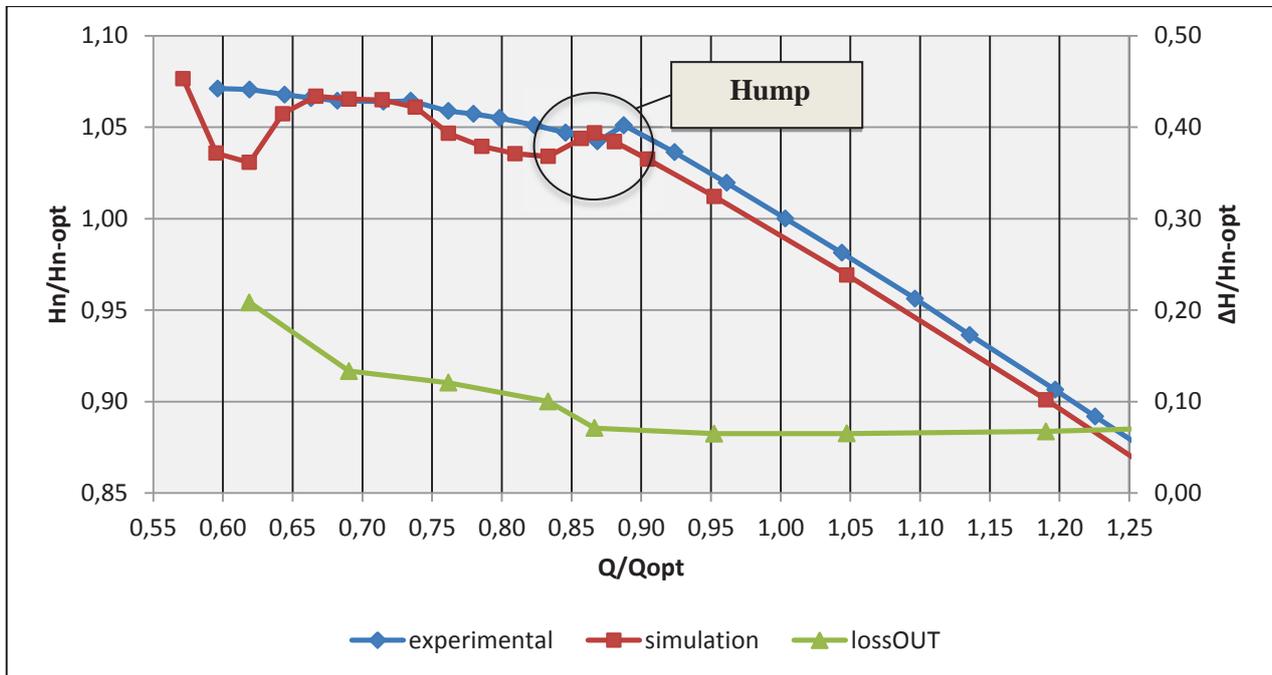


Figure 5. Pump mode head versus discharge characteristics

This part demonstrates the ability of the code to predict qualitatively the position and amplitude of the hump zone. However, the main contribution of this paper is the prediction of cavitation forms which will therefore be the focus of the next sections.

5. Cavitation behavior

To evaluate the level of the cavitation in the pumping regime, it is essential to define the NPSH (Net positive suction head) value. For each operating point it is calculated as

$$NPSH = (P_{REF} - P_v) / (\rho_l g),$$

where P_{REF} represents the total pressure value at the inlet of the domain, P_v vaporization pressure of the water, ρ_l density of the water and g the gravitational constant. According to the mentioned boundary conditions, the NPSH value was calculated after each calculation was finished. The NPSH value was modified by changing the value of vaporization pressure in order to keep the stability of simulations and similar level of the convergence, which plays even more important role on the calculations, when the cavitation model is included. (The described action has no physical background, just numerical one). Similar approach was used and described in the thesis work of Pouffary (12) and Coutier-Delgosha (11).

5.1 Validation of the cavitating model

It is important that the numerical simulations in the cavitating regime can be compared to the experimental results. The incipient cavitation value (figure 6) is presented for the numerical simulations as well as for the experiment. Experimentally, the value is obtained by visual observation of the pump-turbine impeller inlet. The value at different flow rates is defined as an appearance of the first cavitation bubbles. Numerically, the criteria for the incipient cavitation condition were defined by void ratio α . When void ratio in first cavitating cell reached the value $\alpha=0.2$ (20% of vapor in the cell), the corresponding NPSH value was taken as an incipient cavitation value.

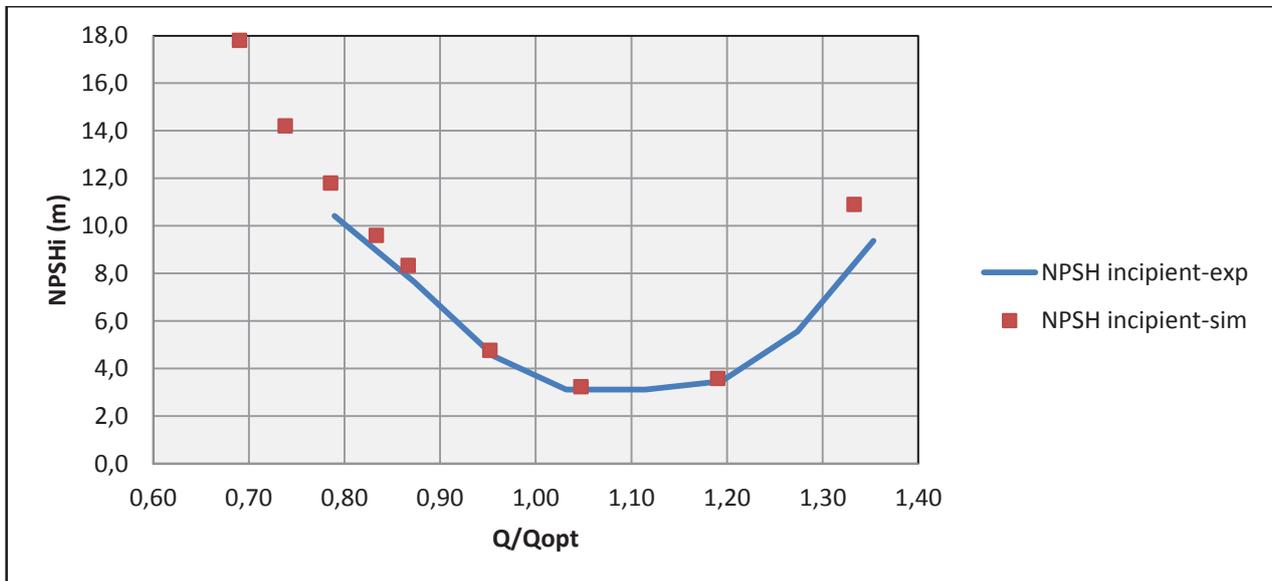


Figure 6. Incipient cavitation value

As presented on figure 6, the matching is good for the wide range of flow rates. The maximum difference between the experimental and the numerical values is around ± 0.5 m of NPSH. However, there is some deviation for over flow rate operating point at $Q/Q_{opt}=1.33$. One reason might be the experimental method for estimating the occurrence of the first bubble. In most of our studied cases, the first cavitation bubble occurs at the leading on the suction side of the impeller blade. On contrary, at $Q/Q_{opt}=1.33$, the cavitation firstly occurs on the pressure side of blade, which is visually more demanding to observe which could explain the higher discrepancy in this area.

5.2 Cavitating flow analysis

Robustness of cavitation model enables us to provide results for a wide range of flow rates and a wide range of NPSH values. Moreover, the NPSH head drop curves can be made for various flow conditions. Figure 7 represents three NPSH head drop curves. The one, close to best efficiency point for the guide vane opening of 14° , the one at over flow rate and another at reduced flow rate.

As seen from figure 7, head can drop up to 10%, because of blade cavitation. In case of reduced flow rate, the performance increase can be observed before head drop at around $NPSH=2$. The cavitation sheet at that point improves the beta angle at the leading edge of the blade. It is clear that cavitation cloud at the low NPSH values is not a steady phenomenon. However, steady simulations enable us to get average size of cavitation sheet for each NPSH value. The different flow rate values can as well be compared between each other and to the experimental results for considerable cases.

The markers on the figure 7 represent experimental values for 1% head drop due to cavitation. As seen, the matching between the numerical curves and experimental data is quite good.

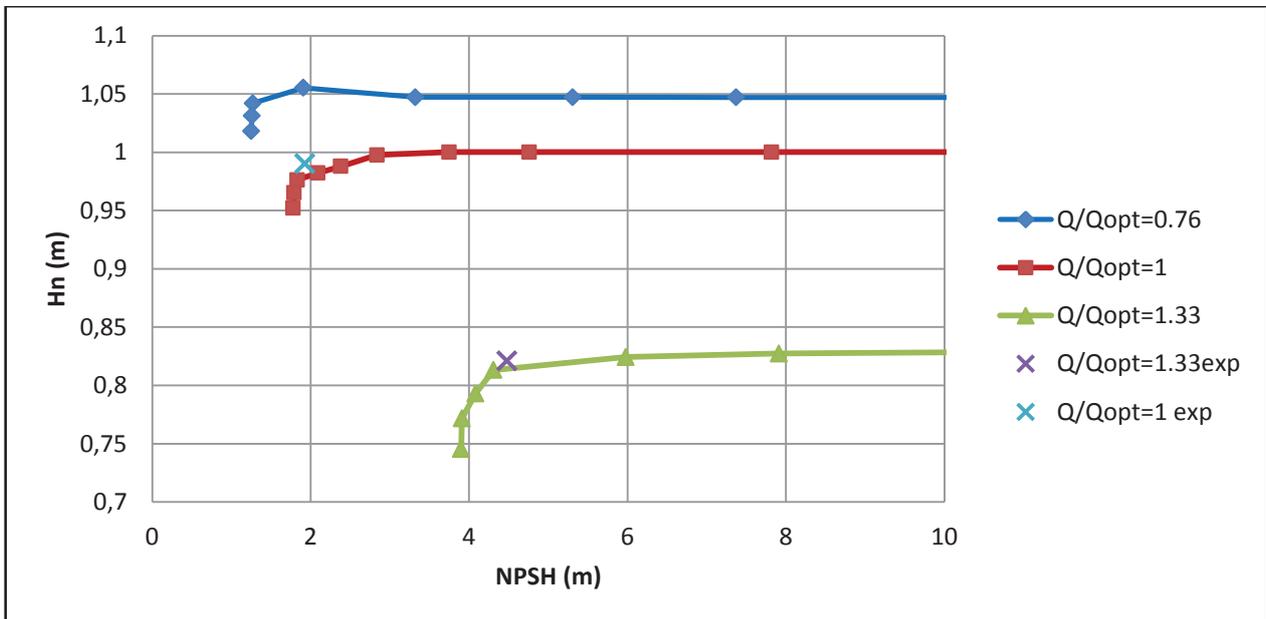


Figure 7. NPSH head drop chart

Related to figure 7, we can observe the cavitation forms for different flow rates. Figure 8 represents four different levels of cavitation inside the pump-turbine close to $Q/Q_{opt}=1$, best efficiency for the considered opening flow rate. Incipient cavitation value is around $NPSH = 4.8$ m. The cavitation first occurs on the suction side of the blade. With lowering the NPSH value, the cavitation is seen mostly on the large area on the suction side of the blade, closer to the shroud. At extremely low NPSH ($NPSH=1.78$ m), we can observe the vapor also on the pressure side. In case of $NPSH=1.78$ m the head loss reaches almost 5% of nominal head.

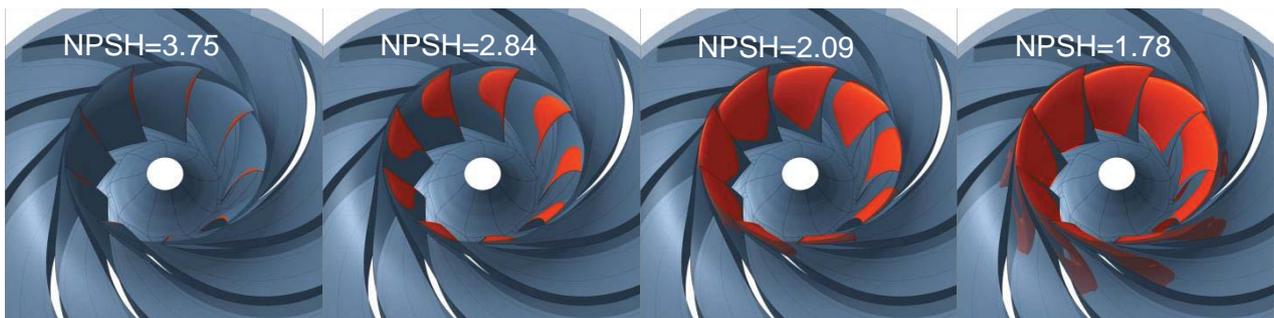


Figure 8. Cavitation forms at BEP for different NPSH values

Figure 9 shows the cavitation forms at flow rate lower than BEP. The NPSH values for similar cavitation intensity are much higher at $Q/Q_{opt}=0.76$ than at BEP and the forms changes a bit. In both cases, the cavitation occurs at the suction side of the blade and grows to the level that it blocks the significant part of the channel between the blades.

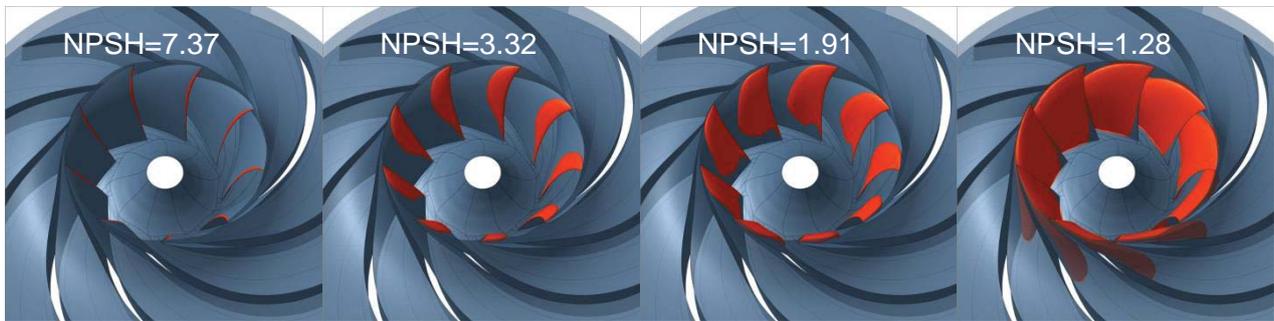


Figure 9. Cavitation forms at $Q/Q_{opt}=0.76$ for different NPSH values

Especially interesting is to observe the cavitation sheet at the NPSH value close to 2. The cavitation form increases the performance of the pump-turbine (figure 7) by fictive changing the inlet beta angle of the blade (figure 10).

To understand the position of the cavitation phenomena on the blades it is essential to check the velocity triangles at the inlet of the pump-turbine (figure 10). For BEP the beta angle close to the leading edge matches the angle of the blade. At reduced flow, the angle decreases due to lower meridional velocity and as a result, we can observe the cavitation at the suction side at higher NPSH values. For the increased flow the beta angle is higher than in case of BEP and also higher than blade angle, therefore the cavitation first occurs at the pressure side of the blades.

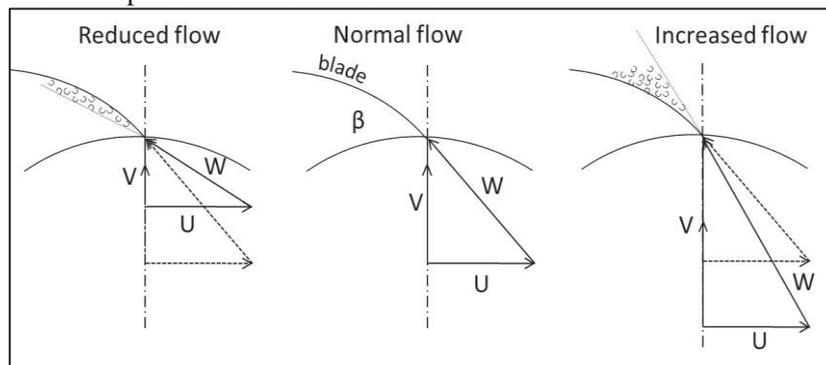


Figure 10. Velocity triangles at the inlet for different flow rates

Figure 11 and 12 represent the cavitation forms for flow rate higher than BEP for a different NPSH values. The first cavitation appears close to the leading edge on the pressure side of the impeller blade (figure 12). With lowering the NPSH value, the cavitation sheet grows mostly on the shroud side of the impeller on both, pressure and suction side of the blade. Around the NPSH=4 the sheets from pressure and suction side of the blade join on the shroud side of the impeller. In the meantime, the cavitation on the guide vanes keeps the form and does not change significantly for lower NPSH.

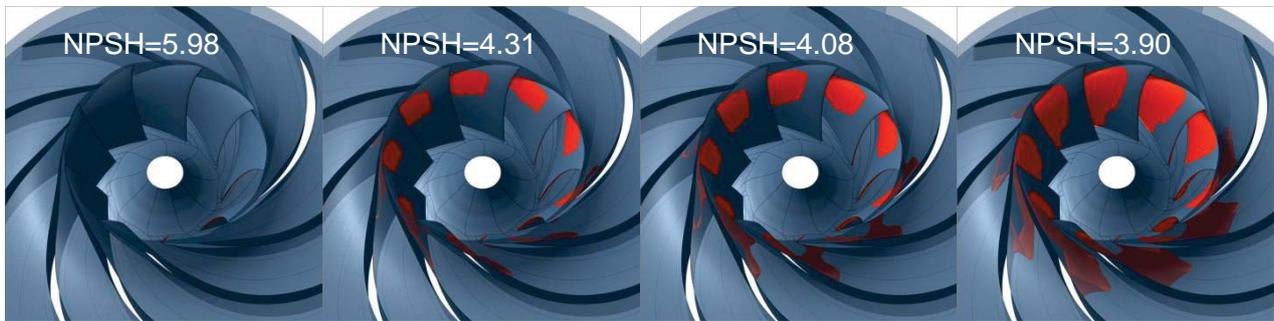


Figure 11. Cavitation forms at $Q/Q_{opt}=1.33$ for different NPSH values - suction side view

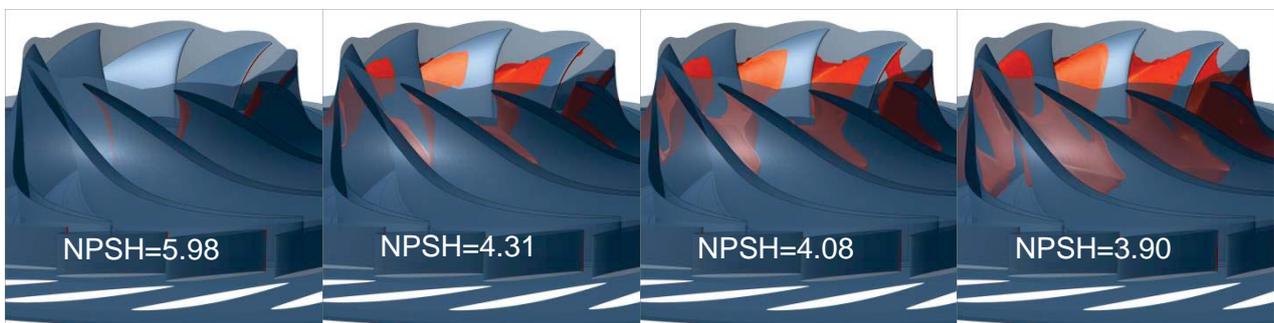


Figure 12. Cavitation forms at $Q/Q_{opt}=1.33$ for different NPSH values - pressure side view

The direct comparison of the cavitation forms between the numerical images and the experimental visualization was possible for the flow rate $Q/Q_{opt}=0.83$. As seen from figure 13, the cavitation structures match the forms, observed by visualization of the suction side of the pump-turbine impeller.



Figure 13. Comparison between numerical and experimental cavitation forms

6. Perspectives and conclusions

The goal of the described studies was to develop the numerical simulations for prediction the operating points around the hump area and to study behavior of the pump-turbine at the presence of the cavitation phenomenon. The prediction of the hump is relatively good, however, for flow rates lower of the $Q/Q_{opt}=0.76$, it would be necessary to use full mesh without periodicity condition.

The cavitation code is robust enough to produce the results at very low NPSH values with satisfied convergence criteria. At some flow rates, the pressure drop can reach up to 10%, which is enough to simulate the effects of the majority off-design pump-turbine operating points. Knowing well the cavitation behavior is essential to design an efficient and reliable pump-turbine impeller. For low NPSH values, where the cavitation is very intensive, steady simulations do not describe completely the cavitation forms.

However, the average cavitation cloud gives us information about global effect of a cavitation on the pump-turbine performance. To produce more accurate results about cavitation forms, unsteady simulations are essential.

There are several possibilities for additional investigations of the hump behavior, including a detailed local flow analysis. Possible impact of the cavitation on the hump zone is going to be investigated. Numerical calculations will be made for other guide vane openings and different guide vane geometries in order to better understand the phenomenon.

7. Acknowledgements

The project is a result of historical cooperation between the LEGI laboratory and the industrial partner Alstom Hydro. The industrial partner has contributed experimental results of pump-turbine model measurements and necessary numerical geometry to examine the effects on the pump-turbine performance. The laboratory LEGI is part of the LabEx Tec 21 (Investissements d'Avenir – grand agreement n°ANR-11-LABX-0030). PhD study is financed by Slovene Human Resources Development and Scholarship Fund.

8. References

- [1] Gentner Ch *et al* 2012 Numerical and experimental analysis of instability phenomena in pump-turbines *26th IAHR Symposium (Beijing)*
- [2] Hasmatuchi V *et al* 2010 High speed flow visualisation in a pump-turbine under off-design operating conditions *25th IAHR Symposium (Timisoara)*
- [3] Houdeline J-B *et al* 2012 Start-up improvement in turbine mode for high head PSP machine *26th IAHR Symposium (Beijing)*
- [4] Zobeiri A *et al* 2006 Pump-turbine rotor-stator interaction in generating mode: pressure fluctuation in distributor channel *23th IAHR Symposium (Yokohama)*
- [5] Liu W C *et al* 2012 Hydraulic optimization of S characteristics of the pump-turbine for Xianju pumped storage plant *26th IAHR Symposium (Beijing)*
- [6] Hasmatuchi V 2012 Hydrodynamics of a pump-turbine operating at off-design conditions in generating mode *PhD Thesis (EPFL, Lausanne)*
- [7] Braun O 2009 Part load flow in radial centrifugal pumps *PhD Thesis (EPFL, Lausanne)*
- [8] Staubli C-T *et al* 2010 Starting pump-turbines with unstable characteristics *Hydro (Lisbon)*
- [9] Pouffary B *et al* 2008 Numerical simulations of 3D cavitating flows: analysis of cavitation head drop in turbomachinery *Journal of Fluids Engineering* **130** 061301
- [10] Coutier-Delgosha O *et al* 2003 Experimental and numerical studies in a centrifugal pump with two-dimensional curved blades in cavitating condition *Journal of Fluids Engineering* **125** pp. 970-978.
- [11] Coutier-Delgosha O 2001 Modelisation des écoulements cavitants: etude des comportements instationnaires et application tridimensionnelle aux turbo-machines
- [12] Pouffary B 2004 Simulation numérique d'écoulements 2D/3D cavitants, stationnaires et instationnaires: analyse spécifique pour les turbomachines *PhD Thesis (INPG, Grenoble)*
- [13] Hakimi N *et al* 2000 Presentation and application of a new extended k- ϵ model with wall functions (*ECCOMAS Barcelona*)
- [14] Delannoy Y and Kueny J-L 1990 Two phase flow approach in unsteady cavitation modeling *Cavitation and Multiphase Flow Forum* pp. 153-158.
- [15] Gonzalo-Flores N *et al* 2008 Head drop of a spatial turbopump inducer *Journal of Fluids Engineering* **130** 111301.
- [16] Kaupert K-A *et al* 1997 First analysis of flow field hysteresis in a pump impeller *Journal of Fluids Engineering*