

Numerical investigation of tip clearance cavitation in Kaplan runners

K. Nikiforova¹, G. Semenov², I. Kuznetsov³, E. Spiridonov⁴

¹ Research fellow, Polytechnical State University, St.Petersburg, Russia

² Leading design-engineer, PJSC "Power Machines», LMZ, St.Petersburg, Russia

Corresponding author: Semenov_GA@lmz.power-m.ru

³ Head of department, PJSC "Power Machines", LMZ, St.Petersburg, Russia

⁴ Head of sector, PJSC "Power Machines", LMZ, St.Petersburg, Russia

Abstract. There is a gap between the Kaplan runner blade and the shroud that makes for a special kind of cavitation: cavitation in the tip leakage flow. Two types of cavitation caused by the presence of clearance gap are known: tip vortex cavitation that appears at the core of the rolled up vortex on the blade suction side and tip clearance cavitation that appears precisely in the gap between the blade tip edge and the shroud. In the context of this work numerical investigation of the model Kaplan runner has been performed taking into account variable tip clearance for several cavitation regimes. The focus is put on investigation of structure and origination of mechanism of cavitation in the tip leakage flow. Calculations have been performed with the help of 3-D unsteady numerical model for two-phase medium. Modeling of turbulent flow in this work has been carried out using full equations of Navier-Stokes averaged by Reynolds with correction for streamline curvature and system rotation. For description of this medium (liquid-vapor) simplification of Euler approach is used; it is based on the model of interpenetrating continuums, within the bounds of this two-phase medium considered as a quasi-homogeneous mixture with the common velocity field and continuous distribution of density for both phases. As a result, engineering techniques for calculation of cavitation conditioned by existence of tip clearance in model turbine runner have been developed. The detailed visualization of the flow was carried out and vortex structure on the suction side of the blade was reproduced. The range of frequency with maximum value of pulsation was assigned and maximum energy frequency was defined; it is based on spectral analysis of the obtained data. Comparison between numerical computation results and experimental data has been also performed. The location of cavitation zone has a good agreement with experiment for all analyzed regimes.

KEYWORDS: 3-D CFD, Tip cavitation, Kaplan runner, Pulsation pressure, Vortex structure

1. INTRODUCTION

Hydraulic turbines operation in many operating conditions is accompanied by cavitation phenomena. Cavitation originated in closed water passages at rather high flow velocities, breaks normal operating process, and, consequently, leads to power and efficiency reduction. In addition, cavitation destroys operational components of machines reducing their service life. There are several types of cavitation in hydraulic machines: surface profile cavitation that occurs during stream over-flow of the blades, local cavitation induced by over-flow of the surface irregularity and tip clearance cavitation caused by the presence of clearance gap between the Kaplan runner blade and the shroud. Leaking through the gap the water rolls-up



into the tip clearance vortices on the suction side of the blade. The pressure in the vortex core in most cases falls below the vaporization pressure leading to cavitation. In the paper [1] numerical investigation of the flow in model Kaplan runner's gap has been performed using RANS model excluding cavitation. It was found that losses caused by existing tip clearance reduce efficiency of turbine by about 0.5%. Thus, at the stage of turbine design the problem of adequate simulation of cavitation flow including tip clearance, and analysis of influence of cavitation on turbine performance is crucial.

2. BACKGROUND

Current approach for calculation of turbulent cavitation flows is generally based on the solution of Reynolds equations supplemented by cavitation model. Besides, calculations using LES are being widely applied. Calculation of unsteady flows with cavitation performed by different authors has shown that using LES and hybrid RANS/LES approaches with application of sufficiently accurate meshes (resolving most of vortex structures) allows to correctly model cavitation flows. And it leads to significant increasing of computational costs. Using of eddy-viscosity URANS models allows to reduce computational time and correctly predicts main frequencies of periodic process.

The influence of gap size on flow structure and loss in the runner were investigated by the authors of the paper [3]. The results were compared with the experimental data obtained by LDV. Qualitative assessment of cavitation zone was given by defining an area of low pressure caused by the formation of vortex on suction side.

In paper [4] various details of modeling the flow through the gap have been discussed: setting of boundary conditions, mesh specification, turbulent model choice, scale-up effect. The model problem of flow profile in the channel with the moving walls taking into account tip clearance is solved. It is shown that the walls motion has a huge impact on the location and intensity of tip vortex. Considerable pressure change in the vortex core with mesh refinement is also observed. The more detailed mesh, the lower pressure in the core is. As for the choice of turbulent model, unlike RSM-model and zonal RANS/LES approach, two-parameter eddy-viscosity models are too dissipative and significantly underestimate pressure in the center of the vortex. This effect is caused by strong impact of rotation on turbulence characteristics. To account for this effect curvature correction option has been included.

Due to lack of public information about investigation of tip clearance cavitation in the runner a necessity surged to research this problem and compare the obtained results with the experimental ones.

3. NUMERICAL MODEL

In the context of this work numerical investigation of the model Kaplan runner has been performed taking into account variable tip clearance. The prototype of the analyzed model runner is installed at Djerdap HPP in Serbia. There are 32 guide vanes in distributor and the runner with diameter 0.35 m contains 6 blades set at an angle $\varphi = 5^\circ$. Two cavitation regimes are considered in this work: Cavitation regime №1, respective cavitation number $\sigma_{Th} = 0.4868$, and Cavitation regime №2 with $\sigma_{Th} = 0.3645$. Due to solving the problem in the cyclic setting it became possible to use the computational domain that includes water passage with only one blade. To define the inlet boundary conditions, a preliminary joint calculation of distributor and runner with the help Mixing Plane model has been performed. As an outlet boundary condition, the fixed static pressure corresponding to the analyzed cavitation regime

is set. The part of the computational domain including runner blade rotates in absolute coordinate system at constant angular velocity ω . Adhesion to the walls is being considered. Calculations have been performed with the help of 3-D unsteady numerical model for two-phase medium. Modeling of turbulent flow has been carried out using full equations of Navier-Stokes averaged by Reynolds. For description of this medium (liquid-vapor) simplification of Euler approach is used. It is based on the model of interpenetrating continuums, within the bounds of this two-phase medium considered as a quasi-homogeneous mixture with the common velocity field and continuous distribution of density for both phases. Equations describing dynamic of this mixture have been written in absolute coordinate system rotating at constant angular velocity $\vec{\omega}$. The system of governing equations includes mixture mass transfer equation (1), mixture momentum transfer equation (2), vapor mass transfer equation (3).

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{V}) = 0 \quad (1)$$

$$\frac{\partial(\rho \vec{V})}{\partial t} + \nabla \cdot (\rho \vec{V}_r \vec{V}) + \rho(\vec{\omega} \times \vec{V}) = -\nabla p + \nabla \cdot \bar{\tau} \quad (2)$$

$$\frac{\partial(\alpha_v \rho_v)}{\partial t} + \nabla \cdot (\alpha_v \rho_v \vec{V}) = S_v - S_l \quad (3)$$

S_v , S_l - source terms corresponding to vapor generation and condensation; $\rho = \rho_l \alpha_l + \rho_v \alpha_v$ - mixture density; $\mu = \mu_l \alpha_l + \mu_v \alpha_v$ - mixture viscosity; $\alpha_l + \alpha_v = 1$ - condition of density distribution's continuity: the sum of phase volume fractions is unity at any point of space; $\rho(\vec{\omega} \times \vec{V})$ - the term, taking into consideration the Coriolis and centripetal forces, caused by the rotation system; $\vec{V}_r$ - relative velocity;

$$\bar{\tau} = \{\tau_{ij}\} = \mu \frac{\partial v_i}{\partial x_j} - \rho \overline{V_i' V_j'} - \text{Reynolds stress tensor};$$

Two-parameter differential turbulent model $k-\omega$ SST with correction for streamline curvature and system rotation was used for closure of the system of the a.m. equations.

In the paper [5] for simulation of the cavitating over flow around cylindrical body, different cavitation models were used. The solution obtained by Zwart, Gerber and Belamri model has a best agreement with the experimental data. Furthermore, it is the least sensitive to changes of control parameters. Therefore, Zwart, Gerber and Belamri model have been used for modeling of source terms S_v , S_l from (3) responsible for the balance of the mass transfer between liquid and vapor:

$$S_v = F_v \frac{3\alpha_{nuc}(1 - \alpha_v)\rho_v}{R_B} \sqrt{\frac{2}{3} \left(\frac{P_v - P}{\rho_l} \right)}, \quad P < P_v \quad (4)$$

$$S_l = F_l \frac{3\alpha_v \rho_v}{R_B} \sqrt{\frac{2}{3} \left(\frac{P - P_v}{\rho_l} \right)}, \quad P > P_v \quad (5)$$

Default value of evaporation coefficient $F_v = 50$, condensation coefficient $F_l = 0.01$. The value of bubble radius and volume fraction of cavitation core by default: $R_B = 10^{-6}\text{m}$, $\alpha_{nuc} = 5 \cdot 10^{-4}$.

The range of programs ANSYS 15.0 has been used. As a preprocessor for mesh generation: TurboGrid and Gambit 2.4.6, as a solver: Fluent. Post processing has been performed using programs Tecplot and CFD-Post. For all transport equations, including the equations for turbulent variables, the second order discretization scheme for the convective terms has been selected. Time advancement with the second order of accuracy is carried out using Coupled scheme with time step size is equal to $\Delta t = 10^{-4}\text{s}$.

The focus is put on investigation of structure and origination of mechanism of cavitation in the tip leakage flow. Therefore, the object of investigation is tip vortex originated on the suction side of the blade. To describe this vortex quality and density of the grid in the gap and in the vicinity of blade's surface is very important. The initial grid includes 10 cells in the tip clearance. Size of the first wall cell $y_1^+ = 10$, on the blade surface $y_1^+ = 5$. Three meshes have been generated for analysis mesh convergence: base grid - Coarse, double refined - Medium, three-fold refined- Fine. Computational grid's characteristics are given in Table 1.

	Cells number	y_1^+	Expansion factor	y_1^+	Cells number
		On the blade		In the gap	
Coarse	865196	5	1.12	10	10
Medium	4119444	2.5	1.06	5	20
Fine	13104210	1	1.04	3	30

Table 1 Table of computational grid's characteristics

A conclusion on strong mesh dependency can be made based on the analysis of solution obtained on meshes with different refinement. Wherein monotonic variation of observed characteristics (pressure in the vortex core, area of low pressure iso-surface, etc.) indicates the existence of mesh convergence. Since there is no great difference between the flow patterns obtained on Medium and Fine meshes, for further calculations more cost saving mesh - Medium was used.

4. RESULTS

Analysis of the results is exemplified by Cavitation regime №2 with less cavitation number. To visualize the areas of low pressure in Fig.1, the isosurface pressure $P = 3000\text{ Pa}$ is shown. The cavitation cavern on blade suction side caused by flow separation on the leading edge is visible. Isosurfaces located along the tip clearance near the leading and trailing edges indicate the presence of cavitation vortex rope.

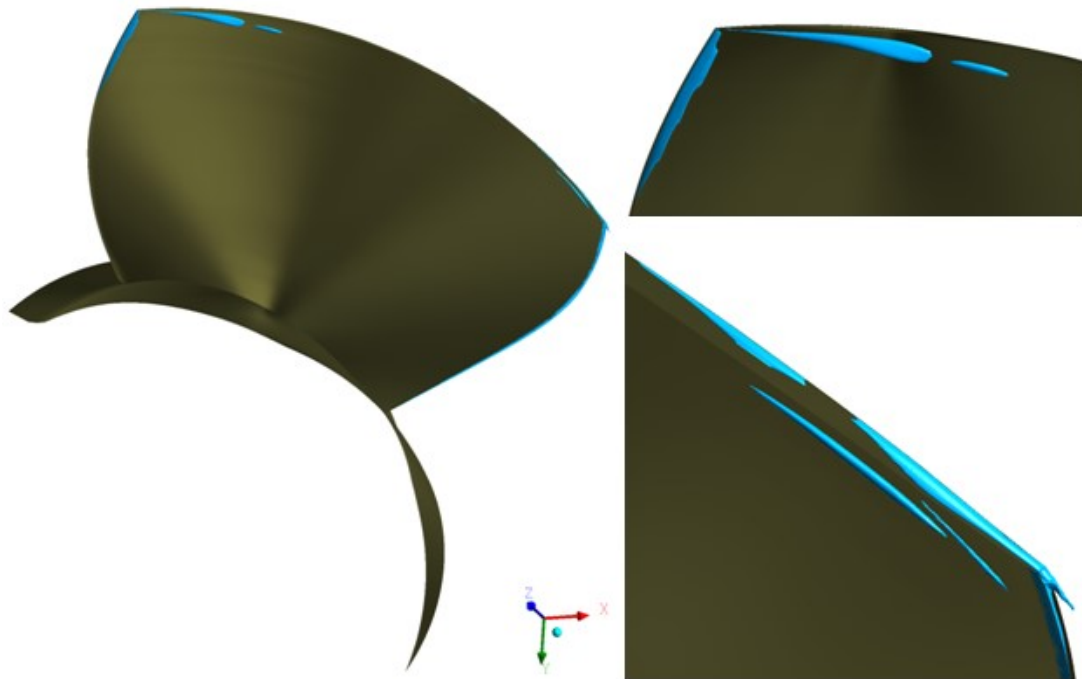


Fig.1 Isosurface of pressure $P = 3000 \text{ Pa}$ (a - general view, b - nearby to LE, c - nearby to TE)

Vortex structure of the ropes that are shaped in this turbine operating condition is shown in Fig. 2 with the help velocity streamlines. It can be seen that water flowing through the gap rolls up under the blade forming vortex thread. There are fields of absolute velocity and vorticity as well as pressure and vapor volume fraction distribution in the plane with constant angular coordinate $\varphi = 134^\circ$ (Plane-134 in Fig.3, left picture) in Fig.3, right picture and Fig.4. Due to considerable pressure difference between pressure and suction sides, water flowing through the gap has a high velocity. At the outlet of the gap, high-speed jet is shaped. On the suction side, it rolls-up to the tip vortices with low pressure in their core. Among others, the flow around the upper edge of the blade tip surface is detached, causing cavity here.

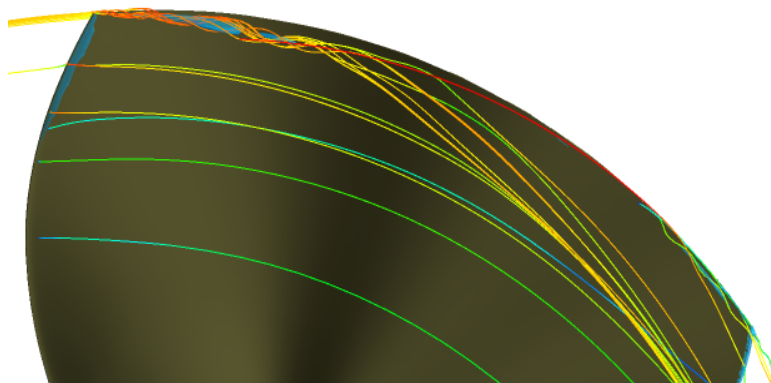


Fig. 2 Streamline velocity at the tip clearance

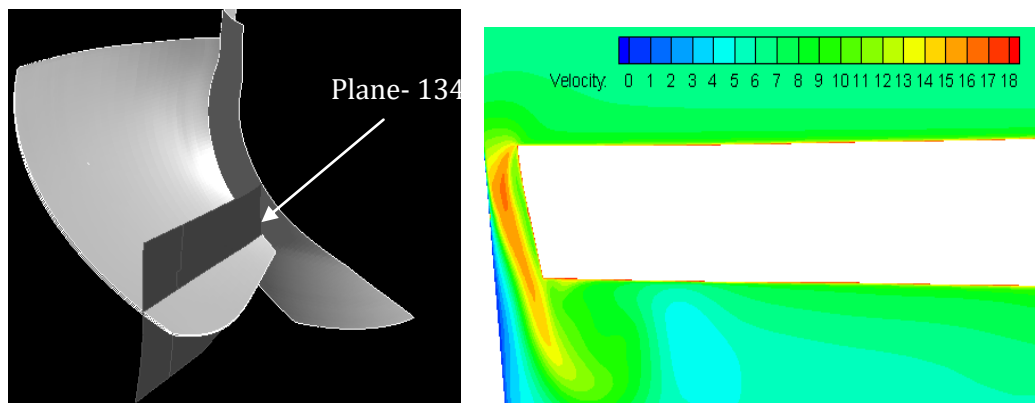


Fig. 3 Illustration of the cross-section location (left), velocity field (right)

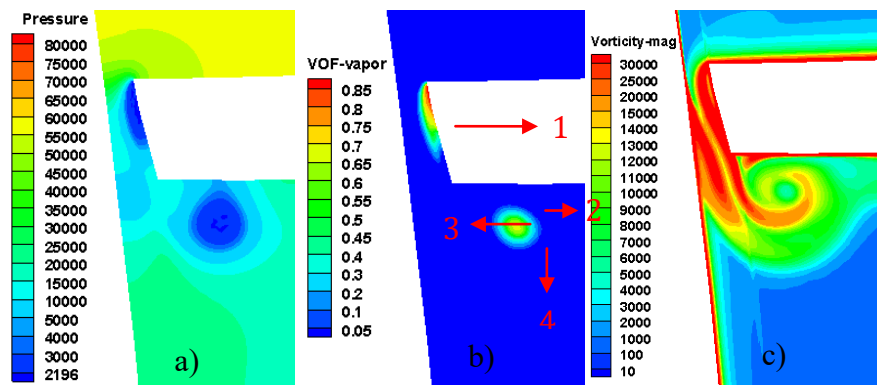


Fig. 4 Fields of pressure (a), volume fraction of vapor (b), vorticity (c) in Plane-134

In the course of solution in unsteady setting monitor points in Plane-134 have been created (Fig.4b): 1 - in the region of cavity on tip surface, 2, 3 and 4 - points in the vicinity of vortex at different distances from vortex core. Pressure pulsation in all monitored points is shown in Fig.5. It should be noted that variation in these points is not periodic and looks more like random pulsation. There is a significant correlation between pulsation in the region of cavity on tip surface and in the vortex. In the place of the cavity closure (1) the mean value of pressure $P_1 = 2412$ Pa, and maximum deviation from the mean value is approximately 5%. In the point (2) the mean pressure $P_2 = 2241$ Pa, and maximum deviation from the mean value is approximately 4%. At distance increase from core vortex, the mean value and amplitude of pressure pulsation also increase. There is no pulsation in the core.

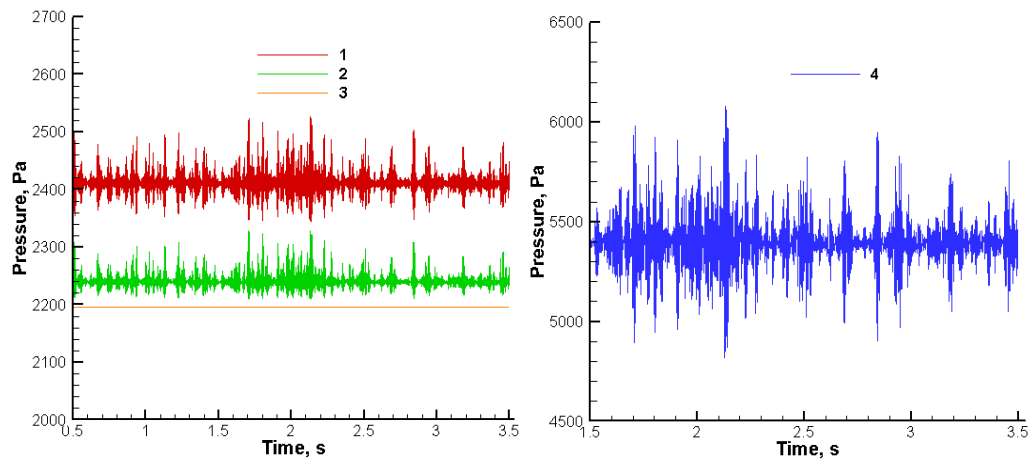


Fig. 5 Variation of pressure in monitor points 1, 2, 3, 4

Shown in Fig.6a are curves of pulsation of velocity dimensionless components. Fourier-analysis amplitude distribution of velocity fluctuation in frequencies was obtained. Pulsation of radial velocity has the largest amplitude (approximately 10%).

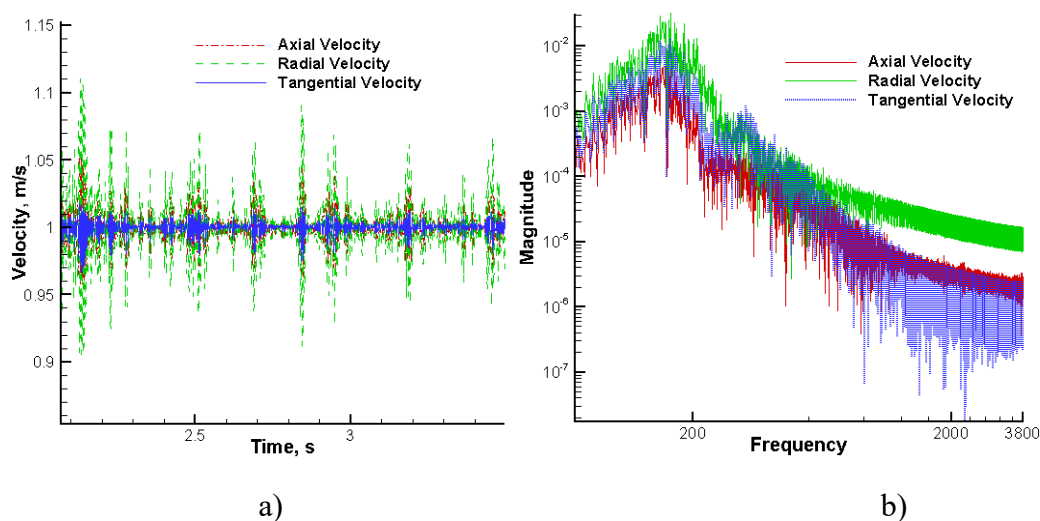


Fig. 6 Fluctuation of dimensionless velocity components in monitored point 4 (a), Frequency response function of signals is given in the left picture (b)

From frequency response function analysis shown in Fig. 6b it can be seen that the most high-amplitude pulsation in the frequency is ranging from 100 to 200 Hz. The lower level of this range is close to the value of blade frequency, and upper level is a double blade frequency. The represented range has a well-defined maximum corresponding to frequency 161.7 Hz. Note that these pulsations are present both in the cavitation mode and in operation without cavitation.

The Strouhal number (Sh) of the pressure and velocity pulsation based on the cavity length in the tip gap was $Sh=0.61$ and for the rolled up vortex $Sh=0.96$. Unfortunately, measuring the pressure fluctuations in the tip gaps not met and we cannot compare the magnitude of pressure fluctuations produced on the basis of calculations and measurements. The

instrumental measurements of pressure pulsations were produced only in the points located on the walls of the draft tube cone. At these points, the low-frequency pressure pulsations were recorded corresponding to the frequency of rotation of the runner in the case of cavitation and without it.

The data obtained during model tests of the Kaplan turbine runner are used for comparison with numerical investigation results. Iso-surfaces of vapor volume fraction $\alpha_v = 0.5$ obtained by computation.

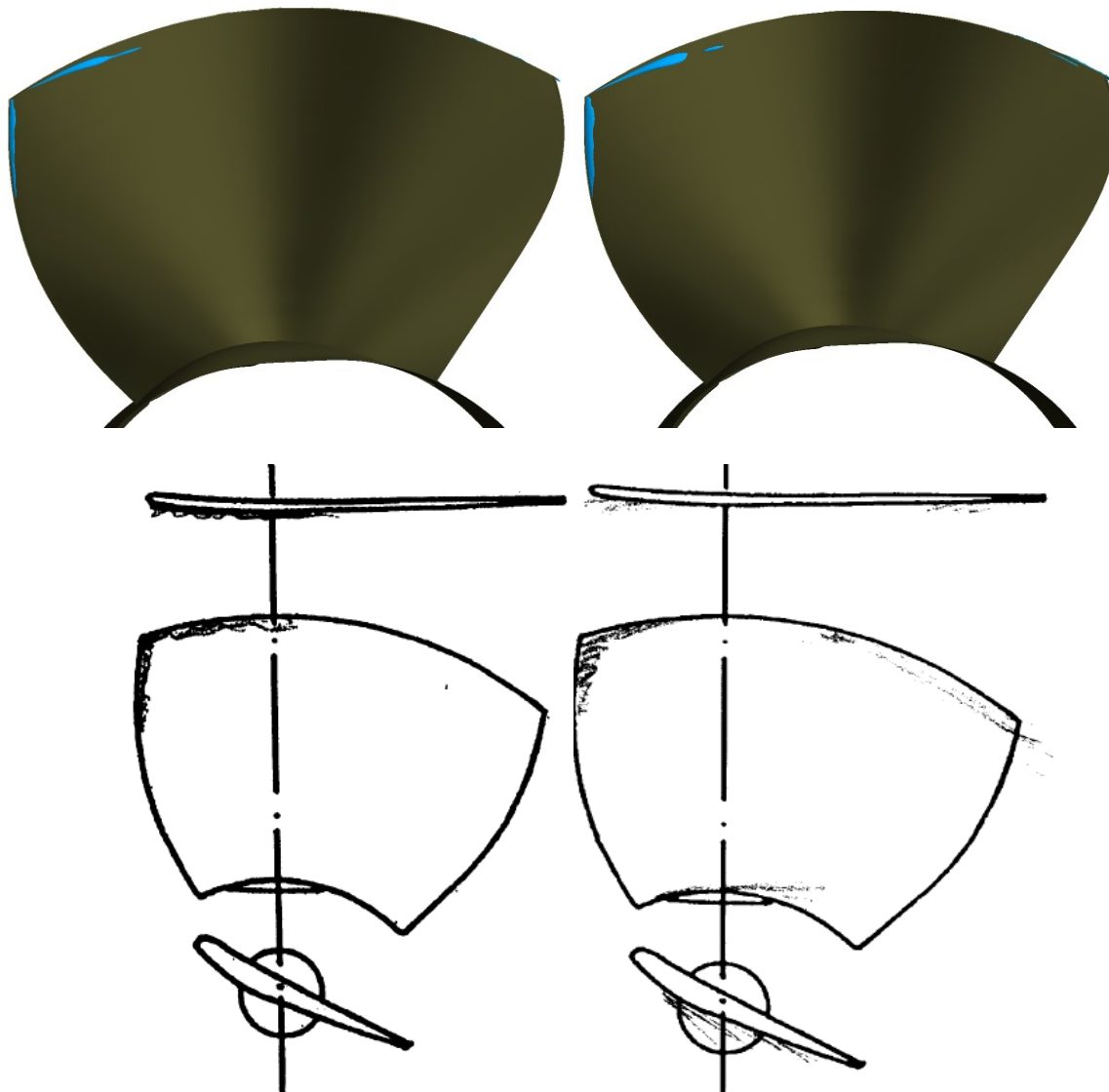


Fig. 7 Comparison of numerical computation results with experimental data for two analyzed regimes (left - Cavitation regime №1, right - Cavitation regime №2)

The view of blade from suction side is shown in Fig.7. For both cavitation regimes existence of surface profile cavitation on the LE is typical. Length of the cavity in this area obtained in experiment has a good agreement with numerical investigation results. As for vortex cavitation along clearance gap, it can be seen that location of zone with high vapor volume

fraction has a good agreement with experiment. Such kind of cavitation has been observed along the gap nearby leading and trailing edges. Photos that are shown in Fig.8 allow to consider the flow in more details. It should be noted that lack of cavitation clouds on hub in the computational results related to the geometry of computational domain was generated without the gap between blade and hub.

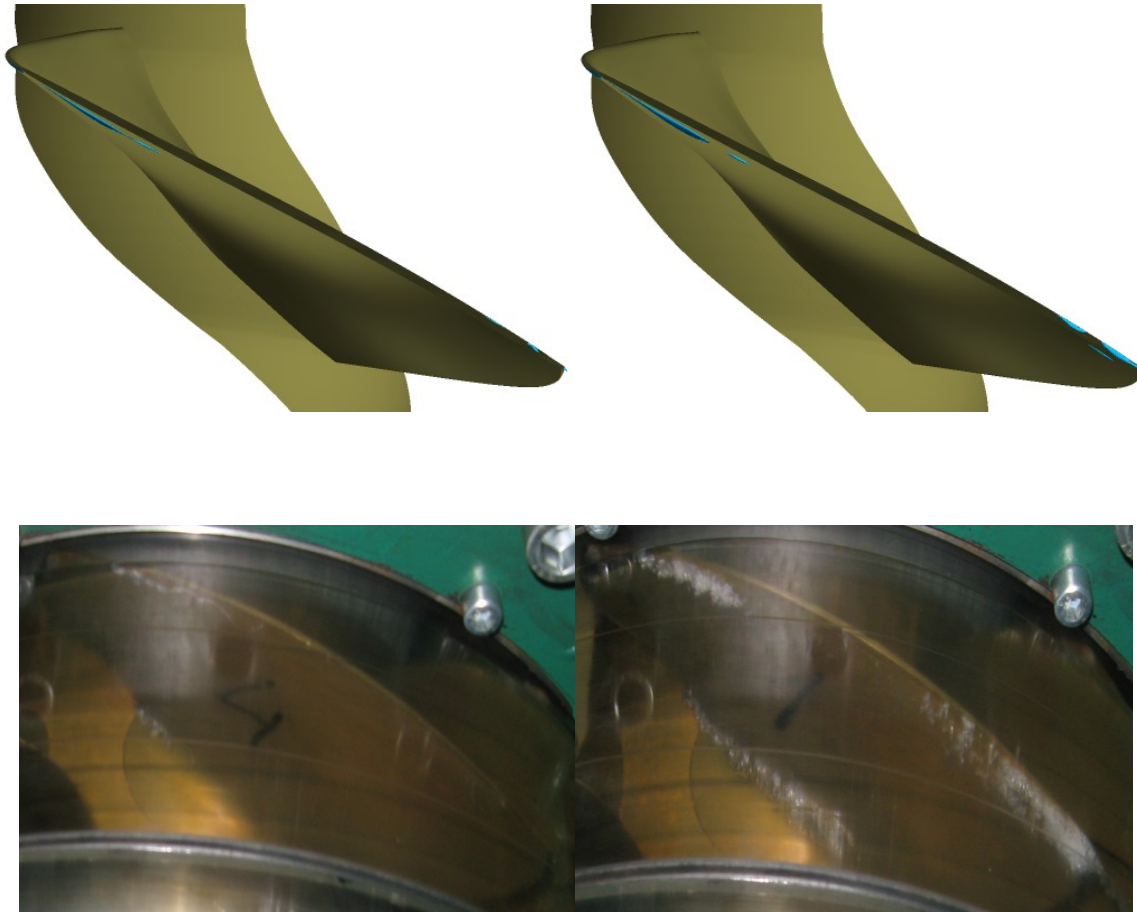


Fig. 8 Comparison of numerical computation results with experimental data for two analyzed regimes (left - Cavitation regime №1, right - Cavitation regime №2)

On photo obtained during model tests for Cavitation regime №1 thin vortex rope full of vapor is clearly visible. For Cavitation regime №2 concentration of single fairly small bubbles is seen. Zones of their location are well predicted with the help numerical computation. Numerical investigation results for Cavitation regime №1 have better agreement with experiment. Firstly, it is due to the inability to resolve small vortex structures with small vapor bubbles in the core using unsteady Reynolds equations. It is known that modeling of cavitation doesn't imply the consideration of every single bubble life cycle and considers the two-phase liquid as a mixture with continuous distribution of density for both phases. The cavitation model describes more accurately the cavitation phenomena that analyze large cavities full of vapor. That is why vaporization intensity in the area of tip clearance obtained by calculating is slightly reduced in comparison with experiment for Cavitation regime №2.

5. CONCLUSION

As a result, the engineering procedures for calculation of cavitation caused by existence of tip clearance in model turbine runner have been developed. The prototype of this runner is taken from the existing Djerdap-1 HPP in Serbia. In the course of the work, the obtained results make it possible to visualize and describe the inner structure of cavitating vortex rope caused by the tip leakage through the gap. Besides, numerical simulation gives us information that is rather difficult and sometimes completely impossible to obtain by experiments. Based on spectral analysis of obtained data the range of frequency with maximum value of pulsation was assigned and maximum energy frequency was defined. As for experiment measurements in the immediate vicinity to the wall, there is a big problem. So, performing of these calculations gives a more complete view of the investigated process. Comparison of numerical computation results with experimental data has been also performed. The location of cavitation zone has a good agreement with the experiment. Still, it should be noted that for regime with less cavitation number vaporization intensity in the vortex is slightly reduced as compared with the experimental data. It can be connected with inability to reproduce bubble cavitation used via approach for description liquid as a quasi-homogeneous mixture in aggregate with URANS modeling of turbulence. In future it is advisable to carry out cavitation calculation using LES approach with aim to more accurately consider the mechanism of vortex cavitation inception and to define the nature of maximum energy frequency.

6. REFERENCES

- [1] Nilsson H., Davidson L. 2002 Numerical Investigations of Turbulent Flow in Water Turbines. *Thesis for the degree of Doctor of Philosophy (Sweden)*.
- [2] Bensow R.E., Bark G. 2010 Simulating Cavitating Flows with LES in Open FOAM, *V European Conference on Computational Fluid Dynamics ECCOMAS CFD (Lisbon)*.
- [3] Ortiz E. R., Gagnon J. M. and Deschênes C. 2010 Numerical Simulation in the Runner of a Propeller Turbine - Tip Leakage Flow and Blade Tip Vortex. *(Annual General Meeting CFD Society of Canada and Lunch 18 May 2010)*.
- [4] Mortier C., Koller M., Braun O. 2011 CFD prediction of cavitation inception in the tip clearance vortex of Kaplan turbines. Impact of mesh refinement and turbulence model choice. *SHF Conference on Cavitation and Hydraulic Machines (Lausanne, May 26 2011)*.
- [5] Isaenko I., Ivanov N. 2013 Cavitation models in ANSYS CFD. *Master's thesis (St.-Petersburg Polytechnical University)*.
- [6] Kevin J. Farrell. 1989 An investigation of end-wall vortex cavitation in high Reynolds number axial-flow pump. *Technical Report No. TR 89-004 (The Pennsylvania State University Applied Research Laboratory, August 1989)*.