

Numerical Prediction of Cavitating Vortex Rope in a Draft Tube of a Francis Turbine with Standard and Calibrated Cavitation Model

This content has been downloaded from IOPscience. Please scroll down to see the full text.

2017 J. Phys.: Conf. Ser. 813 012045

(<http://iopscience.iop.org/1742-6596/813/1/012045>)

View [the table of contents for this issue](#), or go to the [journal homepage](#) for more

Download details:

IP Address: 151.51.185.75

This content was downloaded on 10/05/2017 at 16:47

Please note that [terms and conditions apply](#).

You may also be interested in:

[Numerical simulation and analysis of the internal flow in a Francis turbine with air admission](#)

A Yu, X W Luo and B Ji

[Unsteady hydraulic simulation of the cavitating part load vortex rope in Francis turbines](#)

J Brammer, C Segoufin, F Duparchy et al.

[The optimization of j-groove shape in the draft tube of a francis turbine to suppress the draft surge](#)

Q S Wei and Y D Choi

[Experimental identification and study of hydraulic resonance test rig with Francis turbine operating at partial load](#)

A Favrel, C Landry, A Müller et al.

[Numerical simulation of full load surge in Francis turbines based on three-dimensional cavitating flow model](#)

D Chirkov, L Panov, S Cherny et al.

[Study on flow instability and countermeasure in a draft tube with swirling flow](#)

T Nakashima, R Matsuzaka, K Miyagawa et al.

[Numerical investigation of draft tube pressure pulsations in a Francis turbine with splitter blades](#)

I Kassanos, J Anagnostopoulos and D Papantonis

[Large eddy simulation of pressure fluctuations at off-design condition in a Francis turbine based on cavitation model](#)

W T Su, X B Li, F C Li et al.

Numerical Prediction of Cavitating Vortex Rope in a Draft Tube of a Francis Turbine with Standard and Calibrated Cavitation Model

D Jošt^{1,3}, A Škerlavaj^{1,2}, M Morgut² and E Nobile²

¹ Kolektor Turboinštitut, Ljubljana, Slovenia

² University of Trieste, Trieste, Italy

³ corresponding author dragica.jost@kolektor.com

Abstract. Transient simulations of flow in a Francis turbine were performed with a goal to predict pressure pulsation frequencies and amplitudes caused by rotating vortex rope at part load operating regime. Simulations were done with the SAS SST turbulence model with curvature correction on basic and refined computational meshes. Without cavitation modelling too small values of frequency and amplitudes were obtained. With mesh refinement the calculated amplitudes were a bit closer to the measured values, while the accuracy of predicted frequency did not improve at all. Agreement between measured and numerical values was significantly improved when cavitation was included in simulations. In addition, the predicted value of the dominant frequency was slightly more accurate when, in the Zwart et al. cavitation model, the default condensation and evaporation model constants were replaced with previously calibrated ones.

1. Introduction

Pressure fluctuations caused by cavitating vortex rope at part load operating regimes can be a serious problem for Francis and also for single regulated axial turbines. The consequences of the vortex developed in the draft tube are very unpleasant pressure pulsations, axial and radial forces and torque fluctuation as well as turbine structure vibrations. The result of intensive experimental investigations of draft tube pressure pulsation for various turbines of different specific speed proved that the intensity of the vortex depends strongly on the shape of runner blades and channel [1]. Therefore, an accurate numerical prediction of pressure pulsation caused by vortex rope is crucial in a design stage, before the runner with undesirable dynamic characteristics is manufactured.

The first numerically obtained helical vortex was presented by Skotak [2]. In spite of a too coarse mesh for the Large Eddy Simulation (LES) model, a low pressure zone which agreed well with the rotating rope observed in experiment was obtained. In the following years, several papers about this topic were published. Generally, the frequency of pressure pulsation matched the measured values quite well, but the prediction of amplitudes was less accurate. In [3] the detailed comparison of numerical (SAS SST, no cavitation modelling) and experimental results at different part load operating points showed good agreement of vortex rope shape, length, thickness as well as pressure pulsation frequencies, while the predicted amplitudes were too small. The results improved with mesh refinement, but even on the refined mesh with 12 million nodes in the draft tube, the discrepancy was around 14%. Cavitating vortex rope was modelled in [4, 5, 6] and it was reported that a size of the predicted cavitating rope was smaller than the one observed during experimental tests. In [6] the results obtained with three turbulence models (SAS-SST, RSM and LES) were presented. When cavitation was not modeled, there was no significant difference in the accuracy of the results due to



different turbulence models. In all cases predicted pressure pulsation amplitudes were significantly too small. When cavitation was included the results obtained with SAS-SST and RSM were less accurate than the results obtained with LES. The results of the RSM model were so irregular, that pressure pulsation frequency could not be obtained. For the simulation with the LES model, the mesh in the draft tube was refined (23.5 million nodes) and it is possible that the mesh refinement contributed even more to the improvement of the results than the model itself. Anyway, also with the LES model on refined computational mesh, the amplitudes of pressure pulsation were considerably too small, discrepancy with measurements was at one location 12.3% and at the other location even 31.5%.

Kolektor Turboinštitut (Slovenia) and University of Trieste (Italy) joined in the ACCUSIM EU project with the aim to develop reliable, high fidelity methods for accurate predictions and optimization of performances of hydro-machinery and marine propellers. One of the main goals of the ACCUSIM project is to improve the prediction of all types of cavitation in water turbines including cavitating vortex rope in the draft tube of a Francis turbine. With this purpose a cavitating vortex rope was modelled with the Zwart mass transfer model using standard and calibrated evaporation and condensation parameters. The first results for part load operating regime are presented in this paper.

2. Mathematical Model

2.1. Turbulence Modelling

Even though flow in water turbines is unsteady, the efficiency and cavitation in Francis turbines can be predicted by a steady state flow analysis and the results are usually enough accurate for design purposes. But for prediction of rotating vortex rope transient simulations with one of the advanced turbulence models (Reynolds Stress Models (RSM), Large Eddy Simulation (LES), Detached Eddy Simulation (DES) or Scale-Adaptive Simulation (SAS)) have to be performed. From our previous experiences [3, 6] it was concluded that the most economic turbulence model for the vortex rope prediction is the SAS SST model with the curvature correction. Therefore this model was used for the case presented in this paper. A detailed description about SAS SST and curvature correction can be found in [7] and [8].

2.2. Multiphase Flow and Cavitation Modelling

Cavitation refers to the process by which vapour forms in low pressure regions of a liquid flow. Cavitation is numerically usually modelled by the homogeneous multiphase model. The homogeneous model assumes that the velocity field for the transported quantities (with the exception of volume fraction) for the process is the same for all phases. Therefore it is sufficient to solve bulk transport equations for shared fields rather than solving individual transport equations. Mass transfer from liquid to vapour is modelled by different cavitation models. In this study, the Zwart et al. model [9], which is based on the simplified Rayleigh-Plesset equation for bubble dynamics, was used. The evaporation and condensation processes are regulated with empirical constants F_e and F_c . The default settings in ANSYS CFX are equal to $F_e=50$ and $F_c=0.01$, respectively.

A calibration of empirical constants using an optimization strategy is presented in [10]. The entire calibration process was driven by the modeFRONTIER 4.2 optimization system, which is a general integration and multi-objective optimization platform commonly used for functional and shape optimization of systems and devices. With the aim to reduce computational costs the empirical coefficients were optimized on a two-dimensional sheet cavity flow around the NACA66(MOD) hydrofoil. The evaporation and condensation coefficients of the Zwart model were tuned within the following ranges: $30 \leq F_e \leq 500$ and $0.0005 \leq F_c \leq 0.08$. The best result was found with $F_e = 300$ and $F_c = 0.03$. Details about optimization process can be found in [10]. Calibrated values for F_e and F_c were successfully used for prediction of cavitating flow around model scale propellers in uniform and non-uniform inflow [11, 12] and also for cavitation prediction in a Kaplan turbine [13]. In this study, besides default values, also calibrated constants were used for prediction of cavitating vortex rope.

3. Model Test

Pressure fluctuations on Francis model turbine were measured in accordance with IEC 90193. KISTLER piezoresistive absolute pressure transducers were located at several locations. Signals from

the pressure transducers were wired to the multichannel data acquisition system, based on National Instruments multifunction card with additional SCXI signal conditioning modules. Signals were acquired continuously with 5 KHz sampling frequency and 16 bit resolution. Binary data samples stored on the computer hard disk were at least 30 seconds long. LabVIEW software was used for analysis of pressure signals.

With this measurements information about magnitude and dominant frequency of pressure fluctuations at the spiral casing intake, at two locations at the draft tube cone, at one location at the draft tube elbow and at one location at the extension of the draft tube were obtained. For validation of numerical results only the values at the draft tube cone (see figure 1) were used. Dominant frequency was obtained with the Fast Fourier Transformation. For amplitudes 2% of extreme values in the sample were not taken into account, so actually peak-to-peak amplitudes obtained from 98% of amplitude values were compared to the numerical results.

Although the focus of this study is not efficiency prediction, it was interesting to see the effect of cavitation modelling on accuracy of calculated efficiency values. Therefore calculated values of flow rate, torque on the shaft and efficiency were compared to the measured values.

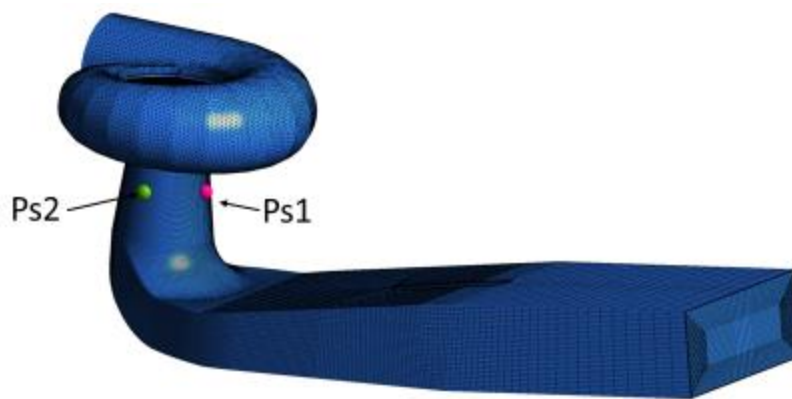


Figure 1: Computational mesh and positions of pressure measurements on conical part of the draft tube

4. Numerical Prediction

Prediction was done for a middle head Francis turbine at one operating point at part load operating regimes ($\phi/\phi_{BEP}=0.8$, $\psi/\psi_{BEP}=0.97$). Transient simulations without cavitation modelling were performed first, then the cavitation was included. The standard and calibrated evaporation and condensation constants were used for cavitation modelling.

4.1. Computational Domain and Meshes

The computational domain for numerical flow analysis consists of a spiral casing with stay vanes, a guide vane cascade, a runner and a draft tube. An extension of the draft tube was added in order to move the outlet boundary condition away from the region of interest. Flow energy losses in the draft tube extension were not taken into account when the turbine efficiency was calculated.

While the mesh in the spiral casing was unstructured, the meshes in all other turbine parts were structured. The meshes are rather coarse, except in the draft tube, where mesh was at first moderately (basic mesh - BM) and later significantly refined (fine mesh - FM). Number of nodes and elements for meshes of all turbine parts are presented in table 1, while the basic computational mesh can be seen in figure 1.

4.2. Boundary and initial conditions

Simulations are more stable, if flow rate is input data and head a result of computation. But the flow rate corresponding to each guide vane opening is not known in advance. The purpose of numerical

simulations is to minimize measurements, therefore it is more sensible to check the accuracy of numerical results with prescribed head, while the value of flow rate is a result of numerical simulation. Therefore in this study input data were geometry, rotating speed and head, while the value of flow rate, torque on the shaft and efficiency were the results of numerical simulation. At the inlet of the computational domain, total pressure $P_{tot,in}$ was prescribed. $P_{tot,in}$ was defined by the expression $P_{tot,in} = \rho g H + P_{tot,out}$, where ρ is water density, g is gravity, H is turbine head and $P_{tot,out}$ is total pressure at the draft tube outlet obtained in the previous iteration of numerical simulation. At the outlet, static pressure calculated from cavitation coefficient σ was prescribed.

Table 1. Number of nodes and elements in various computational meshes

Domain	Nodes	Elements
Spiral casing	167,605	422,336
Stay vane cascade	300,863	270,996
Guide vane cascade	886,200	801,312
Runner	1,000,428	936,832
Draft tube extension	196,353	187,520
Draft tube, BM	2,340,295	2,302,752
Draft tube, FM	12,032,526	11,917,764
Total, BM	4,891,744	4,921,748
Total, FM	15,143,184	15,063,064

In steady state simulations the frozen rotor condition was used between the runner and stationary turbine parts. In transient simulations, a transient rotor stator condition was prescribed. With this condition, the position of runner blades is updated at each time step and all effects of mixing due to runner rotation are taken into account.

At first, a steady state simulation without cavitation modelling was performed on the basic mesh. These results were used as initial condition for transient simulations without cavitation and

also as an initial for steady state simulations with cavitation. The results of the latter ones were used as initial conditions for transient simulations with cavitation. The simulation on the refined mesh was performed only without cavitation modelling, it started from the results obtained on the basic mesh.

Time step in transient simulation corresponded to two degrees of runner revolution. Maximal and average values of Courant number in the draft tube were in case of the basic mesh equal to 7.14 and 0.21, respectively. In case of the fine mesh the time step size was not reduced, therefore maximal and averaged values of Courant number increased to 10.1 and 0.55, respectively. Time of transient simulation without cavitation modelling on BM corresponded to 60 runner revolutions, on the refined mesh additional 10 runner revolutions were performed. With cavitation modelling in both cases, with default and calibrated parameters, about 50 runner revolutions were simulated.

4.3. Results and discussion

At the operating point treated in this study, a strong, compact vortex rope was observed on the test rig. Similar shape and thickness of the rope was obtained numerically (see figure 2). For simulations without cavitation modelling the rope is presented with iso-surface of absolute pressure equal to evaporation pressure, while cavitating vortex rope is presented with iso-surface of Vapour Volume Fraction = 0.1. In case of cavitation modelling, the rope has a staircase-like surface due to insufficiently refined mesh in a vertical direction. The cross section of the rope is more elliptical, which is more typical for the operating regimes closer to the BEP.

Pressure pulsations at Ps1 and Ps2 obtained with numerical simulations are presented in figure 3. It is clearly seen that without cavitation modelling the amplitudes are significantly smaller than when cavitation is included. With mesh refinement pressure pulsations become more regular and larger. With cavitation modelling some extra peaks can be observed. Similarly as for experimental results, where 2% of extreme values were not taken into account, also these peaks were ignored when pressure pulsation peak-to-peak amplitudes were determined.

A comparison of calculated flow rate, torque on the shaft and turbine efficiency to the measured values is presented at the top of figure 4. All simulations overestimated values of flow rate and torque on the shaft. When cavitation was not included, the predicted flow rate was about 2.6% larger than the measured one. With cavitation modelling discrepancy increased to about 3.4%. The value of torque was in all simulations overestimated for about 4%. The effect of mesh refinement on calculated flow rate and torque was negligible. An important reason for discrepancy between predicted and measured values of flow rate and torque on the shaft is that in this study volumetric and torque losses in

labyrinth seals were not taken into account. From the results for a Francis turbine presented in Workshop Francis'99 [14] it is evident, that these losses are significant at part load, but even at the best efficiency point the volumetric losses and especially the torque losses should not be neglected.

An accuracy of a numerically predicted efficiency value depends on accuracy of predicted flow rate and torque. When cavitation was not modelled, calculated efficiency is about 1.73% larger than the measured one, while with cavitation modelling the discrepancy decreased to about 0.7%. Again, the effect of calibration of cavitation constants is negligible.

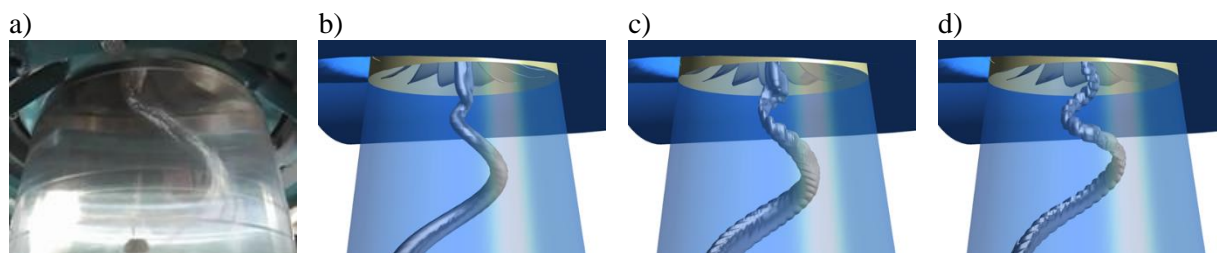


Figure 2. Vortex rope a) on the test rig; b) without cavitation modelling, iso-surface of evaporation pressure, basic mesh; c) default cavitation constants, iso-surface of Vapour Volume Fraction = 0.1 d) calibrated cavitation constants, iso-surface of Vapour Volume Fraction = 0.1.

Due to too large values of flow rate the operating point being treated is in numerical simulations a bit closer to the best efficiency point than in the measurements. This can contribute to the discrepancy between predicted and measured values of pressure pulsation frequencies and amplitudes.

In figure 4 also pressure pulsation frequency and peak-to-peak amplitudes are presented. Pressure pulsation frequencies are divided by frequency of runner rotation. Peak-to-peak amplitudes are presented in relative form: $App = \Delta p / (\rho g H)$, where p is pressure, ρ is water density, g is gravity and H is turbine head. Numerically, without cavitation modelling, significantly too small values of pressure pulsation frequency and peak-to-peak amplitudes were obtained. Predicted pressure pulsation frequency was about 8% smaller than the measured one and no improvement of accuracy was achieved with mesh refinement. On the basic mesh discrepancies between measured and calculated App at Ps1 and Ps2 were equal to 56.3% and 29.4%, respectively. Besides, contrary to the experimental results, where pressure pulsations were stronger at Ps1, higher amplitudes were obtained at Ps2. With mesh refinement only very small improvement was achieved for App at Ps1, while at Ps2 improvement was larger – the discrepancy decreased to 17.8%.

With cavitation modelling simulations were performed only on the basic mesh. The accuracy of results improved significantly. Dominant frequency was most accurately predicted in case of calibrated evaporation and condensation constants where the discrepancy to the measured value was below 1.4%. Pressure pulsation amplitudes were most accurately predicted with default constants where discrepancies between measured and calculated App values at Ps1 and Ps2 were equal to 3.2% and 0.13%, respectively. With calibrated constants the discrepancies for App values at Ps1 and Ps2 were equal to 0.3% and 10%, respectively.

The reason for relatively small positive effect of mesh refinement could be that only the mesh in the draft tube was refined, while mesh in the runner, where vortex rope started to form, was rather coarse. On interface between the runner and the draft tube considerably larger elements were on the runner side. Therefore the flow at the inlet of the draft tube was probably not enough accurately calculated. In [3], where positive effect of mesh refinement was clearly seen, the mesh in the runner had 12 million elements and it was much more refined than in this case. Our experiences with cavitation modelling are that mesh should be fine enough to get proper extend of cavitation. Therefore we plan to repeat simulation without cavitation and with cavitation on computational mesh, which will be refined in the runner and in the draft tube.

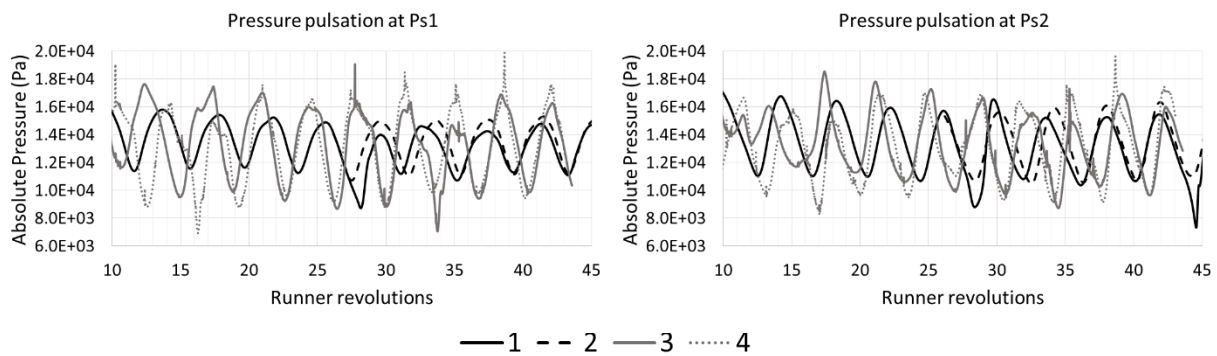


Figure 3. Pressure pulsation at two locations on the cone of the draft tube
 1 – no cavitation modelling, BM; 2 – no cavitation modelling, FM; 3 – cavitation modelled with default constants, BM; 4 – cavitation modelled with calibrated constants, BM.

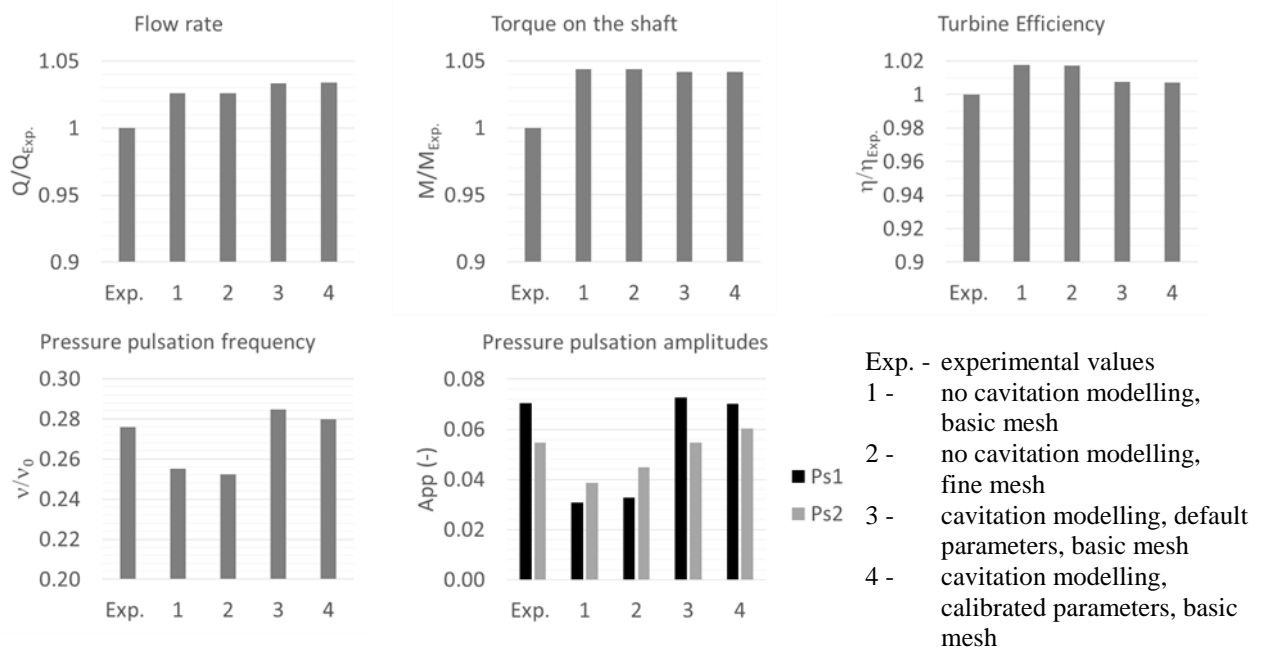


Figure 4. Comparison of numerical and experimental results

5. Conclusions

Transient simulations with SAS SST turbulence model with curvature correction were performed at part load operating regime with and without cavitation modelling. The goal was to predict pressure pulsations caused by rotating vortex rope. Numerical values of dominant frequency and peak-to-peak amplitudes at two locations on the cone of the draft tube were compared to the experimental results. The following conclusions can be deduced:

- Numerical simulations without cavitation modelling significantly underestimated values of pressure pulsation frequency and amplitudes caused by rotating vortex rope. Refinement of the mesh in the draft tube improved only prediction of peak-to-peak amplitude at one location, while generally the results were nearly the same as on the basic mesh.
- Agreement between measured and numerically predicted values of frequency and peak-to-peak amplitudes was significantly improved when cavitation was included in simulations.

- With a previously calibrated evaporation and condensation constants in Zwart at al. cavitation model, a bit more accurate value of dominant frequency was obtained. Accuracy of peak-to-peak amplitudes at one location improved, while at the other location too large value was obtained.

References

- [1] Kercan V, Bajd M, Djelić V, Lipej A, Jošt D 1996 Model and prototype draft tube pressure pulsations. *Proceedings of the XVIII. IAHR Symposium on Hydraulic Machinery and Cavitation, Volume II*, p 994-1003.
- [2] Skotak A 1999 Draft tube swirl modeling. *Proceedings of the 9th International Meeting of IAHR Work Group on the Behavior of Hydraulic Machinery under Steady Oscillatory Conditions*, 6 pages, D4.
- [3] Lipej A, Jošt D, Mežnar M, Djelić V 2008 Numerical prediction of pressure pulsation amplitude for different operating regimes of Francis turbine draft tube. *24th IAHR Symposium on Hydraulic Machinery and Systems*, 10 pages, (IAHR 93.pdf)
- [4] Guo Y, Kato C, Miyagawa K 2006 Large-eddy simulation of non-cavitating and cavitating flows in an elbow draft tube. *23rd IAHR Symposium on Hydraulic Machinery and Systems*, 10 pages, F195.pdf
- [5] Zhou L, Wang Z, Tian Y 2006 Numerical simulation of vortex cavitation in draft tube. *23rd IAHR Symposium on Hydraulic Machinery and Systems*, 8 pages, F102.pdf
- [6] Jošt D and Lipej A 2001 Numerical Prediction of Non-Cavitating and Cavitating Vortex Rope in a Francis Turbine Draft Tube, *Journal of Mechanical Engineering* 57(2011)No. pp.
- [7] Smirnov P E and Menter F 2009 Sensitization of the SST turbulence model to rotation and curvature by applying the Spalart-Shur correction term J. *Turbomach.* 131 041010
- [8] Egorov Y and Menter F 2008 Development and application of SST-SAS turbulence model in the DESIDER project *Advances in Hybrid RANS-LES Modelling* ed. S-H Peng and W Haase (Heidelberg: Springer) pp 261–70
- [9] Zwart P, Gerber A G and Belamri T 2004 A Two-Phase Model for Predicting Cavitation Dynamics, *ICMF 2004 International Conference on Multiphase Flow, Yokohama, Japan*
- [10] Morgut M, Nobile E and Biluš I 2010 Comparison of mass transfer models for the numerical prediction of sheet cavitation around a hydrofoil. *Int. Journal of Multiphase Flow*. Vol. 37. No. 6. 2010. pp. 620-626
- [11] Morgut M Nobile E 2012 Numerical Predictions of Cavitating Flow around Model Scale Propellers by CFD and Advanced Model Calibration. *International Journal of Rotating Machinery*. Vol. 2012. 2012. Article ID 618180. 11 p.
- [12] Morgut M, Jošt D, Nobile E and Škerlavaj A 2015. Numerical investigations of a cavitating propeller in non-uniform inflow. *Fourth International Symposium on Marine Propulsors, smp'15, Austin, Texas, USA*
- [13] Jošt D, Škerlavaj A, Morgut M, Nobile E 2015 Cavitation Prediction in a Kaplan Turbine Using Standard and Optimized Model Parameters, *6th IAHR International Meeting of the Workgroup on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems*, September 9-11, 2015, Ljubljana, Slovenia
- [14] Jošt D, Škerlavaj A, Morgut M, Mežnar P, Nobile E 2014 Numerical simulation of flow in a high head Francis turbine with prediction of efficiency, rotor stator interaction and vortex structures in the draft tube, *Workshop Francis 99*, Trondheim, Norway.

Acknowledgments

The research leading to these results has received funding from the People Programme (Marie Curie Actions) of the European Union's Seventh Framework Programme FP7/2007-2013/ under REA grant agreement n°612279 and from the Slovenian Research Agency ARRS - Contracts No. 1000-15-0263 and P2-0196.