6th IAHR International Meeting of the Workgroup on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems, September 9-11, 2015, Ljubljana, Slovenia

CFD SIMULATION OF THE FLOW THROUGH A PUMP TURBINE DURING A FAST TRANSITION FROM PUMP TO GENERATING MODE

Christine STENS*

Institute of Fluid Mechanics and Hydraulic Machinery, University of Stuttgart, Germany

Stefan RIEDELBAUCH

Institute of Fluid Mechanics and Hydraulic Machinery, University of Stuttgart, Germany

ABSTRACT

Pumped storage power plants are an efficient way to store energy at a large scale. In the last years, the changes between pump and turbine mode have become more and more frequent and the necessity of fast changes has increased.

This paper analyses the flow in a complete model scale pump turbine during a fast transition from pump mode to generating mode by means of CFD. A linear variation of rotational speed over time is chosen. A time-dependent flow rate through the machine is prescribed at the inlet. Due to the varying conditions, a fully transient analysis is carried out using the open-source code OpenFOAM 2.2. Discussed results include the flow in guide vanes, runner and draft tube as well as an analysis of the pressure on the runner blades.

The presentation of the results starts with the examination of the inlet and outlet boundary conditions. Geometry extensions of the pipe at the spiral case as well as the draft tube are compared to each other.

As flow rate decreases in pump mode, stall in the guide vanes leads to large pressure fluctuations especially on the leading edge of the runner blades. In the guide vane region, flow concentrates on single channels while other channels experience conditions near blocking.

In the runner, pressure fluctuations due to stall are clearly visible on the pressure side of the blades near the guide vanes at high head pump operation. In pump break mode, the pressure distribution is found to be inhomogeneous on each blade as well as between the blades. Finally, the pressure near the runner outlet is influenced by the draft tube flow, especially at the beginning of generating mode.

In the draft tube, flow detaches from the outer walls starting from below the runner at low flowrates in pump mode. With decreasing flowrate, the detached zone propagates towards the elbow and a strong swirl forms at the walls of the draft tube. Shortly before the runner reverses its direction, four vortex ropes appear starting from the hub side of the runner, that collapse to two and finally one vortex rope in turbine mode.

KEYWORDS

Pump-Turbine, CFD, transient, mode change

1. INTRODUCTION

Pumped storage power plants offer an efficient way to store electric energy on a large scale. Due to a growing share of renewables in the electric grid, the number of changes between pump mode and generating mode has increased over the last years. While it is common practice to investigate the behaviour of hydraulic machines at stationary operating points by means of CFD, little work has been published regarding the simulation of transients of

* *Corresponding author:* Institute of Fluid Mechanics and Hydraulic Machinery, University of Stuttgart, Pfaffenwaldring 10, 70569 Stuttgart, Germany, phone: +49 711 685 63262, email: stens@ihs.uni-stuttgart.de

turbines and pump turbines, such as start up, shut down, runaway or changes between pump and generating mode. This is partly due to the computational resources required for these simulations that became available only recently.

An overview of possible flow phenomena in reversible pump turbines is given in [1]. First numerical studies were carried out for the case of shutdown on the relatively simple geometry of an axial turbine [2], already featuring mesh adaption as the blades are adjusted. Li et al. [3] investigated the flow through a Francis turbine during load rejection and runaway. Cherny et al. [4] focused on runaway, but combined their CFD model with a 1D hydroacoustic one to include the effect of waterhammer. More recently, Fortin et al. [5] presented results of a numerical analysis of a Francis machine during runaway. Casartelli et al. concentrate on speed-no load conditions [6], which are especially critical for reversible pump turbines, as these often have an S-shaped characteristic in turbine mode. They start the calculation from speed-no load conditions and increase the guide vane opening to reach the S-shaped part of the characteristic.

This paper presents a numerical investigation of the flow phenomena in a model scale reversible pump turbine during a fast transient operation change from pump to turbine mode. It leads from a pump operating point near the instability through pump break mode to a turbine operating point.

2. SIMULATION SETUP

Simulations are carried out for a full model from spiral case to draft tube with the open source code OpenFOAM®. The pump turbine has 24 stay and guide vanes and seven runner blades. The block structured mesh contains 2.8m cells and is intended to capture the predominant flow phenomena at reasonable computational cost. Time step size varies from 2e-4 s to 1e-4 s depending on the stability of the solution. For turbulence, the k-omega-SST model is chosen to combine the advantages of the k-epsilon- and k-omega-models. The convection term is discretized using OpenFOAM®'s linearUpwind scheme, while a first order upwind scheme is used for the turbulent quantities.

Flow rate is determined by the test rig conditions. As the rotational speed of the pump turbine slows down in pump mode, the flow rate quickly reverses and reaches a high level in turbine direction. During the rest of the transient, the increase in flow rate is rather moderate. The evolution of the input parameters flow rate and rotational speed over time is calculated by a 1D transient analysis of the test rig. No coupling was implemented between 1D and 3D analysis.

The variable data for rotational speed and flow rate is passed to CFD via look-up tables. Special attention has to be paid on the choice of boundary conditions. Prescribing velocity at either the draft tube or the spiral case may lead to instability of the solution when the flow is leaving the computational domain. However, stopping the solution and restarting with altered boundary conditions may cause disturbances in the flow that travel through the machine. In this work, it was chosen to stop the simulation near the point of no inflow and adapt the boundary conditions accordingly. Thus, the flow rate is always prescribed at the respective inlet while static pressure is set for the outlet. The prescribed static pressure at the spiral case outlet corresponds to envisaged head at the beginning of the transient. To enforce consistency between the results before and after the change in boundary conditions, pressure is averaged over the draft tube inlet/outlet and the resulting mean value is prescribed as the new static pressure. An extensive comparison of boundary conditions for reduced models is found in [7].

Pressure is evaluated at specified locations throughout the machine. This includes four points in the spiral, two points between the guide vanes, one in the vaneless space between guide vanes and runner, two on pressure and suction side of the runner blades, respectively, and four in the draft tube.

3. VERIFICATION OF THE BOUNDARY CONDITIONS

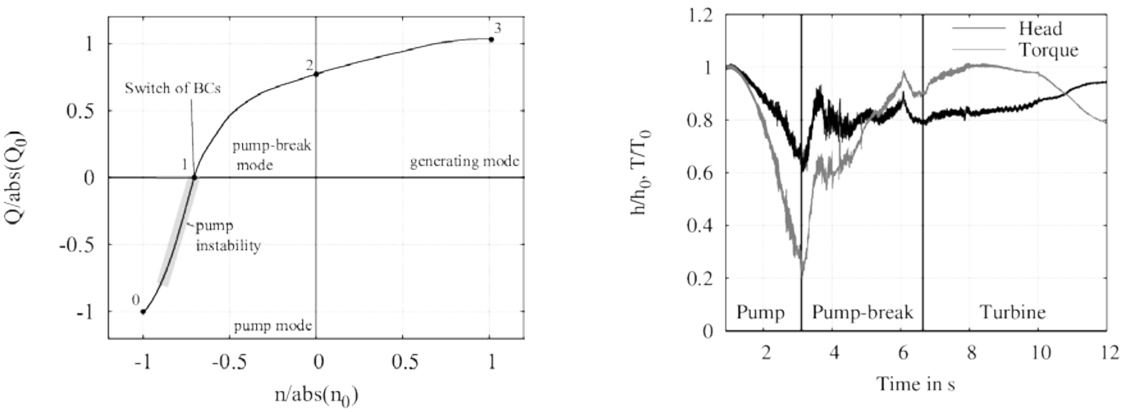
To estimate the influence of the boundary conditions on the flow through the machine, the pipe before the spiral case is modelled with a length of 1500 mm. Approaching the high pressure side of the pipe, cells are significantly enlarged to provide numerical damping for any disturbance caused by the new boundary condition after the switch to turbine inflow conditions. Results are compared to the original model.

Especially on the suction side of the blade close to the guide vanes, large fluctuations in pressure occur a certain time after the change of boundary conditions in the original model. The model with the extended pipe reduces these fluctuations. As this indicates an influence of the boundary conditions on the results, the longer version of the pipe is chosen for subsequent simulations to keep the boundary as far away as possible from the flow phenomena of interest.

An extension of the draft tube, however, shows no significant change in the results in pump mode. After the reversal of the BCs, potential disturbances are carried out of the computational domain rather than into it.

4. RESULTS OF GLOBAL QUANTITIES

Fig.1 shows the simulated transient in a 4-quadrant characteristic and results for simulated head and torque normalized to their initial values. Both variables show a similar overall behaviour during the transient. Their values drop towards the transition from pump to pump-break mode and rise again afterwards to reach almost stable values during the first part in turbine mode. Large fluctuations occur in pump mode at low flow rates as well as in pump-break mode. Additionally, head shows low frequency oscillations after some time in turbine mode. The reason for this behaviour will be investigated in more detail in the following sections.



a) 4-quadrant representation of the simulated transient

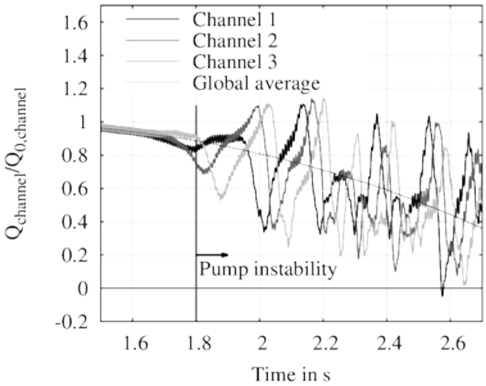
b) Simulated head and torque over time

Fig. 1 Simulated transient and results for global quantities

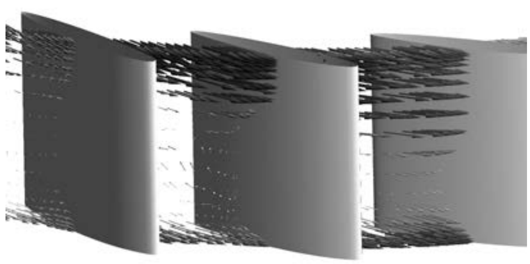
5. FLOW THROUGH STAY VANES AND GUIDE VANES

As volume flow rate decreases to small values in pump mode, large fluctuations occur in torque and pressure on the runner blades, which continue in the first half of the pump break quadrant. This is a result of stall in the guide vanes as observed e.g. in [1, 8-10]. While flow is evenly distributed between the guide vane channels in pump mode, it concentrates on single channels during the pump instability while other passages are nearly blocked. As an example, Fig. 2 shows the flow through three adjacent channels before and during the instability. Before the start of the instability, fluctuations are small and their frequency corresponds to the blade passing frequency of the runner. Thus, the fluctuations in flow rate are slightly shifted in time between the channels.

The instability adds an additional low frequency oscillation. The region of high or low flow rate in a channel can be seen to pass through the channels in the same direction as the runner rotation, with single channels reaching zero flow or even backflow while the global flow rate is still at 40% of its initial value. At the beginning, stall occurs near the bottom ring and in the middle of the channels, while flow near the head cover side remains stable. At lower flow rates, outward flow concentrates near the head cover and bottom ring meridionals, with almost no flow or slight backflow in the middle of the guide vane channels (see Fig. 2b).



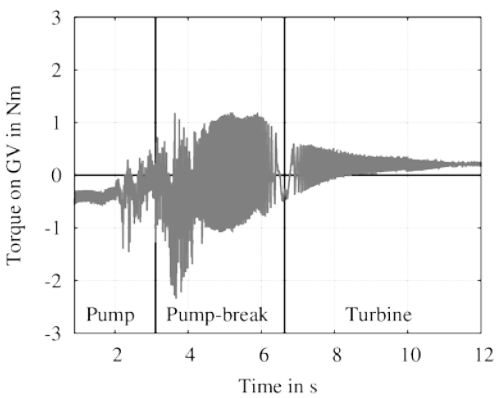
a) Flow rate through adjacent guide vane channels



b) Flow visualization through the guide vanes at t = 2.45 s

Fig 2 Flow through the guide vanes during the pump instability

In pump break mode, flow through the guide vanes is dominated by the passing of the runner blades leading to strong rotor-stator interaction.In the immediate vicinity of the runner blades, flow is forced outward (pump direction), while between the runner blades flow direction in the guide vane channels locally changes to turbine direction. This leads to large fluctuations in pressure and torque on the guide vanes. As an example, Fig. 3 presents torque on a representative guide vane over time and gives the corresponding FFT results for selected time intervals. Due to the changing rotational speed, only short intervals are considered, leading to a rather poor resolution of frequency. However, the dominant frequency at all times can clearly be identified to be approximately seven times the average runner frequency in the time interval, i.e. the blade passing frequency. Amplitudes in pump break mode are seven times higher than during the pump instability.



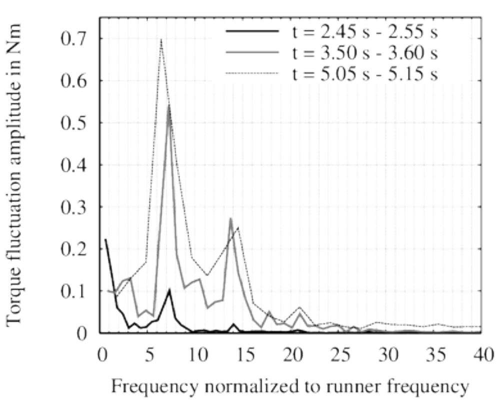
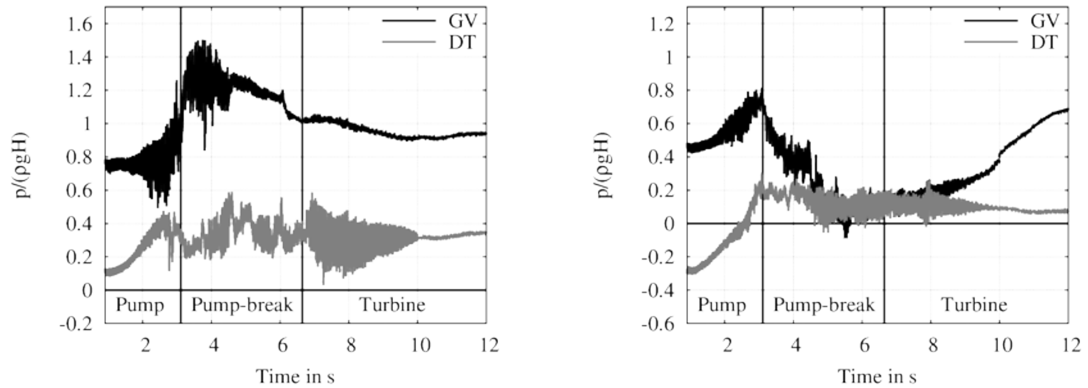


Fig 3 Torque on a guide vane: Signal over time and FFT for selected times

6. FLOW THROUGH THE RUNNER

Fig. 4 presents the pressure signal at four evaluation points in the runner. The pressure side near the guide vanes (GV) is most affected by the pump instability and shows large fluctuations around the transition from pump to pump break mode, accompanied by a rise in mean pressure. However, an anaysis using streamlines does not detect stall in the runner before 2.2 s near the hub and 2.5 s in the middle of the channel, appoximately 0.5 s after the start of the pressure fluctuations. From the hub to the middle of the channel, stall occurs on the pressure side, while closer to the shroud, it first appears on the suction side of the blades.



a) Pressure side *b) Suction side*

Fig. 4 Pressure over time at evaluation points in the runner (see Fig. 5), normalized by head in steady state pump conditions at the beginning of the transition

In pump break mode, flow from the guide vanes hits the runner blades at approximately one third of the chord length. It splits into a part that continues towards the draft tube along the pressure side of the blade, and a second part going in the opposite direction around the blade tip. This flow leads to a vortex on the suction side of the runner blades, which propagates diagonally downward and inward towards the draft tube (DT). Flow is strongly three-dimensional and causes high amplitude fluctuations at all pressure sensors in the runner. On the suction side, the difference between the two sensors disappears with decreasing rotational speed. Fig. 5 shows an example of the pressure distribution on the runner blades in pump break mode. While a steep gradient exists on the pressure side in flow direction, pressure on the suction side varies with the height of the blade, while points on the same meridional height have similar pressure.

a) Pressure side *b) Suction side*

Fig.5 Pressure distribution at t = 5s in pump break mode. Pressure difference between contour lines corresponds to 10 % of envisaged head over the machine. Pressure evaluation points are marked with dots.

In turbine mode, the pressure sensors near the draft tube are especially affected by the flow in the draft tube (see next section). High oscillations occur at the beginning of turbine mode, and a clear change in the frequency can be detected in the suction side signal.

During pump break mode and the first part of generating mode, the contributions to overall torque vary strongly between the blades. Fig. 6 compares the contributions from each blade for selected times. While at the beginning and the end of the transition, deviation from the average torque is around 2 %, it reaches nearly 15 % under transient operation conditions.

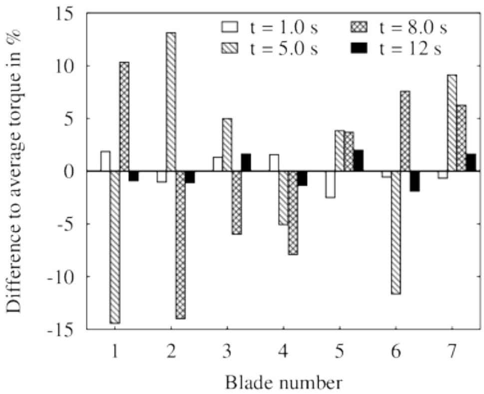


Fig. 6 Contribution to overall torque per blade

7. DRAFT TUBE FLOW

Before the start of the instability in pump mode, flow in the draft tube under the runner is straight towards the runner without significant swirl. Fig. 7 shows the axial and circumferential velocity components in the draft tube over time on a line through the draft tube aligned with the x-axis at a quarter of the runner diameter below the runner outlet. With the beginning of the pump instability, flow detaches from the outer walls and starts rotating in runner direction. While flow continues towards the runner in the center of the draft tube, a swirling downward flow develops at the walls.

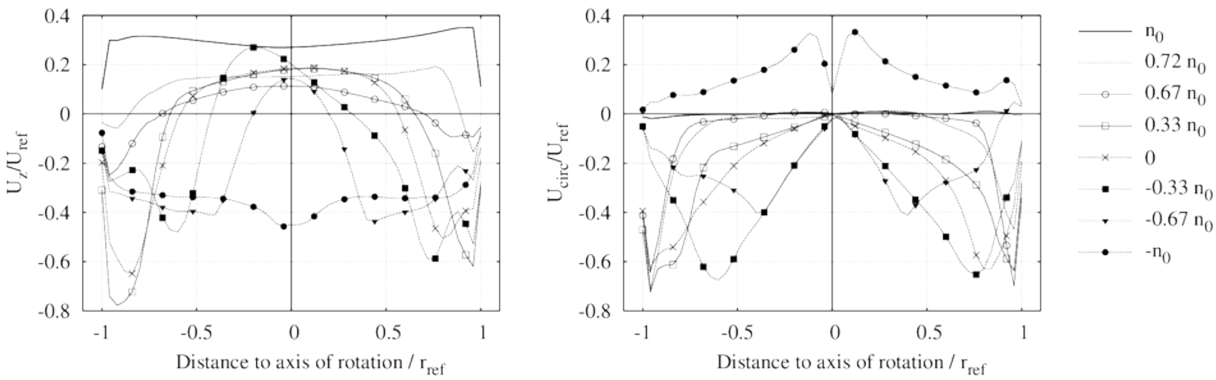


Fig. 7 Axial and circumferenial velocities in the draft tube, normalized by the circumferential velocity of the runner at its outlet at the start of the transition, i.e. at $n = n_0$. Positive circumferential velocity indicates a rotation in turbine direction. No plot markers: pump mode, with plot markers: pump break or tubine mode

This flow behaviour also affects pressure in the draft tube. Pressure near the walls is evaluated at four points located below the runner outlet at the same height at 90° from each other. The output of the first evaluation point is shown in Fig. 8. It drops in pump mode, but stays stable in pump break mode. Shortly before the runner reverses its direction of rotation, four low pressure zones appear in the runner and draft tube starting from the runner hub behind the trailing edge. With increasing rotational speed, they collapse to two and finally to a single vortex rope, as shown in Fig. 9. In consequence, the frequency of the pressure fluctuations near the wall of the draft tube changes over time. The vortices rotate in the same direction as flow in the draft tube (see Fig. 7), i.e. against the runner's direction of rotation. This is the result of a very low rotational speed combined with a relatively high volume flow rate. As the runner speeds up, the rotation slows down and reverses its direction.

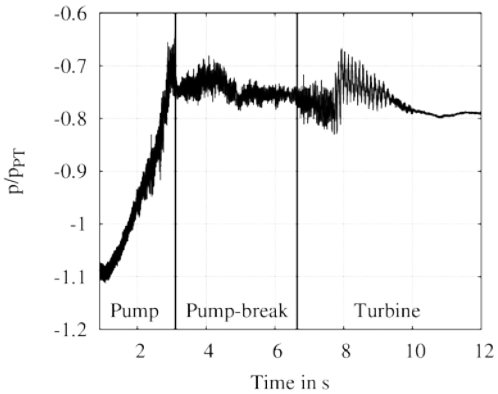


Fig.8 Pressure on the draft tube wall under the runner

a) t = 6.3 s   b) t = 7.2 s   c) t = 8.2 s

Fig. 9 Pressure isosurface under the runner at different times. Vortices move with the rotation of the fluid in the draft tube, i.e. against the runner direction.

8. CONCLUSION AND OUTLOOK

The fast transition from pump mode to generating mode exposes a reversible pump turbine to serious off-design conditions. Misaligned flow leads to stall in the guide vanes and vortices in runner and draft tube, thus causing an unconventioal loading of the components. Pressure is distributed unevenly on each blade and the contributions to overall torque vary between the blades, especially in pump break mode. Additionally, high amplitude fluctuations occur on the pressure side near the guide vanes at the reversal of flow direction and near the draft tube at the beginning of turbine mode.

The pressure results from CFD will be used as an input to a mechanical analysis of the runner to determine stresses and strains. Furthermore, an experimental campaign with different parameters will provide results for comparison with both CFD and mechanical analysis. Preliminary results from that campaign are presented in [11].

9. ACKNOWLEDGEMENTS

The authors would like to thank Nicolas Ruchonnet (Andritz Hydro SA) for providing the 1D input data. The research leading to the results published in this paper is part of the HYPERBOLE research project, granted by the European Commission (ERC/FP7-ENERGY-2013-1-Grant 608532). Part of this work was performed on the computational resource ForHLR Phase I funded by the Ministry of Science, Research and the Arts Baden-Württemberg and DFG ("Deutsche Forschungsgemeinschaft").

10. REFERENCES

[1] Kerschberger, P. and Gehrer, A.: Hydraulic development of high specific-speed pump-turbines by means of an inverse design method, numerical flow-simulation (CFD) and model testing. *IOP Conference Series: Earth and Environmental Science*. Vol. 12. 2010. 012039.

[2] Kolšek, T., J. Duhovnik, J. and Bergant, A.: Simulation of unsteady flow and runner rotation during shut-down of an axial water turbine. *Journal of Hydraulic Research,* Vol. 44/1. 2006. pp. 129-137.

[3] Li, J., Yu, J. and Wu, Y.: 3D unsteady turbulent simulations of transients of the Francis turbine. *IOP Conference Series: Earth and Environmental Science*. Vol. 12/1. 2010. 012001.

[4] Cherny, S., Chirkov D., Bannikov, D., Lapin, V., Skorospelov, V., Eshkunova, I. and Avdushenko, A.: 3D numerical simulation of transient processes in hydraulic turbines. *IOP Conference Series: Earth and Environmental Science*. Vol. 12/1. 2010. 012071.

[5] Fortin, M., Houde, S. and Deschênes, C.: Validation of simulation strategies for the flow in a model propeller turbine during a runaway event. *Proceedings of 27th symposium of hydraulic machinery and systems.* Montreal. 2014.

[6] Casartelli, E., Mangani, L., Romanelli G. and Staubli, T.: Transient Simulation of Speed-No Load Conditions With An Open-Source Based C++ Code. *Proceedings of 27th symposium of hydraulic machinery and systems*. Montreal. 2014.

[7] Côté, P., Dumas, G., Moisan, É. and Boutet-Blais, G.: Numerical investigation of the flow behavior into a Francis runner during load rejection. *Proceedings of 27th symposium of hydraulic machinery and systems.* Montreal. 2014

[8] Braun, O.: Part load flow in radial centrifugal pumps. Dissertation. Lausanne. 2009.

[9] Hasmatuchi, V.: Hydrodynamics of a Pump-Turbine Operating at Off-Design Conditions in Generating Mode. Dissertation. Lausanne. 2012.

[10] Xia, L. S., Cheng, Y. G., Zhang, X. X. and Yang, J. D.: Numerical analysis of rotating stall instabilities of a pump-turbine in pump mode. *IOP Conference Series: Earth and Environmental Science.* Vol. 22/ 3. 2014. 032020.

[11] Ruchonnet, N., Braun, O.: Reduced scale model test of pump-turbine transition. *IAHR WG Meeting on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems*. Ljubljana. 2015.