NUMERICAL SIMULATION OF THE FLOW THROUGH A KAPLAN DRAFT TUBE

Raluca Gabriela IOVĂNEL[1], Georgiana DUNCA[2], Diana Maria BUCUR[3], Valeriu Nicolae PANAITESCU[4], Michel J. CERVANTES[5]

This paper presents a comparison between steady turbulent flow simulation results in the U9 Kaplan turbine draft tube and experimental velocity and pressure measurements. Two turbulence models were tested, k-epsilon and Shear Stress Transport (SST). The results show that the k-epsilon model performs better than the SST model.

The objective is to find a correlation between the pressure measured below the runner in the draft tube cone, and the optimal guide vane angle for a given blade angle. Such correlation may allow the continuous online optimization of the cam characteristic. For this purpose, the influence of the tangential velocity on the pressure in the draft tube was specifically investigated.

Keywords: CFX, Kaplan turbine, draft tube, cam characteristic, validation

1. Introduction

With the introduction of renewable energy sources, the role of hydropower has changed. The increase in the load variations and the recurrent start-stops must be taken into consideration when designing turbines or optimizing their efficiency. Transient operation may lead to high pressure fluctuations that often account for the damages or mal-functioning of hydraulic turbines.

Kaplan turbines are operated according to the cam characteristic. In order to determine the optimum cam combination, the Winter-Kennedy method is usually used. This method is time and resources consuming and de facto not performed often. Because the cam characteristic may vary over time due to wear,

[1] PhD Student, Dept. of Hydraulics, Hydraulic Machinery and Environment Engineering, University POLITEHNICA of Bucharest, Romania, e-mail: raluca.iovanel@gmail.com
[2] Lect. PhD, Dept. of Hydraulics, Hydraulic Machinery and Environment Engineering, University POLITEHNICA of Bucharest, Romania, e-mail: georgianadunca@yahoo.co.uk (corresp. author)
[3] Lect. PhD, Dept. of Hydraulics, Hydraulic Machinery and Environment Engineering, University POLITEHNICA of Bucharest, Romania, e-mail: dmbucur@yahoo.com
[4] Prof., Dept. of Hydraulics, Hydraulic Machinery and Environment Engineering, University POLITEHNICA of Bucharest, Romania, e-mail: valeriu.panaitescu@yahoo.com
[5] Prof., Division of Fluid and Experimental Mechanics, Luleå University of Technology, Sweden, Department of Energy and Process Engineering, Norwegian University of Science and Technology Norway, e-mail: michel.cervantes@ltu.se

head variation, drifting adjustment, it must be evaluated periodically. Therefore a continuous online optimization would improve the efficiency of Kaplan units.

There are studies on Kaplan turbines operation carried out both experimentally and numerically [1, 2]. The continuously monitoring of the cam is a recent research subject at Lulea University of Technology from Lulea, Sweden. Based on experimental and numerical studies [3, 4], it has been investigated if the cam characteristic can be determined by different quantities that are easier to measure.

The present paper investigates the possibility to optimize the cam characteristic based on monitoring the pressure variation downstream the runner. For this purpose a correlation between the pressure measured below the runner in the draft tube cone and the optimal guide vane angle for a given blade angle must be found.

2. Numerical setup

A numerical model of the flow through the draft tube of the Kaplan turbine model, Porjus U9, was developed and validated in the present paper. The runner diameter is 0.5 m. During the measurements period, the operational net head was H = 7.5 m and the runner rotational speed was N = 696.3 rpm. The model was investigated at the best efficiency point (BEP) having a guide vane angle of 26º and the corresponding volume flow rate Q = 0.71 m^3/s. A more detailed description of the Porjus U9 model can be found in [5].

The turbine has been experimentally investigated for three loads by Mulu [3] and Amiri [4]. Their experimental data was used for defining the inlet boundary conditions and validating the numerical simulation.

The velocity measurements were performed using the Laser Doppler Anemometry (LDA) technique at four different sections placed at angular positions, 90º around the cone circumference. For each angular position, three different locations have been investigated: section I, II and III (fig.1). The exact location of the three sections and the pressure measurement locations in the draft tube as described in [5] are presented in Fig. 1.

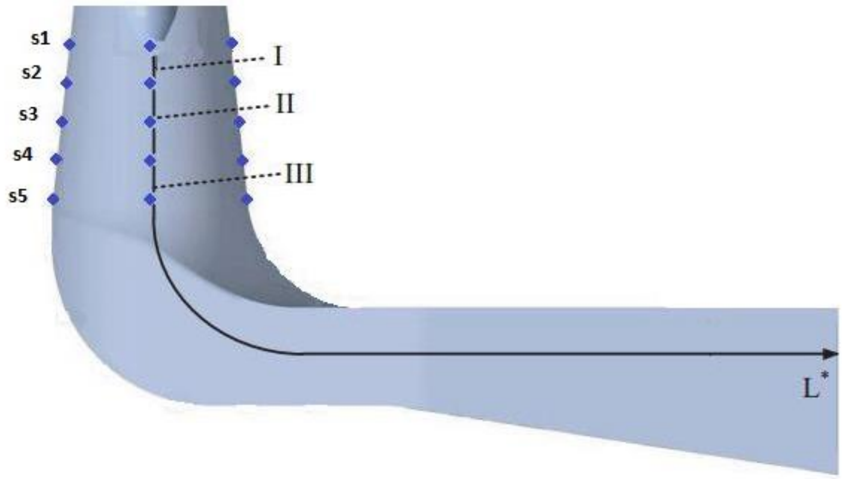


Fig 1. Pressure taps positions (s1 to s5), marked with blue dots and LDA sections I, II and III

For this study, the inlet boundary condition was the velocity specified in a local cylindrical coordinate system. The axial and tangential velocities were measured by Amiri in his experimental investigation [6]. The values for the radial component were obtained from a CFD simulation done by Mulu [3]. The wall boundary is a solid impenetrable boundary to fluid flow. The no slip boundary condition assumes that the fluid near the wall boundary moves at the same velocity as the wall. For the walls of the draft tube, the fluid velocity is zero. The hub is part of the Kaplan runner and is therefore a rotating wall.

Two different turbulence models, the k-epsilon and the SST model, are used in the present paper. The aim is to evaluate different turbulence models and boundary conditions in order to simulate accurately the flow in a draft tube at the best efficiency point (BEP).

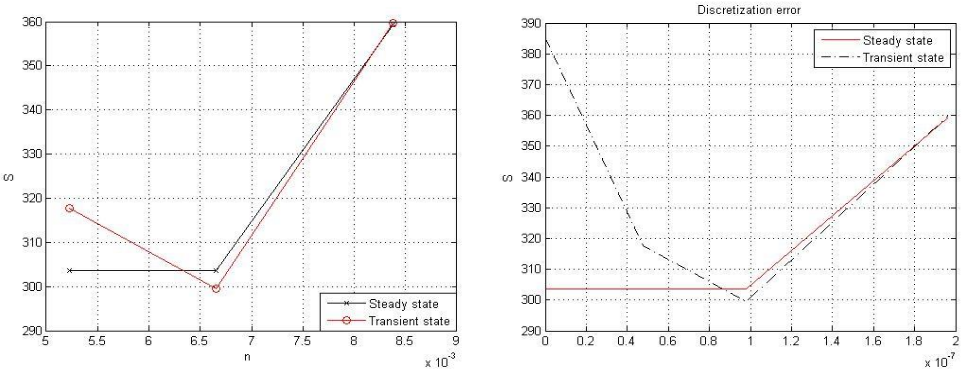
3. Mesh sensitivity analysis

First a numerical model employing the k-epsilon turbulence model was created. A mesh sensitivity analysis was performed in order to choose an adequate level of refinement that is also efficient considering the computer resources. Three different hexahedral meshes were made with sizes of 1.7, 3.4 and 7 million elements, and were studied in both steady-state and transient mode. The parameter selected to evaluate the mesh independency is the standard deviation S between the measured and simulated pressure values for each measuring section:

$$S = \sqrt{\frac{1}{4}\sum_{i=1}^{5}\left(P_i - P_m\right)^2} \tag{1}$$

where: P_m is the measured pressure, P_i is the simulated pressure, i = 1..5 for each measuring section. This parameter is plotted versus *n*, a variable dependent on the number of mesh elements, *N*:

$$n = \frac{1}{\sqrt[3]{N}} \tag{2}$$



(a) (b)

Fig. 2. The variation of the S parameter function of n (a) and the extrapolated values for n=0 (b)

The optimal value for the *S* parameter is expected to be obtained for “the infinite size mesh”. In other words, when $n = 0$; in this case the simulation is independent of the error due to the space discretization, i.e., the mesh. This optimal value is extrapolated in order to estimate the discretization error.

The discretization error is calculated as follows:

$$\varepsilon = \frac{|S_{\inf} - S_i|}{S_{\inf}} \cdot 100 \tag{3}$$

where: S_{inf} is the standard deviation calculated for the infinite size mesh ($n = 0$); S_i is the same parameter calculated for each size of mesh.

The mesh with $1.7 \cdot 10^6$ elements provides very similar results for the steady and transient case. The relative discretization error is 18.33% in the steady case and 6.5% in the transient case.

With $3.4 \cdot 10^6$ elements, the steady case performs better with 0% discretization error, compared to 22.25% estimate for the transient simulation.

The $7 \cdot 10^6$ elements mesh behaves unusual. The steady state gives the exact results as the $3.4 \cdot 10^6$ elements mesh, thus leading to the 0% discretization error. The transient simulation is even more particular. Instead of constantly decreasing, the standard deviation reaches a minimum for the $3.4 \cdot 10^6$ elements mesh and then increases up to 17.49% for the $7 \cdot 10^6$ elements mesh. This may be a sign that the transient mesh do not allow results in the asymptotic region, i.e., monotone gradient of the result.

Considering this analysis, and the limited computational resources, the mean sized model with $3.4{\cdot}10^6$ elements was selected for further analyses.

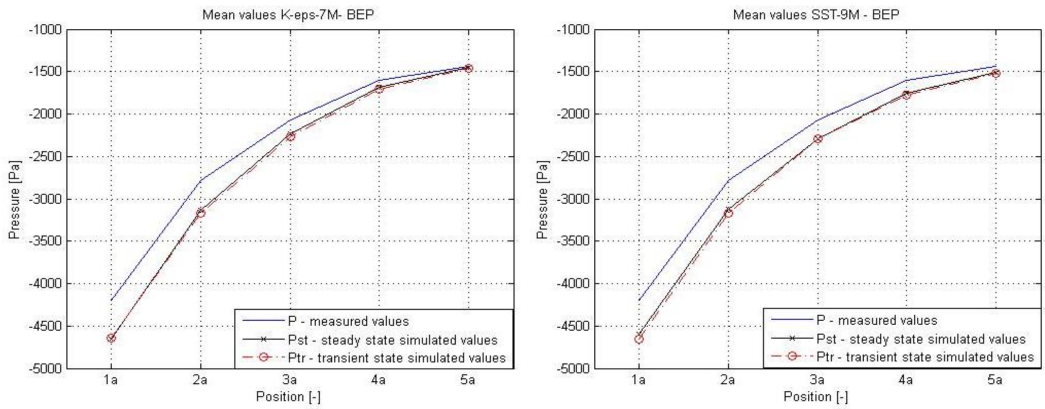
4. The influence of the turbulence model over the simulation accuracy

In order to investigate in detail the flow in the boundary layer, another model was created. This model is using the SST turbulence model instead of the k-epsilon model. Compared to the k-epsilon model which had a dimensionless distance from the wall $y^+ > 30$, the SST model had $y^+ < 2$. The mesh had approximately $9 \cdot 10^6$ elements and was finer according to the CFX mesh quality criteria.

The advantage of the SST model is that it provides an accurate prediction of flow separation problems. On the other hand the disadvantage would be that due to its increased complexity, the requirement for computational resources is higher.

For each of the two turbulence models, the simulation was run in the steady-state and transient mode.

The pressure values, the tangential and the axial velocity values obtained from the simulations were compared to the measured ones (Fig. 3-7).



(a) (b)

Fig. 3. Simulated pressure values in steady and transient state and measured values: point s1 to s5 for the (a) k-epsilon model and (b) SST model

For the k-epsilon model, the relative errors between the measured and simulated pressure values decrease in size from point s1 (closest to the hub) to s5 (Fig.3.a). The errors range from approximately 0.1% in the first section point below the runner, to 0.02% in the last point.

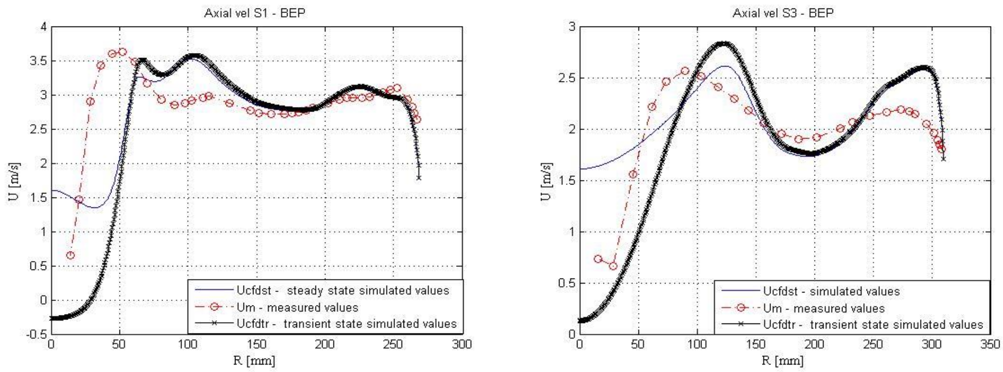
The differences between the results given by the steady-state and the transient simulations are small. Comparing them in all five measuring points, the largest difference is 0.005% and it is obtained in the first measuring point.

The errors for the SST model (Fig. 3.b) are larger but have the same variation, decreasing from point s1 to point s5. In this case, the absolute errors range from 461 Pa at the point closer to the inlet (11% of the measured value), to 80 Pa at the fifth point (5.6%). The largest value is 62 Pa at the first point, because of the hub influence due to the stationary boundary condition.

Both turbulence models have large errors when predicting the axial velocity near the runner axis (Fig. 4). At section I, the peak value given by the k-epsilon model is 3.5 m/s and it occurs at 100 mm from the axis. The measured axial velocity reaches the maximum value of 3.6 m/s closer to the center at 50 mm from the axis (Fig. 4.a). This shifted velocity peak can be noticed for both the steady-state simulation and the transient one. It is attributed to the inlet boundary conditions which promote an earlier separation on the runner cone and create a recirculation below it. A rotating inlet boundary condition is necessary to avoid such discrepancy.

The steady-state simulation overestimates the axial velocity near the center of the draft tube cone with nearly 250% when using the k-epsilon model. On the other hand, the transient model calculates a negative axial velocity showing that the flow is recirculating in the vicinity of the hub.

Closer to the shroud, the steady-state and the transient values align, the errors in section I being smaller. However, the differences between the measurements and the simulations are larger in the third section (Fig.4.b).

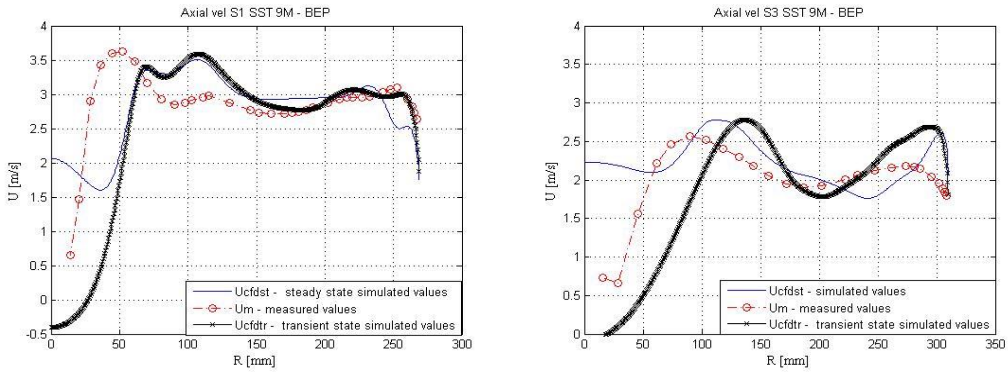


(a) (b)

Fig. 4. Axial velocity simulated values with the k-epsilon model (steady and transient state) and measured values: (a) - section I, (b) - section III.

When using the SST model (Fig. 5), the steady-state simulation overestimates the velocity values near the axis by 300%. The errors are larger than in the k-epsilon case. The transient simulation also gives negative axial velocity values in the center of the section (Fig.5.a).

As opposed to the k-epsilon model, the SST has larger errors when computing the axial velocity further from the runner, in the third measuring section (Fig.5.b).

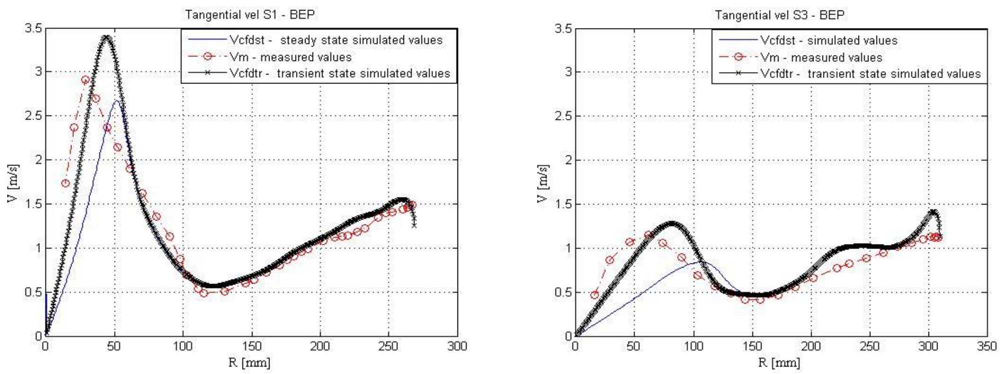


(a) (b)

Fig. 5. Axial velocity simulated values with the SST model (steady and transient state) and measured values, (a) - section I, (b) - section III.

The errors are smaller when it comes to the tangential velocity (Fig. 6). The steady-state model gives a maximum value of 2.6 m/s in the first measuring section at 50 mm from the runner axis (Fig. 6.a). The transient simulation overestimates the peak velocity at 3.4 m/s while the measured value is 2.9 at 30 mm from the axis (Fig. 6.a). Closer to the shroud the differences between the steady-state and the transient mode are very small and the values follow the measured values.

In the third section, the errors increase closer to the shroud (Fig. 5.b). As in the first section, the steady-state values differ from the transient ones near the center of the section but further from the axis, they resemble the measurements.

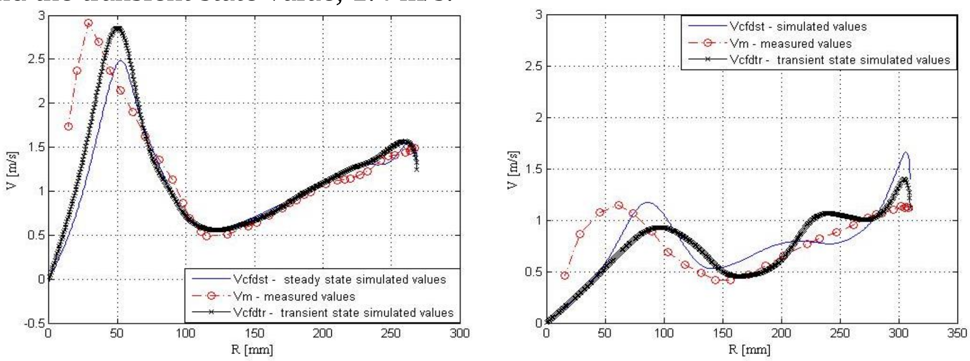


(a) (b)

Fig. 6. Tangential velocity simulated values with the k-epsilon model (steady and transient state) and measured values: (a) - section I, (b) - section III.

In the first section the SST model seems more accurate when calculating the velocity near the runner axis (Fig. 7.a). The peak velocity is 2.8 m/s given by the transient simulation and 2.4 m/s in the steady-state simulation. Both peaks are found at 50 mm from the runner axis as opposed to the measured maximum value which is closer to the axis.

In the third section the errors are larger than in the first one (Fig. 7.b). The maximum tangential velocity is now obtained near the shroud. Both the steady-state and the transient state simulation overestimate the peak values. The maximum measured value is 1.14 m/s, whereas the steady-state value is 1.66 m/s and the transient state value, 1.4 m/s.

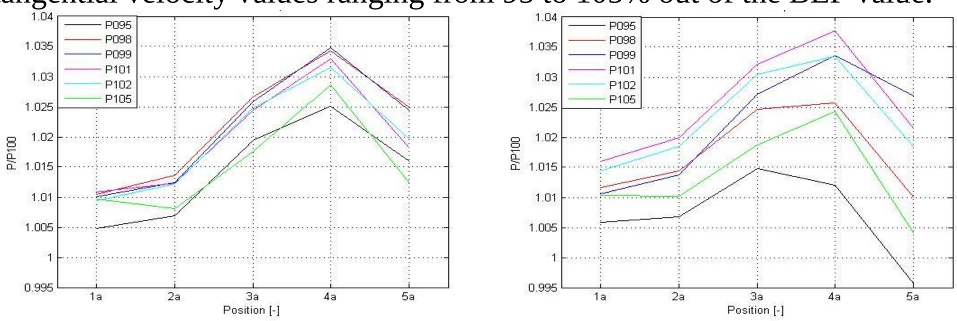


(a) (b)

Fig.7. Tangential velocity simulated values with the SST model (steady and transient state) and measured values: (a) - section I, (b) - section III.

1. Swirl influence

In order to find a possible correlation between the pressure measured below the runner in the conical part of the draft tube and the optimal guide vane angle, the influence of the tangential velocity was investigated. The inlet boundary condition was modified and several simulations were run for different tangential velocity values ranging from 95 to 105% out of the BEP value.



(a) (b)

Fig. 8. Pressure ratio variation function of the tangential velocity: (a) steady state and (b) transient state

If the tangential velocity at the inlet boundary of the draft tube cone is different from the BEP value, the pressure below the runner is expected to be larger. However, the pressure values obtained with 95% of the tangential velocity case are the closest to the values corresponding to the 100% case. The simulations using 99 and 101% of the tangential velocities show the largest difference when compared to the original case.

The fact that the curves intersect (Fig.8.a and 8.b) shows that the same pressure values are expected to be obtained for different percentages of the tangential velocity. This problem is more obvious in the steady-state simulation.

In consequence, a correlation between the tangential velocity and the pressure values below the runner cannot be clearly determined. A transient simulation with a more accurate model and boundary conditions might provide more information concerning this hypothesis. Therefore, further investigations should be done, before drawing a conclusion considering this aspect.

2. Conclusions and discussions

In this paper, the flow in the draft tube of a Kaplan turbine was investigated at the best efficiency point using numerical simulations.

The numerical model was developed and validated using pressure and velocity measurements performed in the draft tube cone. Two different turbulence models, k-epsilon and SST, were evaluated and different boundary conditions were tested. For each model, different sizes of mesh were created. The influence of the mesh discretization was also investigated.

Simulated pressure and velocity values are presented together with the measurements and compared in order to validate the model. According to the mesh sensitivity analysis and discretization errors, two numerical models were selected: one model with $3.4{\cdot}10^6$ elements and one SST model with $9{\cdot}10^6$ elements.

The k-epsilon turbulence model gives better results than the SST model. The pressure, the axial and tangential velocity values are closer to the measured values in the case.

The influence of the tangential velocity over the draft tube pressure was studied to determine if an online optimization of the cam characteristic is possible. A conclusion cannot be drawn at this stage of the research.

Further research is recommended in order to better investigate this theory. A stage simulation could give more accurate results and show a more clear correlation between the tangential velocity at the runner outlet and the pressure measured in the draft tube cone.

Acknowledgements

The work has been funded by the Sectoral Operational Programme Human Resources Development 2007-2013 of the Ministry of European Funds through the Financial Agreement POSDRU 187/1.5/S/155420 and by the Executive Agency for Higher Education, Research, Development and Innovation, PN-II-PT-PCCA-2013-4, ECOTURB project.

R E F E R E N C E S

[1]. *Y. Wu, S. Liu, H. Dou, S. Wu and T. Chen,* "Numerical prediction and similarity study of pressure fluctuation in a prototype Kaplan turbine and the model turbine", in Computers & Fluids, **vol. 56**, March 2012, pp. 128–142.

[2]. *A. Javadi and H. Nilsson*, "Unsteady numerical simulation of the flow in the U9 Kaplan turbine model", in IOP Conf. Series: Earth and Environmental Science. 27th IAHR Symposium on Hydraulic Machinery and Systems, Montreal, Canada. **vol. 22**, Art. no. 022001, 2014.

[3]. *B. G. Mulu*, An experimental and numerical investigation of a Kaplan turbine model, PhD Thesis, Luleå University of Technology, 2012.

[4]. *K. Amiri*, An experimental investigation of flow in a Kaplan runner: steady state and transient, Licentiate Thesis, Luleå University of Technology, 2014.

[5]. *B. G. Mulu and M. J. Cervantes*, "Experimental investigation of a Kaplan model with LD", in Proceedings of the 33rd IAHR congress, Vancouver, Canada, 2009, pp. 155–162.

[6]. *K. Amiri, B. G. Mulu, M. J. Cervantes and M. Raisee*, "Load variation effects on the pressure fluctuations exerted on a Kaplan turbine runner", in IOP Conf. Series: Earth and Environmental Science. 27th IAHR Symposium on Hydraulic Machinery and Systems, Montreal, Canada. **vol. 22**, Art. no. 032005, 2014.