The numerical simulation of the rope vortex creation and the possibilities of its control

P Szulc¹, A Machalski¹, J Skrzypacz¹, D Blonski¹  
¹ Wroclaw University of Science and Technology, Faculty of Mechanical and Power Engineering, Wybraneze Wyspianskiego 27, 50-370 Wroclaw, Poland  

E-mail: przemyslaw.szulc@pwr.edu.pl

Abstract. The rope vortex is a phenomena which can be observed in many hydraulic machines, especially turbines and pump–turbines. Its creation is based on a non uniform velocity profile characterized by high value of the circumferential component. This phenomena limits the range of operating conditions, which results in reduction of the energy production. The article presents the flow analysis of the hydraulic Francis turbine, where the rope vortex was observed. The structure of the vortex was identify and some solutions were analysed to prevent or reduce the size of multiphase flow.

1. Introduction  

Hydro power engineering is a widely available source of energy. The potential for water storage and simplicity in fast coupling with the grid are main advantages of mentioned stations. The utilization of hydropower and any other forms of renewable energy are being intensively developed because of their ability to regenerate and avoiding deterioration of the environment. As an example renewable sources reduce carbon footprint. However, those sources are characterized by the unpredictable nature which leads to the adverse influence on the national electrical grid. From all available renewable sources of energy hydropower is the most predictable and allows to create foreseeable production plans. Moreover, the output power can be controlled in a wide range of loads and its energy is full power capacity is available on request. The opportunity of flexible utilization of renewable sources of energy is a crucial problem, which leads to problems of its smooth integration into the existing power grid. It is obvious that the diurnal demand for electrical energy is changing, therefore, renewable sources should have a possibility to be regulated, which sometimes leads to operational difficulties connected with energy storage. When the water turbine is utilize for frequency modulation or peak regulation the water engine must operate in not optimal condition, far from the best efficiency point. One of potential application of water turbine is the Francis turbine. This type of turbines is very sensitive to the low load conditions which strongly limits its operation range.

An off–designed operation regime of Francise turbine is often characterized by high value of the tangential velocity which leaves the impeller. It leads to the creation of a very strong swirl in the draft tube [1]. This causes flow instabilities and as a result, an enormous stagnate zone could be observed. Moreover, when the vortex is strong, cavitation can be observed. These deteriorated flow structures in the draft tube leads to efficiency loss and high gradient pressure pulsation [2, 3]. It also leads to frequency vibration could be observed. In most situations the influence of the draft tube on the power generation diminishes, which is responsible for even 35–40% of energy recovery [4].
In course of last decades the phenomena of rope vortex was a subject of numerical simulations and real investigations. Sidel et al. [5] presented the flow analyses in the draft tube in course of the turbine regulation, and define the correlation between the partial load operation and the range of the rope vortex. The application of particle image velocimetry was made to understand the flow structure in function of the deepness of the turbine regulation. As main conclusions, the strong dependence of tangential component of velocity and partial load operation [6] were drawn. Trivedi et al. [7] noticed the asymmetrical flow profile leaving the impeller after observation of rapidly pressure change in the draft tube transducers.

Also the numerical simulation of the rope vortex flow structure was a subject of analysis. In recent years, CFD methods have allowed to obtain a lot of valuable information about off–designed operation of the Francis water engines. Sick et al. [8] realized simulations with the application of the RMS (Reynolds stress) model. Obtained results were close to the real parameters. In [9] the unsteady simulation of the rope vortex with the application of the Reynolds Averaged Navier–Stokes model was presented. The consistency with experimental data was equal to 13%. In this case k–ε viscous model was used. Zhang et al. [10] applied the same turbulence model, but with logarithmic wall function. Vu et al. [11] claim that an applied turbulence model does not conform with the results of the experiment tests when the regulation of the turbine increases. The investigations were made as steady and unsteady, with k–ε viscous model. Yaras and Grosvenor [12] made the simulations of a strongly whirling vortex. The researchers utilized different turbulence models. It is important to emphasize that the k–ω SST model provided the worst prediction. This same investigation approach was assumed by Dhiman et al. [13]. What is interesting, a two–equation standard, RNG, realizable k–ε, SST k–ω and the Reynolds stress model are not proper for the analysed flow. In [14] the swirling character of the flow was not fully reflected by the RANS models. Thus, the unsteady statistical turbulence models were applied in [14, 15] and [16], mostly as the hybrid of RANS and LES.

The occurrence of the rope vortex phenomena is difficult to predict during the designed process. When its presence was observed, for example by means of vibrations, there are some common control methods which allows to reduce its negative impact. The control methods could be divided into active and passive. The active control techniques allow to adjust for different operating conditions of the turbine, so there is a possibility for the regulation of this method. In the best efficiency point, where the rope vortex is not present, device responsible for control can be turned off. The passive control techniques allow to reduce the rope range and there is no possibility of alteration the intensity of operation. In some cases it can decrease efficiency at optimal operation conditions.

The passive control methods often introduces additional hydraulic losses in draft tube and efficiency reduction. Their application is based on the reduction of circumferential velocity or elimination of stagnation zone by means of solid bodies. In [17] the authors compared three methods of control rope vortex: applications of runner cone, damping gate and flow deflector – figure 1. Obtained results show that application of damping gate can reduce the circumferential velocity and adding flow deflector cause reduction of vortex eccentricity. Both solutions eliminate draft tube vortex rope.

**Figure 1.** Passive control methods: a) runner cone, b) with damping gate, c) with flow deflector [17].
Figure 2. Passive and active control methods: a) fins [19], b) air addition [19], c) water addition.

Another passive methods are fins mounted on the draft tube wall (Fig. 2a) [19]. They could be an effective means of vortex reduction [18]. The authors performed experimental tests to assess the effect of the application of this method. The drawbacks of the fins are the increase of the head losses and tendency to cavitation erosion. The extension of the fins, could be defined as flow splitter, which is another, potential method. They have similar advantages and disadvantages like fins.

Other passive methods are: application of the impeller hub extension [18], Figure 1a. It is also possible to use a grooved runner, [20]. Proposed methods do not decrease the efficiency as much as fins, but could be responsible for decreasing of lateral force acting on the shaft.

Rope vortex control could be achieved by addition of air or water in a place where vortex is observed. This methods reduce the amplitude of pressure fluctuation. Air or water could be injected in several locations – spiral casing, guide vane, the runner hub, band and snorkel located near impeller cone. The advantage of this solution – the possibility of regulation, must be correlated with the operational problems and amount of additional water or air in the installation. More techniques of the cavitation rope vortex control could be found in [21–25].

The literature review shows that the rope vortex is the main reason for instabilities in the flow structure. This leads to vibration in the draft tube. To reduce the negative impact, the passive or active method could be applied. It is important to emphasize that there is no single solution that could eliminate this negative impact, so each case must be analysed separately. The main purpose of the presented paper is to investigate the flow structures of the Francis water turbine in a nominal and partial load operation regime and flow comparison of chosen control method.

2. Object of research
During the short term operation of Francis turbine located in a hydroelectric power plant situated close to the dam, the rope vortex phenomena was observed. The water turbine is working mainly in the run on river regime, but it can also work with the possibility of water storage operation. To analyse the flow structure in the object, a standard, vertical axis construction was taken into consideration. The unit was equipped with a fixed impeller and had adjustable guide vanes. Main geometrical and operational parameters of the analysed object were presented in Table 1.

| No. | Name                  | Symbol | Value |
|-----|-----------------------|--------|-------|
| 1   | Flow                  | \(Q\) (m\(^3\)/s) | 6.75  |
| 2   | Total head            | \(H\) (m)    | 38    |
| 3   | Rotational speed      | \(n\) (rpm) | 500   |
| 4   | Power consumption     | \(P\) (kW)   | 2307  |
|     |                       | \(d_2\) (mm) | 1050  |
| 2   | Specific speed        | \(n_{SN}\)  | 296   |
3. Numerical model

The CFD Ansys FLUENT software was used for analysing the fluid flow through Francis turbine. The geometrical model reflected the real geometry of the machine. The main purpose of conducted research was to obtain the pressure and velocity fields of the flow as a function of geometrical changes of the turbine (guide vane, draft tube). The tests were conducted as steady RANS and unsteady URANS for the determined boundary conditions. The medium was defined as clear water of viscosity $\mu_{\text{H}_2\text{O}} = 0.001003$ Pa and density $\rho_{\text{H}_2\text{O}} = 998.2$ kg/m$^3$. The models of turbulence such: $k$–$\omega$ SST, $k$–$\epsilon$ Realizable and DES were used for simulations. Convergence criterion was assumed to be $0.0001$ for all equations. The unstructured mesh was generate by means of the Ansys ICEM CFD. Tetra elements were made in the centre of the flow body and the prisms layers were applied near the walls (first layers were 0.025 mm). The total number of nodes was assumed in accordance with the test of the mesh independence and it was approximately equal to 50 million. The computational mesh and 3D model were presented in Figure 3. The numerical model consisted of four fluid domains: volute casing, guide vane, impeller and draft tube. The boundary conditions were defined as the pressure at the inlet and the outlet.

![Figure 3. The geometrical model of the turbine dedicated for numerical calculations and computational mesh.](image)

The conducted simulations were divided into two parts.

In the first one, the flow in the water turbine was identified. It was realized for two guide vane positions: $\alpha_n = 91\%$ an $\alpha_n = 61.4\%$. $\alpha_n$ is understood as a percentage of opening. In course of the real experiment for the first point, turbine operates correctly. In second, low frequency vibrations were recorded. For both points the validation was made based on comparison between the turbine actual power and power obtained as a result of numerical simulations. The deviations with respect to CFD are merely 1.36% and 1.51%.

The second step of analysis was focused on testing the effectiveness of several rope vortex reduction control techniques. The pressure and velocity fields of flow in a function of geometrical changes of the turbine were identified. Following passive methods were analysed: the change of the hub shape, impeller cone, fins, flow deflector, additional pipe (with rotation), diffuser cone inside the draft tube.
4. Results of the flow simulation in current geometrical structure of the turbine

The utilization of consider water turbine was characterized by strong ‘hits’, which could be noticed during the reduction of opening the guide vane $\alpha_n$. To identify the reason of its occurrence the appropriate CFD simulation were made. Two operation point were analysed $\alpha_n = 90\%$ and $\alpha_n = 61.4\%$.

In the design point the flow in all passages of the turbine is uniform. All elements operate correctly. The efficiency obtained from CFD simulation was equal $\eta_{\text{CFD}} = 92\%$. Due to the possibility of rope vortex occurrence the flow in the draft tube was meticulously analysed. In Figure 4 the distribution of the velocity (a) and pressure (b) in the draft tube was presented. The flow visualization could be characterized as proper and uniform. There is no stagnation zones. The pressure field indicate that draft tube operates correctly. We can observe pressure recovery that takes place uniformly along the length of analysed diffuser. The velocity vectors are also uniformly distributed. This means that the runner is correctly designed for this operation point. The small separation in the end of the drat tube was observed, which is the reason of presents of the wall in the outlet chamber. To sum up the angle and the length of the draft tube was assumed properly.

![Figure 4](image4.png)

**Figure 4.** The distribution of the velocity: a) and pressure, b) in draft tube, $\alpha_n = 90\%$.

In the point of operation $\alpha_n = 61.4\%$, the flow in the turbine could be described as irregular. An analysis of the flow structure in the turbine volute element indicates the uniform distribution of liquid particles in all its parts. Also guide vanes and impeller work correctly. Lack of separation and clear stagnation zones can be noticed. While analysing the operation of the guide vane and runner it should be pointed that they smoothly changes liquid flow direction, the medium does not separate from the sides of vanes. The conducted unsteady simulations did not show any differences in the distribution of the flow parameters.

The results of numerical simulations in the draft tube were presented in figure 5. While analyse the obtained results it could be pointed that the liquid leaves the impeller with a big value of circumferential component of absolute velocity, which is the reason of the partial load operation. Such strongly swirling flow entering the draft tube causes large stagnation zones and is a source of reverse flows. The liquid close to the wall of the draft tube is characterized by the highest velocity. In the centre part of the diffuser there is wide zone of a decreased velocity which does not take part in the flow. Mentioned zone fills almost $1/2 - 1/3$ of pipe volume. Such area leads to the reduction of the draft tube active cross–section. It means that it cannot work correctly and the recovery of kinetic energy is also reduce.

![Figure 5](image5.png)

**Figure 5.** The distribution of the velocity: a) and pressure, b) in draft tube, $\alpha_n = 61.4\%$. 
Figure 6. The cavitation rope vortex in analysed turbine, $\alpha_n = 61.4\%$.

Figure 7. Pressure distribution in analysed object: a) model; perpendicular to rotation axis cross-section in the draft tube refer to impeller; b) $z=+1.0$ m, c) $z=+1.5$ m, d) $z=+2.0$ m, e) $z=+3.0$ m, f) longitudinal cross-section, $\alpha_n = 61.4\%$.

The analysis of the static pressure distribution indicate inappropriate operation of diffuser. The areas of large pressure gradients were noticed. Low pressure region resembles a sine wave in its shape. This area indicate that the occurrence of rope vortex is possible. Figure 6 presents low pressure iso-surface. The peripheral components of the velocity is inversely proportional to the radius, therefore, pressure will also depend on it. If, on the specific diameter the value of pressure equals the saturation pressure, a cavitation core or a cavitation rope vortex will appear. After analysing the results of numerical simulations one can notice a wide range swirl of a spiral structure. The observed rope vortex covers $1/2 - 1/3$ length of the draft tube.

In Figure 7, pressure distribution of analysed object was presented. Obtained views will be treated as reference when assessing the proposed modifications.
5. Comparison of control techniques

The utilization of Francis turbine requires wide range of regulation, thus the application of the rope vortex control method is desirable. Because of the local conditions of the turbine only passive, selected methods were taken into consideration. Follow techniques were investigated:

- flow deflector proposed in [17],
- additional diffusor cone inside draft tube,
- additional motion cylinder inside draft tube, connected to the hub,
- modification of impeller hub shape,
- fins.

All models were defined as three dimensional objects and proper simulations were made. Due to the fact, that rope vortex was observed only during partial load, guide vane opening $\alpha_n = 61.4\%$ was analysed.

In figures below results of conducted research was presented. For all cases, pressure distributions were treat as a main source of information of the impact on the rope vortex strength reduction. In each geometry a 3D model was also presented.

The first step of the research was to identify the flow structure inside the draft tube in off–design point while a flow deflector was introduced. The deflector consist of two flat blade rows. They are $b = 80$ mm wide and $l = 500$ mm long with an angle of divergence equals $\theta = 9^\circ$. The blades were connected to the draft tube with rods.

Figure 8 shows the pressure distributions in longitudinal and perpendicular cross-sections when flow deflector was applied. It should be noticed that the rope vortex was not extinguished. Moreover, the range of impact is higher than for the reference model. Obtained result are opposite to those presented in [17]. The deflector strengthen the vortex in analysed structure. The flow structure

Figure 8. Pressure distribution in analysed object: a) model; perpendicular to rotation axis cross-section in the draft tube refer to impeller: b) $z=+1.0$ m, c) $z=+1.5$ m, d) $z=+2.0$ m, e) $z=+3.0$ m, f) longitudinal cross-section, $\alpha_n = 61.4\%$. 
**Figure 9.** Pressure distribution in analysed object: a) model; perpendicular to rotation axis cross-section in the draft tube refer to impeller: b) \( z = +1.0 \) m, c) \( z = +1.5 \) m, d) \( z = +2.0 \) m, e) \( z = +3.0 \) m, f) longitudinal cross-section, \( \alpha_n = 61.4\% \).

**Figure 10.** Pressure distribution in analysed object: a) model; perpendicular to rotation axis cross-section in the draft tube refer to impeller: b) \( z = +1.0 \) m, c) \( z = +1.5 \) m, d) \( z = +2.0 \) m, e) \( z = +3.0 \) m, f) longitudinal cross-section, \( \alpha_n = 61.4\% \).
inside the draft tube in strongly irregular. The wide stagnation zone in the centre part of the draft tube was noticed. The observed rope vortex fills 2/3 – 3/4 length of the tube.

The second approach for draft tube modification was assumed to be the application of additional diffuser located behind the hub. From geometrical point of view the cone was characterized by the angle equal to $\theta = 11^\circ$, inlet diameter $d_{in} = 300$ mm and length $l = 1300$ mm. The main idea lying behind proposed solution was to prevent the radial flow and create the separate sections in the draft tube.

In Figure 9 pressure distributions in longitudinal and perpendicular cross–sections were presented for the case with additional cone. Conducted simulations show that the flow profile in the draft tube is not uniform. However, the stagnation zone was reduced. Adding a cone does not allow to exchange the energy between both section of draft tube. Near the inlet cross–section of additional diffuser the spiral ring with lower pressure could be pointed. Proposed solution reduces the rope vortex. Unfortunately, it is not very effective.

The third variant of draft tube modification was defined as additional motion cylinder inside draft tube, connected to the hub of the impeller. The geometry was characterized by the diameter $d = 200$ mm and length $l = 1000$ mm. The pipe is fixed with the runner, and rotates with it. The main reason of the application of proposed solution was to fill the inner part of the tube, were the pressure is the lowest.

In Figure 10 pressure distributions in longitudinal and perpendicular cross–sections were presented. In this case additional, rotating pipe was added. As the result of conducted simulation, the flow structure was defined. Realized investigations show that the flow in the draft tube is regular, the zone of stagnation was strongly reduced. The diffuser operates correctly, only small flow deteriorations are visible. Proposed solution reduces the rope vortex strength and this reduction is sufficient. The main disadvantage of this method is that it could lead to increased radial force acting on the turbine shaft. This fact does not simply allow to use this solution in real object. A structural simulation should be performed before implementing this solution.

**Figure 11.** Pressure distribution in analysed object: a) model; perpendicular to rotation axis cross–section in the draft tube refer to impeller: b) $z=+1.0$ m, c) $z=+1.5$ m, d) $z=+2.0$ m, e) $z=+3.0$ m, f longitudinal cross–section, $\alpha_n = 61.4\%$. 
Figure 12. Pressure distribution in analysed object: a) model; perpendicular to rotation axis cross-section in the draft tube refer to impeller: b) \( z=+1.0 \) m, c) \( z=+1.5 \) m, d) \( z=+2.0 \) m, e) \( z=+3.0 \) m, f) longitudinal cross-section, \( \alpha_n = 61.4\% \).

The fourth method of rope vortex control was defined as modification of impeller hub shape (crown). The geometry was characterized by the addition of the cone with the filet instead of the flat cup shape. This element is fixed with the runner, and rotates with it.

In Figure 11 pressure distributions in longitudinal and perpendicular cross–sections were presented for the application of modified impeller cup. The simulation show that the flow in the draft tube is quite regular. The stagnation zone was decreased. Proposed method is interesting because it does not affect turbine efficiency. Proposed solution reduces the rope vortex strength, yet it is not really effective.

The last proposition for rope vortex control are fins in the side walls of the draft tube. The geometry of fins was characterized by the angle of divergence equals \( \theta = 5.5^\circ \), refer to the rotation axis, length \( l = 1500 \) mm, high \( b = 200 \) mm and number of \( z_{\text{fin}} = 6 \). At the beginning the fins were chamfered. The main reason of the application of proposed geometrical change was to reduce the circumferential velocity component in the draft tube.

In Figure 12 pressure distributions in longitudinal and perpendicular cross–sections were presented for the fins application. According to numerical simulation results, the flow structure was can be described as regular. The draft tube operates properly. The zone of stagnation was strongly reduced. Only small flow deteriorations could be noticed near the sides of the fins, which was caused by the flat shape of the blade. It could be considered to replace the flat plane of fins and utilize the semi–oval one. The main disadvantage of this method of vortex control could be the incensement of the vibration level in the draft tube. Proposed solution meets all requirements.

6. Conclusion

Results of the experimental, unsteady and steady numerical simulations of the Francis water turbine were presented in this paper. In accordance with the conducted analyses and obtained results it could
be concluded that the flow image in the draft tube during turbine operation in the off designed condition is highly deteriorated. The liquid leaves the impeller with high tangential component of velocity and such profile feeds the draft tube. Finally, the zone of low pressure occurs in the central part of the pipe immediately after the impeller. The whole core of the draft tube fills the area of back flow – dead zone. It leads to creation of the rope vortex, and its appearance causes the reduction of the turbine regulation range.

The control methods of the rope vortex were analysed. From all possibilities of passive techniques only the application of fins seems to be the most reasonable. Simulations show that, in this case, the vortex was strongly reduced and the draft tube operates correctly. Also solution with additional pipe seems to give accurate results. Opposite to the outcomes described in [17] the flow deflector does not impact properly on the flow character in the draft tube.

References
[1] Dorfler P, Sick M, Coutu A, 2013 Flow-Induced Pulsation and Vibration in Hydroelectric Machinery Engineer’s Guidebook for Planning, Design and Troubleshooting
[2] Bronstein L A , Gierman A I, Goldin W E, Konovalov I I, Robuk I I, Umikov I N, Cogin I A, Sirro I I 1971 Hydro turbines hand book (Maschinostroenie)
[3] Michalowski S, Plutecki J 1975 Energetyka Wodna (Warsaw)
[4] Gubin M F 1970 Draft tubes of water power stations (Energiya)
[5] Seidel U, Mende C, Hu’ bner B, et al. 2014 Dynamic loads in Francis runners and their impact on fatigue life IOP Conf Ser Earth Environ Sci 22 32054
[6] Goyal R 2018 Gandhi BK and Cervantes MJ. PIV measurements in Francis turbine – a review and application to transient operations Renew Sust Energ Rev 81 2976
[7] Trivedi C, Cervantes MJ, Gandhi BK, et al. 2014 Experimental investigations of transient pressure variations in a high head model Francis turbine during start–up and shutdown J Hydrodyn 26 277
[8] Sick M, Dorfler P, Michler W, Saillalberger M, and Lohmberg A 2004 Investigation of the draft tube vortex in a pump-turbine 22nd IAHR Symposiumon Hydraulic Machinery and Systems
[9] Ciocan G D, Iliescu M S, Vu T C, Nennemann B, Avellan F 2007 Experimental study and numerical simulation of the fluidt draft tube rotating vortex ASME J. Fluids Eng. 129(2) 146
[10] Zhang R K, Mao F, Wu J Z, Chen S Y, Wu Y L, Liu S H 2009 Characteristics and control of the draft–tube flow in part–load Francis turbine ASME J. Fluids Eng. 131(2) 021101
[11] Vu T C, Devals C, Zhang Y, Nennemann B, Guibault F 2011, Steady and unsteady flow computation in an elbow draft tube with experimental validation Int. J. Fluid Mach. Syst. 4(1) 85
[12] Yaras M I, Grosvernor A D 2003 Evaluation of One– and Two– Equation Low–Re Turbulence Models. Part I—Axisymmetric Separating and Swirling Flow Int. J. Numer. Methods Fluids 42(12) 1293
[13] Dhiman S, Foroutan H, Yavuzkurt S 2011 Simulation of flow through conical diffusers with and without inlet swirl using CFD ASMEJSME– KSME Joint Fluids Engineering Conference AJK2011–03005
[14] Ruprecht A, Helmrich T, Aschenbrenner T, Scherer T 2002 Simulation of vortex rope in a turbine draft tube 21st IAHR Symposium on Hydraulic Machinery and Systems
[15] Paik J, Sotiropoulos F, Sale M 2005 Numerical simulation of swirling flow in complex hydroturbine draft tube using unsteady statistical turbulence models J. Hydraulic Eng. 131(6) 441
[16] Foroutan H, Yavuzkurt S 2014 Flow in the simplified draft tube of a Francis turbine operating at partial load—part 1: simulation of the vortex rope Journal of Applied Mechanics 81 61010
[17] Feng J 2014 Investigation on pressure fluctuation in a Francis turbine with improvement measures IOP Conference Series: Earth and Environmental Science 22 (2014) 032006
[18] Nishi M, Wang X M, Yoshida K, Takahashi T, and Tsukamoto T 1996 An Experimental Study
on Fins, Their Role in Control of the Draft Tube Surging \textit{Hydraulic Machinery and Cavitation, E. Cabrera, V. Espert, and F. Martinez, eds.} 905

[19] Ayli E 2019 Cavitation in Hydraulic Turbines \textit{International Journal of Heat and Technology} \textbf{37}(1) 334

[20] Sanol T, Maekawa M, Okamoto N, Yano H, and Miyagawa K, 2012, Investigation of Flow atter Downstream of Spiral Grooved Runner Cone in Pump–Turbine \textit{26th IAHR Symposium on Hydraulic Machinery and Systems} (Beijing, China)

[21] Bhan S, Codrington J B, Mielke H 1988 Reduction of Francis turbine draft tube surges \textit{5th International Symposium on Hydro Power Fluid Machinery} (Chicago, IL)

[22] Qian Z D, Li W, Huai W X, Wu Y L 2012 The effect of the runner cone design on pressure oscillation characteristics in a Francis hydraulic turbine \textit{Proc. IMechE A J. Power Energy} \textbf{226}(1) 137

[23] Zhang R K, Mao F, Wu J Z, Chen S Y, Wu Y L, Liu S H 2006 Characteristics and control of the draft–tube flow in part–load Francis turbine \textit{ASME J. Fluids Eng.} \textbf{131}(2) 021101

[24] Susan-Resiga R, Muntean S, Hasmatsuchi V, Anton I, Avellan F 2010 Analysis and prevention of vortex breakdown in the simplified discharge cone of a Francis turbine \textit{ASME J. Fluids Eng.} \textbf{132}(5) 051102

[25] Foroutan H, Yavuzkurt S 2014 Flow in the simplified draft tube of a Francis turbine operating at partial load-part 2: control of the vortex rope \textit{Journal of Applied Mechanics} \textbf{81} 061011

Acknowledgements

Calculations have been made using resources provided by Wroclaw Centre for Networking and Supercomputing (http://wcss.pl), grant No. 444/2017.

Nomenclature

\begin{itemize}
\item $Q$ (m$^3$/s) – flow
\item $H$ (m) – total head
\item $n$ (rpm) – rotational speed
\item $P$ (kW) – power consumption
\item $d$ (mm) – diameter
\item $d_2$ (mm) – external diameter of the impeller
\item $d_a$ (mm) – inlet diameter
\item $b_0$ (mm) – width of adjustable guide vane
\item $z_{\text{imp}}$ – number of impeller blades
\item $z_{\text{gv}}$ – number of guide vanes
\item $z_{\text{fin}}$ – number of fins
\item $n_{\text{SN}}$ – specific speed
\item $a_{\text{n}}$ – sets of guide vane
\item $\eta_{\text{CFD}}$ – efficiency calculated by means of CFD simulation
\item $b$ (mm) – width,
\item $l$ (mm) – length of location equals
\item $\theta$ ($^\circ$) – angle of location
\item $z$ – component of location on axis $z$, 0 in the centre of impeller
\end{itemize}