Study of the velocity distribution influence upon the pressure pulsations in draft tube model of hydro-turbine

V Sonin¹, A Ustimenko¹, P Kuibin², I Litvinov²,³ and S Shtork²,³

¹ Leningradsky Metallichesky Zavod, Power Machines-LMZ, St.-Petersburg, Russia
² Institute of Thermophysics SB RAS, Novosibirsk, Russia
³ Novosibirsk State University, Novosibirsk, Russia

Abstract. One of the mechanisms of generation of powerful pressure pulsations in the circuit of the turbine is a precessing vortex core, formed behind the runner at the operation points with partial or forced loads, when the flow has significant residual swirl. To study periodic pressure pulsations behind the runner the authors of this paper use approaches of experimental modeling and methods of computational fluid dynamics. The influence of velocity distributions at the output of the hydro turbine runner on pressure pulsations was studied based on analysis of the existing and possible velocity distributions in hydraulic turbines and selection of the distribution in the extended range. Preliminary numerical calculations have showed that the velocity distribution can be modeled without reproduction of the entire geometry of the circuit, using a combination of two blade cascades of the rotor and stator. Experimental verification of numerical results was carried out in an air bench, using the method of 3D-printing for fabrication of the blade cascades and the geometry of the draft tube of hydraulic turbine. Measurements of the velocity field at the input to a draft tube cone and registration of pressure pulsations due to precessing vortex core have allowed building correlations between the velocity distribution character and the amplitude-frequency characteristics of the pulsations.

1. Introduction
The crucial problem of hydropower engineering is expanding the range of sustainable and reliable hydro power plant (HPP) operation, including non-optimal operation points of hydro turbines. The notorious problem, resulting in reduced efficiency and safety of the hydraulic unit operation, is formation of a precessing vortex core in the cone of the hydro turbine draft tube (DT). The rotating vortex core generates strong periodic pressure pulsations, and due to the presence of a bend in DT design these pulsations are transmitted over the entire water head of the hydro turbine [1,2]. It is therefore necessary to develop methods that allow predicting the development of this effect at off-design operation points of the hydraulic unit.

To date, the main tools for calculating and optimizing the geometry of the turbine draft tubes are methods of computer modeling (CFD) [3–6]; but the ambiguity of selection of a turbulence model and large velocity gradients in the swirling flow do not give due confidence in the results of the calculations. This implies additional verification of numerical codes on the basis of empirical data. However, the detailed experimental study of full-scale turbines is impossible or extremely difficult and expensive. The solution to this problem lies in using scaled down laboratory models. Such units are available in almost all leading organizations, dealing with designing, production and development of hydraulic turbines. With the wide development of numerical methods, it is not the possibility of
Transfer of results from the model unit to the full-scale one that is brought to the fore, but the opportunity to verify numerical methods in a large range of parameters. Such a range can be achieved at experimental modeling, where models are simplified, but, nevertheless, reproduce all the features of the full-scale circuit of the hydraulic turbine. To obtain reliable empirical data, it is also important to avoid perturbations in the flow, i.e. to use contactless methods. In recent years, modern experimental methods of velocity measurement, such as LDA and PIV, have provided new information about physical processes in draft tube models [7–9]. In addition works [10–12] show the possibility of physical modeling without geometric repetition of the full turbine duct (spiral case, stay vane and guide vane).

Preliminary analysis of existing and possible velocity distributions in hydraulic turbines have allowed selecting the typical velocity distributions at the runner output. The optimum operation point is characterized by an almost uniform distribution of the axial velocity component. The circumferential component has a small value at the periphery and is virtually absent near the hub. The part load operation points are characterized by substantially uneven distribution of both circumferential and axial components of the velocity vector. Both components increase from the hub to the periphery.

In this work, the input velocity distribution close to the distribution behind the real turbine is created by means of a system of two blade cascades. One of the cascades is still (the stator) and functions as a spiral case and the turbine guide vane (it is evenly flown over with a volumetric flow rate \( Q \), \( \text{m}^3/\text{h} \)). The second cascade (the rotor) forcibly rotates with a frequency \( n \), rpm, and is an analog to the runner of hydraulic turbine. The authors of this work have developed a methodology to design the cascades that provide the specified velocity distribution with good accuracy. Thus, by choosing the values of \( Q \) and \( n \) and designing the appropriate blade cascades, different operation points of the turbine have been simulated.

Another significant simplification in experimental modeling is the replacement of the water medium by air [13,14]. This allows assembling the unit, avoiding the problem of reliability of junction sealing, and quickly changing the geometry of the draft tube. The structural elements of the test sections are not subjected to strong dynamic impact that in turn allows using the available plastic materials. In our work we used the method of rapid prototyping, based on 3D-printing, to create the model of draft tube and blade cascades. Thus, this approach opens up new opportunities for a model experiment, aimed at obtaining arrays of experimental data useful both for optimizing the circuit of hydraulic turbines and verifying the numerical codes.

2. Methods

At the first stage we selected the required simulated velocity distributions, corresponding to both the optimal operation points of hydro turbines (geometry \# 4 and 9 in table 1) and part load operation points (geometry \# 14 in table 1). Based on the selected distributions, the shape and the number of blades of the stationary and rotating cascade were determined as a rough approximation (see Fig. 1a). Various options complying with turbines of Kaplan type (turbine with adjustable blades – AB30) and Francis type (radial-axial turbine – RA115) with different hub-tip ratio have been considered.

Variants of swirlers differed in distributions over the radius of tangential and axial components of the velocity vector just behind the edges of the rotating runner. Based on the results of calculations of spatial turbulent water flow in the swirler, the actual velocity distributions at the runner output in the control section were built (see Fig. 1c). According to the analysis of the flow calculation results the initial geometry was corrected to provide a more exact match of the actual velocities, specified in the initial design.

To calculate the flow through the stator and rotor the ANSYS Fluent software was used. The calculations were performed in a stationary periodic statement for the cascades with a diameter of 100 mm. The calculation domain (see Fig. 1b) consisted of a guide vane segment and a segment of the
runner. The computational grid was block-structured (built in the software ANSYS TurboGrid), and its size was about 250 thousand cells per segment.

| # | Type of turbine | Hub-tip ratio | The number of stator blades | The number of rotor blades | $Q_{n}$, m$^3$/h | $n$, rpm | The angle of inclination of the output edge of the stator | The angle of inclination of the output edge of the rotor |
|---|----------------|--------------|-----------------------------|---------------------------|-----------------|--------|------------------------------------------------|--------------------------------------------------|
| 4 | AB30           | 0.4          | 10                          | 6                         | 175             | 2500   | -23.5°                                          | 54.15°                                          |
| 9 | RA115          | 0.2          | 10                          | 5                         | 175             | 2432   | -17.8°                                          | 58.7°                                           |
| 14| RA115          | 0.2          | 9                           | 8                         | 180             | 2703   | -13.3°                                          | 36.2°                                           |

The working medium was water. The calculations were carried out with the turbulence model “k-$\varepsilon$ Realizable”, utilizing the pressure-based coupled solver, and Least Squares Cell Based method for the gradient. The schemes used for convection terms and pressure were QUICK and PRESTO, respectively. In the input section of the guide vane segment the constant axial velocity was determined for a given flow rate. In the output section of the runner, the pressure distribution was put in accordance with the condition of radial equilibrium. Conditions for the turbulence at the input and output boundaries were set using the intensity (3%) and the hydraulic diameter (0.1 m). The flow in the runner segment was calculated in a rotating coordinate system (rotation frequency is shown in table 1). For the exchange of data between the rotor and the stator the “mixing plane” model was used. Figure 1c shows the types of boundary conditions, used in the input and output sections of the segments.

At the next step, using rapid prototyping with 3-D printer CubeX Duo, we made the physical model of draft tube and 15 pairs of swirlers with the geometry of blades, obtained in the calculations. Figure 2 shows the geometry of the experimental section of the aerodynamic stand. The air flow in a model draft tube was supplied using a blower MT-M1C ($Q_{max}$=550 m$^3$/h, $P$=0.4 bar). The air flowrate was measured with the ultrasonic flowmeter IRVIS-Ultra RS-4 and regulated by the frequency converter Danfoss FC-51. The original software was used to control the flow rate in the mode with feedback "flowmeter – frequency converter" through RS-485 Modbus interface.

The frequency of the runner rotation was set by a servomotor. The servomotor SPSH-3410 provided exact setting of the runner rotation frequency in the range from 0 to 4000 rpm. All bench control was
carried out using a computer. Thus, with the help of special software, it was possible to maintain the specified flow regime parameters within the required time with an error of 1.5 and 0.5% for setting a flow rate $Q$ and the rotation frequency of the runner $n$, respectively.

![Flow calculations over a stator-rotor combination: (a) general view of the blade cascades, (b) area of flow calculation, (c) projection of the computational domain.]

Figure 1. Flow calculations over a stator-rotor combination: (a) general view of the blade cascades, (b) area of flow calculation, (c) projection of the computational domain.

The flow before the stator was aligned with the system of grids. Additionally, more uniform flow distribution became possible due to the confuser. This configuration was proven based on several iterations, including measurements and calculations. Finally, in front of the stator it was possible to achieve a rather uniform distribution of axial velocity in both radial and angular directions with a turbulence level below 5%.

In the cone of the draft tube model, part of a wall of 35 mm width was replaced by rectangular glass window for the access of the measuring beams of the Laser-Doppler anemometer (LDA). To measure the velocity distributions we used the LDA "LAD 06-i", designed and manufactured by the Institute of Thermophysics SB RAS [15]. For LDA it was necessary to provide the required number of
tracers in the flow, so the airflow was seeded with paraffin oil aerosol with an average particle diameter of 1-3 microns.

Figure 2. Air working part for modeling the flow in the draft tube at different operation points of the turbine operation. Inset: (a) cone of draft tube with blade cascade of AB30 type, (b) cone of draft tube with blade cascade of RA115 type.

To measure pressure pulsations the following acoustic sensors were used: type 2250 sound level meter Bruel&Kjaer and measurement microphones Behringer ECM8000. At a removable cone without optical access the holes with 12 mm diameter were made for acoustic sensors to enter flush with the wall. Signals were digitalized using ADC L-Card E-440. In our experiments, the ADC sampling frequency was 2 kHz, and the signal recording time varied from 15 to 135 seconds. The acquired signals were stored on the PC hard disk for further processing, which included the procedure of the digital Fourier transform.

3. Comparison of average inlet velocity profiles

At the first stage, for all combinations of the blade cascades (table 1), the designed and calculated inlet velocity profiles (at a distance of 4 mm from the back edge of the rotor) were compared with the ones, measured in the experiment (Figure 2). It can be seen that for geometries # 4 of AB30 type and # 9 of RA115 type there is a good agreement between the velocity profiles, obtained by all methods. For geometry # 14 there is some deviation of the measured data from the calculated and designed velocity profiles. Thus, the technology of rapid prototyping has given satisfactory results in achieving the required velocity distributions, at the same time enabling rapid testing with much lower costs.

The next step was to identify the operation points with maximum pressure pulsations, caused by the precessing vortex core, which, as mentioned above, are considered to be potentially dangerous in the hydropower industry. These operation points arise at a shift from the calculated flow \( Q_n \) to the region of smaller flow rates, corresponding to partial loads of the turbine.

4. Determining operation points with maximum flow pulsations

To determine the operation points with maximum pressure pulsations the flow rate was varied in the range from 0.4 \( Q_n \) to \( Q_n \) with a step of 0.05 \( Q_n \) for fixed \( n \), which specified in table 1. Pressure pulsations obtained using a calibrated sound level meter B&K placed in the half-height cross-section of the cone were recorded. The obtained mean-square pressure pulsations for each value of \( Q/Q_n \) are shown in Fig. 4a. From the data it is seen that for all geometries a local extremum arises (labeled as "max"). This extremum occurs due to formation of the precessing vortex core in the draft tube cone. This is illustrated by the spectra of differential signal (Fig. 4b) calculated using signals from two
identical microphones Behringer ECM8000 installed opposite each other at the same axial plane as the B&K microphone. Procedure of the differential signal analysis enables extracting coherent part of the pressure pulsations related to the precessing motion of the vortex core [16]. Owing to the distinct feature of the asymmetrical flow structure induced by the PVC, the signals from the oppositely installed pressure probes are registered out of phase. So that the pulsation mode appearing due to the PVC is doubled in the differential signal but the pulsation components related with other sources are normally registered by the probes in phase and thus suppressed.

Figure 3. Averaged velocity distributions: (a) axial and (b) tangential (geometries #4, 9 and 14 from top to bottom).
Figure 4. Pulsation flow characteristics with variation of the \( Q/ Q_n \) parameter: dependence of the RMS pressure fluctuations on the airflow (a), amplitude spectrum (in arbitrary units) of differential signals of two pressure sensors for three flow rates – 0.4 \( Q_n \), \( Q_n \) and \( Q = Q_{(\text{max})} \) (b) (geometries #4, 9 and 14 from top to bottom).
The dominant frequency \( f^* (Q) \) (different from the frequency \( n \)) appears on the spectra of pressure pulsations for geometries # 4, 9, and 14 (Fig. 4b). For geometry # 4, starting from the flow rate \( Q = 0.4 Q_n \), the peak with frequency \( f^* = 17 \) Hz emerges; further at the rate of 0.55 \( Q_n \) the peak amplitude of \( f^* \) reaches maximum. When approaching the design condition \( (Q_n) \) the “rope” frequency \( f^* \) disappears from the spectrum completely. For geometry # 9 there is a similar evolution of the dominant frequency that occurs from the flow rate 0.4 \( Q_n \), reaches maximum in the regime of 0.5 \( Q_n \), and then completely disappears from the spectrum for the optimum mode \( Q_n \), indicating the complete absence of the vortex core precession. For geometry # 14, the dominant frequency \( f^* \) in the spectrum exists for all regimes of flow rate, reaching maximum at point 0.85 \( Q_n \). Information about all of the obtained dominant frequencies and operation points with maximum pulsations is collected in table 2.

### Table 2. Operating parameters at maximum pulsations.

| #   | Flow rate in the mode of maximum pulsations “max”, \( Q/Q_n \) | \( f^*, \) Hz |
|-----|-------------------------------------------------------------|-------------|
| 4   | 0.55                                                        | 17          |
| 9   | 0.50                                                        | 19          |
| 14  | 0.85                                                        | 33          |

Now consider the distribution of the averaged velocity for the three geometries and different air flow rates from 0.4 \( Q_n \) to \( Q_n \) (Fig. 5). The distributions were measured along the x-axis (see inset in Fig. 2). As it is shown by the axial distribution of averaged velocity component (Fig. 5, a) for all geometries there is a substantial reforming of the flow. Small central area of recirculation (flow moving towards the rotor) \((x = 0.2 D)\) is formed in all cases and for all operation points and is related to flow separation behind the bluff body. A wide area of recirculation in the profile center \((x = 0.6 D)\), according to some authors, is accompanied by the formation of a vortex core [17–21]. For geometries # 4 and 9 the wide recirculation area disappears completely when the flow rate increases to 0.7 \( Q_n \). For geometry # 14 the recirculation region attenuates only at approaching \( Q_n \). This is confirmed by the spectra of pressure pulsations for geometry # 14, where the region of the vortex core formation is shifted closer to the design condition (see Fig. 4a).

The distributions of the tangential velocity (Fig. 5b) show that the intensity of the flow swirling increases at low flow rates, and at higher flow rates (approaching the optimum operation mode) it is significantly reduced.

### 5. Conclusions

Based on preliminary numerical calculations we have designed and then adjusted the geometry of the blade cascades (stator and rotor) that allows simulating typical velocity distributions at the draft tube input. With the use of rapid prototyping technology, the blade cascades of swirlers and the model draft tube have been manufactured that served to experimental verification of numerical results using air as the working medium. In the course of the experiment it was shown that for the given parameters \( Q \) and \( n \) the appropriately designed blade cascades of the stator and rotor allow simulating the required operation points of the hydro turbine. During detection of the mode with maximum pressure pulsations it was shown that such a regime occurs in the area of low flow rates. The analysis of signals of pressure pulsations allowed demonstrating the development of the phenomenon of vortex core precession in the DT cone. The occurrence of non-stationarity in the form of a vortex core is also expressed in the reform of the distribution of the axial velocity, when the area of the reverse flow is greatly extended for such regimes.
Figure 5. Distribution of averaged velocity at variation of $Q/Q_n$ parameter: axial component (a), tangential component (b) (geometries # 4, 9 and 14 from top to bottom).

Acknowledgments
Experimental study of distribution of velocity and pressure pulsations made by I. Litvinov and S. Shtork was financially supported by the Russian Science Foundation (grant No. 14-29-00203)
References

[1] Falvey H T 1971 Draft Tube Surges—A Review of Present Knowledge and an Annotated Bibliography (Bureau of Reclamation)

[2] Wang F, Li X, Ma J, Yang M and Zhu Y 2009 Experimental Investigation of Characteristic Frequency in Unsteady Hydraulic Behaviour of a Large Hydraulic Turbine J. Hydrodyn. Ser B 21 12–9

[3] Xiao Y, Wang Z, Zhang J and Luo Y 2014 Numerical predictions of pressure pulses in a Francis pump turbine with misaligned guide vanes J. Hydrodyn. Ser B 26 250–6

[4] Zuo Z, Liu S, Liu D, Qin D and Wu Y 2015 Numerical analyses of pressure fluctuations induced by interblade vortices in a model Francis turbine J. Hydrodyn. Ser B 27 513–21

[5] Minakov A V, Platonov D V, Dekterev A A, Sentyabov A V and Zakharov A V 2015 The analysis of unsteady flow structure and low frequency pressure pulsations in the high-head Francis turbines Int. J. Heat Fluid Flow 53 183–94

[6] Krappel T, Kuhlmann H, Kirschner O, Ruprecht A and Riedelbauch S 2015 Validation of an IDDES-type turbulence model and application to a Francis pump turbine flow simulation in comparison with experimental results Int. J. Heat Fluid Flow

[7] Susan-Resiga R, Dan Ciocan G, Anton I and Avellan F 2006 Analysis of the Swirling Flow Downstream a Francis Turbine Runner J. Fluids Eng. 128 177

[8] Mulu B G, Jonsson P P and Cervantes M J 2012 Experimental investigation of a Kaplan draft tube – Part I: Best efficiency point Appl. Energy 93 695–706

[9] Tridon S, Barre S, Ciocan G D and Tomas L 2010 Experimental analysis of the swirling flow in a Francis turbine draft tube: Focus on radial velocity component determination Eur. J. Mech. - BFluids 29 321–35

[10] Bosioc A, Tanasa C, Susan-Resiga R and Muntean S 2009 2D LDV Measurements of Swirling Flow in a Simplified Draft Tube Conference on Modelling Fluid Flow (CMFF’09)

[11] Chen C-K, Nicolet C, Yonezawa K, Farhat M, Avellan F, Miyazawa K and Tsujimoto Y 2010 Experimental Study and Numerical Simulation of Cavity Oscillation in a Diffuser with Swirling Flow Int. J. Fluid Mach. Syst. 3 80–90

[12] Susan-Resiga R, Muntean S, Tanasa C and Bosioc A Hydrodynamic Design and Analysis of a Swirling Flow Generator Proceedings of the 4th German – Romanian Workshop on Turbomachinery Hydrodynamics (GRoWTH), June 12-15, 2008, Stuttgart, Germany.

[13] Povh N 1974 Aerodynamic experiment in machinery (Leningrad)

[14] Nishi M, Yano M and Miyagawa K 2007 A preliminary study on the swirling flow in a conical diffuser with jet issued at the center of the inlet Sci. Bull. “Politehnica” Univ. Timisoara Trans. Mech. 52(66) 198–202

[15] Institute of optical electronic information technology [electronic resource] 2D Laser Doppler Anemometry (LDA)

[16] Vinokurov A P, Shtork S I, and Alekseenko S V 2014 Gas phase effect on characteristics of a precessing vortex in the axisymmetric hydrodynamic chamber Thermophysics and Aeromechanics 21 771-774

[17] Alekseenko S V., Kukin P A, Okulov V L and Shtork S I 1999 Helical vortices in swirl flow J. Fluid Mech. 382 195–243

[18] Syred N 2006 A review of oscillation mechanisms and the role of the precessing vortex core (PVC) in swirl combustion systems Prog. Energy Combust. Sci. 32 93–161

[19] Shtork S I 2005 Coherent Helical Structures in Swirl Flows Tech. Phys. Lett. 31 660

[20] Shtork S I, Cala C E and Fernandes E C 2007 Experimental characterization of rotating flow field in a model vortex burner Exp. Therm. Fluid Sci. 31 779–88

[21] Litvinov I V., Shtork S I, Kukin P A, Alekseenko S V. and Hanjalic K 2013 Experimental study and analytical reconstruction of precessing vortex in a tangential swirler Int. J. Heat Fluid Flow 42 251–64