Numerical and experimental study of low-frequency pressure pulsations in hydraulic units with Francis turbine

D. Platonov, A. Minakov, D. Dekterev, A. Sentyabov, A. Dekterev

Siberian Federal University, Russia 660041 Krasnoyarsk, Svobodny ave., 79
Institute of Thermophysics SB RAS, Russia, 630090 Novosibirsk, Acad. Lavrentyev ave., 1

platonov-08@yandex.ru, tov-andrey@yandex.ru

Abstract. The paper presents the numerical simulation method of three-dimensional turbulent flows in the hydraulic turbine. This technique was verified by means of experimental data obtained on a water model of the Francis turbines. An aerodynamic stand, which is a miniature copy of the real hydraulic turbine, was designed. A series of experiments have been carried out on this stand and the corresponding calculations were performed. The dependence of the velocity and pressure pulsations profiles for different operation regimes are presented.

1. Introduction

The development of hydraulic turbo machinery have long reached a mature stage. At design conditions, the turbines usually perform with high efficiency and reliability. However, at part loads and in transient regimes the stability of the system can seriously be impaired leading to a decrease in efficiency, mechanical damage and system failure. The intrinsic unsteadiness at suboptimal conditions leads often to vortex breakdown and precessing helical vortices in form of a single or twin rope behind the runner, which cause intense flow pulsations and vibrations of the turbine structure that pose a serious risk of the system reliability and safety. These problems have long been in focus of the experimental researches and simplified analytics, but the full three-dimensional time-dependent dynamics has remained beyond the most advanced diagnostics techniques. Computer simulations have been seen as a potential tool for capturing the details of the instabilities and the vortical and turbulence structures. Such information can be obtained by direct and large-eddy simulations (DNS, LES) over a range of important scales. However, the accuracy of DNS and LES depends much on the numerical resolution. For complex configurations the Reynolds-averaged Navier-Stokes (RANS) approach, being computationally less demanding, has been considered as a more rational option for industrial purposes. However, because if its empirical contents, the RANS have not been regarded as a trustful research tool, and moreover, their credibility depends much on the choice of the turbulence model to close the time- or ensemble-averaged conservation equations.

Despite the extensive literature (e.g. Doerfler [1], Alligne et al. [2]), comprehensive experimental data for hydro turbines are still insufficient, apart from mean velocity and second- moment turbulence statistics measured at selected locations in laboratory models of some popular turbine types [3, 4]. On the other hand, there are still many open questions regarding the practically relevant numerical simulation of the flow in turbines of realistic configuration. Because of typically very high Re numbers, DNS and even LES, requiring enormous numerical mesh and computer resources, are usually beyond the capabilities of the designing. The URANS or hybrid RANS/LES methods remains the available approaches of the simulation.

Here the first challenge is related to selecting the appropriate RANS model that can reproduce satisfactorily the unsteady flow and the dynamics of at least the dominant large-scale vortex structures subjected to strong swirls, curvature and adverse pressure gradients in complex geometric shapes. The popular linear eddy-viscosity modes (*k*-ε, *k*-ω, and their many "improved" variants) lack of the sufficient physics and have shown to perform poorly in describing such flows.

2. Mathematical model and numerical method

Describing the flows in the flow path of water turbines one have to face several problems. The first challenge is related to modeling of turbulence in channels of complex geometric shapes and with strong swirl flow. At the same time for engineering calculations turbulence models, not only quite accurate describing of the average fields, but also large-scale flow pulsations is required. Widely spread in engineering calculations *k*-ε and *k*-ω turbulence models poorly describe such flows. To improve the modeling of turbulent swirling flows scientists are trying to either modify existing URANS turbulence models or use techniques, which resolve large-scale turbulent eddies (LES, DES).

The precession of the vortex core can accompany the swirling flow in hydraulic turbines. For modeling such phenomenon it is necessary to use non stationary simulation methods such as the method of large-eddy simulation. However, it requires a very detailed grid, especially near the walls. At the same time, RANS models are quite economical and describe the boundary layers well. To combine the advantages of these approaches the method of detached eddy simulation (DES) was proposed in [5].

In the simulation of hydraulic turbines it is necessary to consider the rotation of the runner. There are many approaches for modeling flows with rotating bodies, they include: dynamic grid, sliding mesh, moving mesh and method based on a moving reference frame. The most common and simple way to model the runner rotation is to use a rotating reference frame. The rotating coordinate system allows simulating flow in an approximation in which the runner is fixed and swirl flows pass through it. In this paper, modeling of the runner rotation was performed with rotated reference frame for runner zone. Numerous test calculations show the correctness of this approach, both in the description of integral characteristics of flow and pulsation [4, 6-7].

Below, the basic equations of the mathematical models in the rotating coordinate system are presented. The continuity equation (conservation of mass):

$$\nabla \mathbf{v} = 0 .$$

Momentum equation (conservation of momentum) in a rotating frame for relative velocities:

$$\frac{\partial \rho \mathbf{v}}{\partial t} + \nabla\left(\rho \mathbf{v}\mathbf{v}\right) = -\nabla p + \nabla(\boldsymbol{\tau}^{m} + \boldsymbol{\tau}^{t}) - \rho(2\boldsymbol{\Omega} \times \mathbf{v} + \boldsymbol{\Omega} \times (\boldsymbol{\Omega} \times \mathbf{r})) ,$$

where: $\mathbf{v}$ - velocity vector, $\boldsymbol{\tau}$ - stress tensor, Ω - angular velocity of runner rotation, p - static pressure, ρ - density. On the right side of the equation of conservation of momentum Coriolis' force and the centrifugal force are written.

Two turbulence modeling approach were used. The first one is second order closure, i.e. Reynolds stress model (RSM) and the second one is detached eddy simulation (DES).

One should say a few words about the numerical approach. Discretization of the transport equation was carried out by the control volume method on unstructured grids. Velocity and pressure fields for incompressible fluid were obtained by means of the SIMPLE procedure. For the approximation of the convective terms in the equation for momentum components, second order scheme was used. For the approximation of convective terms in the equations for turbulent characteristics, up-wind scheme was used.

3. Experiment

An experimental study was performed on an open aerodynamic stand. The configuration considered is a 4:1 scaled-down laboratory model of a Francis turbine (RO-230) of an LMZ (Leningrad Metal

Industry). Air supply in the draft tube model was carried out using a high performance air blower with the maximum flow rate $Q_{\max}$ =170 m^3/h and the excess pressure of Δp=0.4 bar. An ultrasonic flow meter «IRVIS-Ultra» and a frequency converter of the blower with feedback controlled air flow rate were used. For capturing the LDA signal the air flow was seeded with tracers produced by a standard-type generator of the paraffin oil aerosol with a mean particle diameter of 1-3 μm. Figure 1 provides a photo of the experimental turbine model. The draft tube model, runner, spiral case and guide vanes geometry were provided by the company LMZ and were manufactured using the technology of 3D printing.

Figure 1: Photo of the experimental stand

Servo drive SPSh10-3410 ensured precise rotation frequency of the runner in the range from 0 to 3000 rpm. A computer ensured the control of the experiment. Using the original software, it was possible to maintain the flow regime with an accuracy of 1.5% and 0.5% of the flow rate and the runner rotation frequency respectively.

The experiments included measurements of velocity fields in the conical part of the draft tube using a system of two-component LDA “LAD-06i” [8], and measurements of the pressure pulsations with special acoustic sensors. The acoustic sensors were based on high precision microphones attached with tiny tips for capturing pressure pulsations at the local points [9]. Two of these sensors were used to conduct point-to-point measurements [9]. The perturbation mode that corresponds to the precessing vortex core was identified by analysing the signal from the sensors located in diametrically opposite angular positions. Note that the possibility of using well-proven acoustic technology is another advantage of air model stand.

Pressure fluctuations were recorded using a microphone Bruel & Kjaer Type 2250. The signals from the microphones were digitized by an analog-to-digital converter L-card E-440. The experiments were conducted at the runner rotation frequency from f = 0 to 2000 rpm with a step 500 rpm and a flow rate, varing in the range from Q =100 to 150 m^3/h with a step of 50 m^3/h. At each point, the signal detected by the acoustic sensors was digitized for 5 seconds with a sampling rate of 2 kHz.

The LDA measurements of the axial and tangential fluid velocity components were performed at a cross-section in the draft tube cone.

4. Francis-99 result

The section presents the results of simulation of the flow in the turbine Francis-99. Three operation point proposed for the Francis-99 workshop were calculated [4]. Geometry data provided by the workshop committee were used. Computational domain included the runner and the draft tube. The mesh consisted of 6 mln. control volumes. Constant radial and tangential velocity components were set at the runner inlet. Angle between velocity direction and the runner inlet corresponded to guide vane angle. Outflow boundary conditions were set at the draft tube outlet.

Three operation points were calculated with of Reynolds stress model. Part load and high load operation points were also calculated with DES method. Figure 2 shows the velocity magnitude in central cross-section for two operation points. At the best efficiency point, narrow concentrated vortex is formed under the runner tip. On the other hand, at part load operation point, axial flow is close to the draft tube wall. At this operation point, wide recirculation zone is formed at the axis in the draft tube cone.

Tables 1 present parameters of the turbine at the operation points (net head in calculations was defined between runner inlet and the draft tube outlet). Computational results were obtained with steady Reynolds stress model. The results of calculations agree with experimental results qualitatively. Part load operation point shows the largest discrepancy of the efficiency.

Fig. 3 – 4 show velocity components along the top line in the draft tube for two operation points. The results of the calculations are in close agreement with experimental data at two operation points

both for Reynolds stress model and DES method. Part load operation point shows the smallest discrepancy.

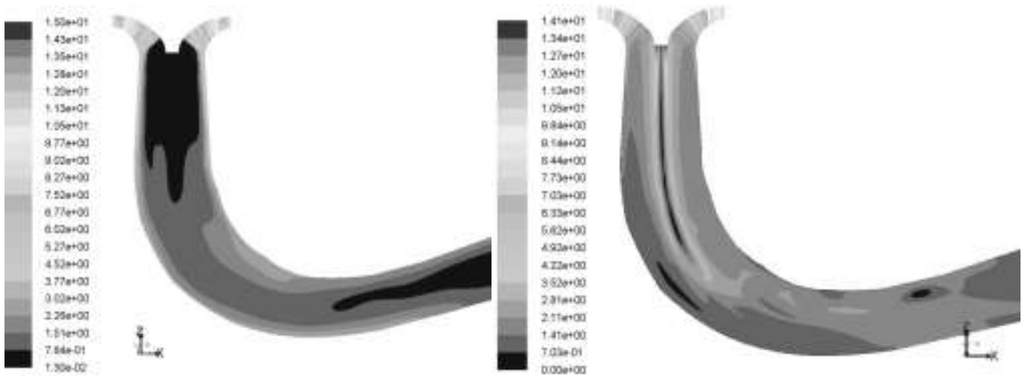

Figure 2: Velocity magnitude in central plane; part load and the best efficiency points

Table 1. Part load and the best efficiency point

Parameter	Part load		Best efficiency point	
	RSM	Experiment	RSM	Experiment
Net head [m]	11.30	12.29	11.61	11.91
Pressure torque [N m]	165.7	137.52	650.8	619.56
Friction torque [N m]	-3.5	6.54	-6.9	8.85
Total torque [N m]	162.2	–	643.9	–
Efficiency [%]	80.60	71.69	95.34	92.61

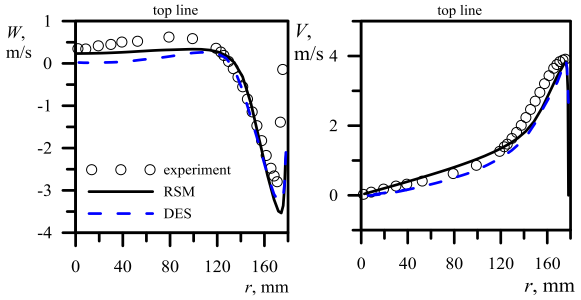

Figure 3: Velocity components in the draft tube, part load point

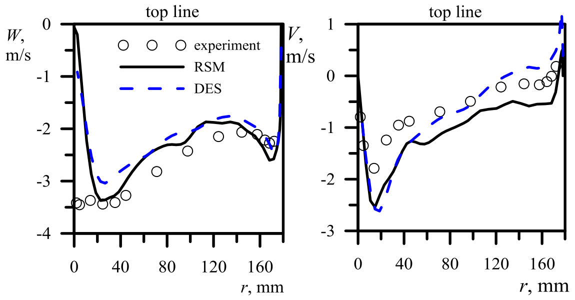

Figure 4: Velocity components in the draft tube, high load point

5. Aerodynamic stand

5.1 Experimental results

The structure of the flow downstream the runner was visualized and analyzed using the laser knife and flow tracers. The forming of the precessing vortex core in the draft tube diffuser was observed. Further instrumental and numerical study of the flow confirmed the presence of the PVC.

Measuring of the flow velocity downstream the runner was carried out using a LDA (Laser-Doppler anemometer) LAD-06I. Figure 6 shows velocity profiles for different flow rates (Q = 100, 150, 200 m^3/h). In addition, velocity profiles were measured for modes with different rotation speed of the runner including stationary rotor mode. As can be seen from the figure the stream in the draft tube is near the wall. A powerful recirculation zone is formed in the center and causes the PVC forming.

Using the sound-level meter (Bruel & Kjaer Type 2250) and piezo-electric sensors an amplitude and frequency of the pressure pulsations were measured. The analog-to-digital converter processed signals of sound-level meter and piezo-electric sensors. For example, spectra of the pulsations for mode with flow rate Q = 200 m^3/h and rotation speed ω = 2000 rpm shown at figure 7.

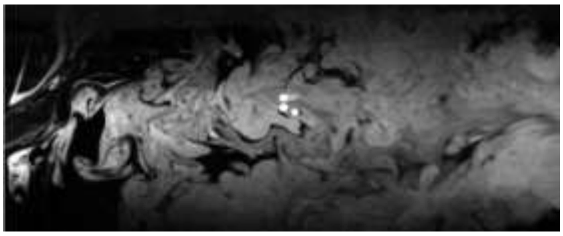

Figure 5: Visualization of the flow structure in the draft tube cone

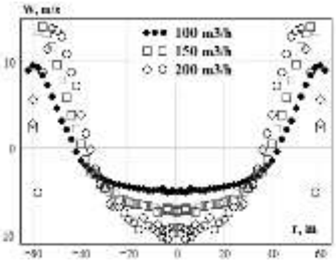

Figure 6: Profiles of the axial velocity in the draft tube diffuser, ω = 1000 rpm

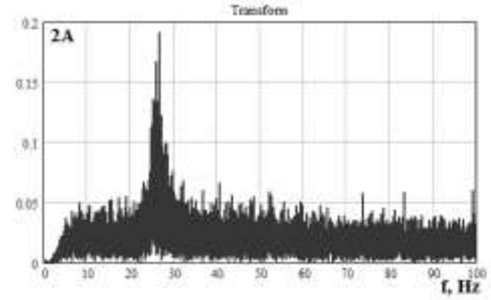

Figure 7: Spectra of the pressure pulsations

It is evident that increasing the flow rate leads to increasing the pulsations frequency. At the same time with increasing the runner speed the pulsations frequency reducing. It is quite logical because with increasing runner speed the flow swirling is reduced due to the fact that runner takes the whole flow swirling created by the stator and guide vanes.

5.2 Numerical research

The section presents the results of the simulation of the flow in the aerodynamic turbine stand. The calculations used unstructured grids of varying refinement. Figure 8 shows a comparison of the calculated and experimental axial velocity components. There is a satisfactory agreement. As has already been noted in the experiment there is a wide area of reverse flow, which indicates the presence of the precessing vortex core. This assumption is confirmed by the calculated data. Figure 9 shows the instantaneous pressure field in the cone of the draft tube. As seen in the diffuser of the draft tube is formed sufficiently strong PVC.

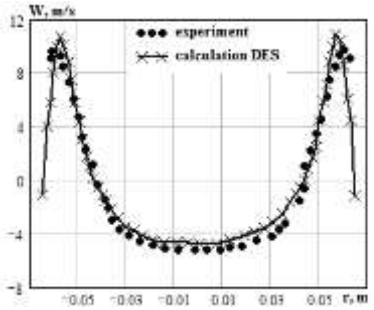

Figure 8: Profiles of the axial velocity in the draft tube diffuser

Figure 9: Vortex flow structure in the draft tube of the aerodynamic turbine (DES, RSM)

6. Conclusion

The paper proposed a numerical method for the simulation of flow in hydraulic turbines that was verified by means of experimental data. A unique aerodynamic stand, which is a miniature copy of the actual hydroelectric power, was designed. At the booth, a series of experiments simulating different modes of hydroelectric works. For this stand the velocity components and the pressure pulsation in the draft tube diffuser were measured. Using the numerical simulation technique the flow in the aerodynamic stand was calculated. The comparison of the calculated and experimental data showed close agreement that suggests the using possibility this numerical methods for the engineering applications.

We thank Russian Science Foundation for financial support (project No. 16-19-00138).

References

[1] Doerfler P. System dynamics of the Francis turbine half load surge. 1982, Proc. *IAHR Symp. on Hydraulic Machinery and Systems*, Amsterdam, Netherlands.Another reference

[2] Alligne S., Maruzewski P., Dinh T., Wang B., Fedorov A., Iosfin J. and Avellan F., 2010, Prediction of a Francis turbine prototype full load instability from investigations on the reduced scale model. Proc. *IAHR Symp. on Hydraulic Machinery and Systems*, Timisoara, Romania.

[3] Cervantes M.J., Engstrom T.F., Gustavsson L.H., 2005, Proc. *The third IAHR/ ERCOFTAC workshop on draft tube flows Turbine 99*. Sweden, Porjus,. 193 p.

[4] Minakov, A.V. Sentyabov, D.V. Platonov, A.A. Dekterev, A.A. Gavrilov, 2014, Numerical modeling of flow in the Francis-99 turbine with Reynolds stress model and detached eddy simulation method. *Journal of Physics: Conference Series* Volume 579, doi:10.1088/1742-6596/579/1/012004.

[5] Spalart P.R., Jou W.-H., Strelets M., Allmaras S.R. Comments on the feasibility of LES for wings and on a hybrid, RANS/LES approach / In Lue, C. and Lue, Z., (eds)// Advances in DNS/LES, Proceedings of 1st AFOSR International Conference on DNS/LES, Ruston, LA, August, 4-8, Greyden Press, Columbus, OH. – 1997. – pp. 137-147.

[6] Gavrilov A., Dekterev A., Sentyabov A., Minakov A., Platonov D. Application of hybrid methods to calculations of vortex precession in swirling flows // *Notes on Numerical Fluid Mechanics. 2012.* T. 117. P. 449-459.

[7] Kuznetsov I., Zakharov, A., Orekhov, G., Minakov, A., Dekterev, A., Platonov, D. Investigation of free discharge through the hydro units of high head Francis turbine. *IOP Conference Series: Earth and Environmental Science* 15 (PART 5). 2012. doi: 10.1088/1755-1315/15/5/052002

[8] Meledin V., Anikin Yu., Bakakin G., Glavniy V., Dvoinishnikov S., Kulikov D., Naumov I., Okulov V., Pavlov V., Rakhmanov V., Sadbakov O., Ilyin S., Mostovskiy N., Pylev I. Laser Doppler Diagnostic of Flow in Draft Tube behind Hydroturbine Runner // Turbomachines: Aeroelasticity, Aeroacoustics and Unsteady Aerodynamics, Edited by V.A. Skibin, V.E. Saren, N.M. Savin, S.M. Frolov. Moscow: Torus Press Ltd., 2006. –pp. 446-457.

[9] Litvinov I. V., Shtork S.I., Kuibin P.A., Alekseenko S. V., Hanjalic K., 2013, Experimental study and analytical reconstruction of precessing vortex in a tangential swirler, *Int. J. Heat Fluid Flow* 42: 251–264.