Numerical investigation of flow structure and pressure pulsation in the Francis-99 turbine during startup

A Minakov\textsuperscript{1,2}, A Sentyabov\textsuperscript{1,2} and D Platonov\textsuperscript{1,2}

\textsuperscript{1}Siberian Federal University, Krasnoyarsk, Svobodniy 79, 660079 Russia
\textsuperscript{2}Institute of Thermophysics SB RAS, Novosibirsk, Lavrentyeva 1, 630090 Russia

E-mail: tov-andrey@yandex.ru, sentyabov_a_v@mail.ru, platonov-08@yandex.ru

Abstract. We performed numerical simulation of flow in a laboratory model of a Francis hydroturbine at startup regimes. Numerical technique for calculating of low frequency pressure pulsations in a water turbine is based on the use of DES (k-\omega Shear Stress Transport) turbulence model and the approach of "frozen rotor". The structure of the flow behind the runner of turbine was analysed. Shows the effect of flow structure on the frequency and intensity of non-stationary processes in the flow path. Two version of the inlet boundary conditions were considered. The first one corresponded measured time dependence of the discharge. Comparison of the calculation results with the experimental data shows the considerable delay of the discharge in this calculation. Second version corresponded linear approximation of time dependence of the discharge. This calculation shows good agreement with experimental results.

Keywords: Francis turbine, numerical simulation, pressure pulsation, the precession of the vortex rope, turbulence, CFD, LES, RANS and DES models.

1. Introduction

One of the main role of hydroelectric power plants is providing peaking power production. Therefore, quick varying of the operating mode, as startup, shutdown or adjustment of power output, are often required for the hydraulic turbines. In phase of transient between different modes of operation, pulsation phenomena are more intensive than in steady operation [1]. Transient operation results in high amplitude of the dynamic forces on the runner. High pressure pulsations occurs in the part load zone with vortex rope.

Flows in hydroturbine systems are characterized by several features which pose a challenge to the RANS models. The flows in the guide vanes and the rotating runner are dominated by the bounding wall and a strong favorable pressure gradients. The phenomena in a draft tube are different. Here the flow is governed by a strong swirl and a bluff-body recirculation zone behind the runner hub, which is usually reinforced by the negative pressure due to the swirl. The 90 degree bend encountered in elbow draft tubes, with a subsequent circular-to-rectangular change of the cross-section further modify the flow especially at part loads creating very complex vortical patterns with additional recirculation and secondary flows. On the other hand, the walls are smooth, and despite a strong elbow curvature, the wall-adjacent azimuthal fluid velocity and turbulence seem to follow reasonably well the common wall scaling so that computations seem manageable by the common wall-integration schemes or even with using the standard wall-function approach.
Since transients between different modes of turbine operation are very important for the hydraulic unit and accompanied by a large number of complex hydrodynamic phenomena, it is important to be able to simulate dynamic of these processes together. Accordingly, the aim of this paper is to test the numerical algorithm, suitable for simulation of turbine startup process.

Unsteady processes in hydraulic turbines were studied in many papers. The paper [2] presents a mathematical model of hydro power units for different operating conditions, based on the basic version of the software TOPSYS. The simulation and on-site measurements are compared for start-up, no-load operation, normal operation, and load rejection in different control modes (frequency, opening, and power feedback). As a result, the model application is proved trustworthy for simulating different physical quantities of the unit (e.g., guide vane opening, active power, rotation speed, and pressures at volute and draft tube).

The examples of 1D-3D modeling of transient processes in turbine can be found in works [3–5]. In the paper [3] a new approach for numerical simulation of 3D hydraulic turbine flows in transient operating regimes is presented. The method is based on a coupled solution of incompressible RANS equations, runner rotation equation, and water hammer equations. The issue of setting appropriate boundary conditions is considered in detail. As an illustration, the simulation results for runaway process are presented. The evolution of vortex structure and its effect on computed runaway traces are analyzed.

The study Yanna Liu et al. [4] establishes a mathematical model of the transient process in hydropower stations and presents a new method to calculate the hydraulic turbine boundary based on an error function of the rotational speed. The mathematical derivation shows that the error function along the equal-opening characteristic curve is monotonic and has opposite signs at the two sides, which means that a unique solution exists to make the error function null.

The paper [5] presents the simulation and the analysis of the transient process of a Francis turbine during the load rejection by employing a one-dimensional and three-dimensional (1-D-3-D) coupling approach. The coupling is realized by partly overlapping the 1-D and 3-D parts, the water hammer wave is modeled by defining the pressure dependent density, and the guide vane closure is treated by a dynamic mesh method. To verify the results of the coupling approach, the transient parameters for both typical models and a real power station are compared with the data obtained by the 1-D approach, and good agreements are found. To investigate the differences between the transient and steady states at the corresponding operating parameters, the flow characteristics inside a turbine of the real power station are simulated by both transient and steady methods, and the results are analyzed in details.

2. Mathematical model and numerical method
The computations here reported were performed with the ANSYS-FLUENT code using the DDES method (based on the $k-\omega$ SST Menter’s model). The basic equations of the mathematical models express the conservation laws in the rotating reference frame.

The continuity equation (conservation of mass):

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_j)}{\partial x_j} = 0$$

Momentum equations (conservation of momentum) in a rotating reference frame for absolute velocities:

$$\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}(\tau_{ij} + \tau_{ij}^w) - \rho \Omega_x \Omega_y u_i$$
Where: $u_i$ – absolute velocity components, $u'_j$ – relative velocity components, $\tau^{m}_{ij}$ – viscous stress tensor, $\tau^{v}_{ij}$ – turbulent stress tensor, $\Omega$ – angular velocity of runner rotation, $p$ – static pressure, $\rho$ – density, $\epsilon_{ijk}$ – Levi-Civita symbol.

Components of the viscous stress tensor are defined as:

$$
\tau^{m}_{ij} = \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)
$$

where $\mu$ is dynamic molecular viscosity. In constructing two-equation models of turbulence for defining the components of Reynolds’ stress tensor $\tau^{v}_{ij}$, Boussinesq’s hypothesis of isotropic turbulent viscosity is used:

$$
\tau^{v}_{ij} = \mu_i \left[ \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} \rho k \right]
$$

In this paper we use the DES approach based on $k$-$\omega$ SST model [6]. The dissipation term of the turbulent kinetic energy is modified for the DES turbulence model as described in [7]. In this model dissipative term in $k$-equation is modified by means of switcher $F_{DES}$:

$$
\frac{\partial (\rho k)}{\partial t} + \frac{\partial}{\partial x_j} \left( \rho u_j k \right) = \frac{\partial}{\partial x_j} \left( \Gamma_k \frac{\partial k}{\partial x_j} \right) + G_k - Y_k \cdot F_{DES},
$$

where:

$$
F_{DES} = \max \left( \frac{l_i}{C_{DES} \Delta}, 1 \right), \quad l_i = \frac{k^{1/2}}{\beta \omega}, \quad C_{DES} = 0.61
$$

where: $l_i$ – turbulent length scale, $C_{DES}$ – empirical constant, and $\Delta$ is defined as the maximum of three sizes of the control volume $\Delta_x, \Delta_y, \Delta_z$.

A specific issue in the computer simulation of hydraulic turbines is the treatment of the rotating impeller and the rotor-stator interaction. Several approaches can be found in the literature, i.e. dynamic, sliding and moving grid methods, and those based on a moving reference frame. The latter is the most common and the simplest way to model the runner rotation. It assumes that the runner is fixed and the equations are solved in a rotating reference frame. This formulation is often referred to as the "frozen rotor" approach. In this paper, the modelling of the runner rotation was performed in the rotated reference frame for the runner zone. The earlier test calculations proved that this approach is credible for describing the integral flow characteristics including the dominant flow pulsations [8–10]. The "frozen rotor" (known also as “moving reference frame”) is a well-established and widely used method for treating stator-rotor interaction. In this approach the rotating and the stationary parts have fixed relative positions. A frame transformation is done to include the rotating effect on the rotating sections. Simultaneous computations of stator and rotor flow using sliding meshes is undoubtedly superior and more exact, but it is computationally more demanding, it requires longer iterations, and it is less robust. The frozen rotor approach is more practical, economical and robust; our earlier test for a similar configuration showed a very small, almost marginal difference between the results obtained by the two approaches.

The computations were performed using the finite-volume method on unstructured polyhedral grids. The coupling of the velocity and pressure fields for incompressible flow was ensured using the SIMPLE-C procedure. The second-order central difference scheme was used for DES. The time
derivatives were approximated by an implicit second-order scheme. The time step was set from the condition of averaged CFL < 2.

3. Computational domain, boundary conditions and grids
The computational domain included the runner and the draft tube (see Fig.1). The 3D geometry of the turbine has been taken on the website workshop Francis (2016). The computations have been carried out using unstructured grids with the total number of 7.06 million nodes for the whole domain (Fig.2). The distance of the wall-nearest grid nodes $y^+$, was approximately from 40 to 1000. Despite the large number of nodes, this mesh is insufficient for the correct resolution of the boundary layers. Nevertheless, the mesh is suitable for simulation of the large-scale vortices in the draft tube [6].

The inlet boundary conditions were set at the runner inlet by means of time-dependent functions of the radial and tangential velocities (Fig. 3, version 1). The radial velocity component was obtained from the approximated time dependence of the experimental value of discharge (Fig. 4, version 1), and the tangential velocity component was obtained from the radial velocity and experimental value of the guide vanes angle (Fig. 5). Experimental discharge was approximated by means of polynomial (Fig. 4, version 1). However, for the transient measurements, there is some delay in the flowmeter. For this reason, another approximation of the discharge evolution was also tested. Second version of the discharge time dependence was is linear between 2 and 10 s (Fig 4). Such period correspond to the experimental velocity data.

Figure. 1. The computational domain.
Figure. 2. Views of the computational grid with 7 million nodes in the turbine mode (the draft tube and the runner).

Figure. 3. Velocity components on the inlet: a) radial velocity, b) tangential velocity. Version 1 is based on measured discharge, version 2 is based on linear approximation of the discharge.
4. The results of the startup simulation in the Francis-99 turbine

As shown in Fig. 6, at the initial time at the small guide wicket gate there is opening long vortex rope under the runner and in the draft tube elbow (Fig. 6, time marks correspond Fig. 3 – 5). Calculated pressure pulsations closely agree with the experimental data (Fig. 7, 8). The vortex rope precession induces intensive low-frequency pressure pulsations in the draft tube at \( f \approx 1.5 \) Hz which are clearly seen for the initial stage of the startup (Fig. 9). Due to the long vortex rope, that reaches draft tube elbow, pressure pulsations are very intensive at this stage. Likely, interaction of the vortex rope and the draft tube elbow induces synchronous pressure pulsation in addition to asynchronous pulsations generated by vortex rope precession [10]. At the end of the startup the experimental spectrum contains a number of high frequencies (Fig. 9).
Figure 6. Vortex evolution in the draft tube visualized by pressure iso-surface (red), velocity magnitude in the central longitudinal cross-section (obtained for the calculation No 1).
Figure 7. Pressure pulsations at the point DT5 (blue line – simulation, red line - experiment).

Calculation No 2

Figure 8. Pressure pulsations at the point DT6 (blue line – simulation, red line - experiment).

Calculation No 2
**Figure. 9.** Spectra of the pressure pulsations at the point DT5 (blue line – simulation, red line – experiment): a) time window is 0-2 s; b) time window is 2-4 s; c) time window is 4-8 s; d) time window is 8-24 s. Calculation No 2
As wicket gate opening increase and, accordingly the flow swirl number decreases, intensity of the vortex rope reduces (Fig. 6, $t = 4$ s). As a result, magnitude of the pressure pulsations decreases. Then (Fig. 6, $8 - 10$ s), intensive vortex rope in the draft tube collapses and recirculation zone under the runner vanishes (Fig. 11, $8 - 10$ s). At this stage, only high-frequency pressure pulsations, associated with the runner rotation, remain in the experimental pressure pulsations but these pulsations were not considered in the calculation (Fig. 9d). Near the best efficiency point, the vortex under the runner has straight form (Fig. 6, $t = 12$ s) and intensity of the pressure pulsations reduces to the minimum value (Fig. 7, 8).

Comparison of the velocity profiles in the draft tube are shown in Fig. 11. The figure presents calculated and experimental velocity profiles averaged during 2 s. Such period is rather long to include many computational and experimental time steps and is rather short compared to time of the startup. Satisfactory agreement of the calculated and experimental results can be seen in the figure. As Fig. 11 shows, before the turbine startup and at initial transient stage ($t = 0 - 2$ s, $t = 4 - 6$ s) there is a wide zone of the backward flow in the draft tube. This recirculation zone forms due to high swirl number of the flow under the runner and, accordingly, precessed vortex rope in the draft tube. At the end of the startup transient process, calculation No 1 shows considerable discrepancy with the experimental data. The plot reveals that the calculated discharge is smaller than the experimental one. It is due to delay of the flowmeter. As Fig. 11 shows, calculation No 2 describes the startup well.
6. Conclusions

Thus, as the opening of wicket gate increase, the flow under the runner pass different stages corresponding to the stable-state operation. At low wicket gate opening, the flow in the draft tube is highly swirled, wide recirculation zone forms under the runner and vortex rope rotates around it. The vortex rope has several period of the rotation during the startup and induce intensive low-frequency pressure pulsations. Then, the vortex rope gradually destroys and pressure pulsations becomes non-periodical. At last, only weak straight vortex core remains under the runner and the pressure pulsations reaches minimal magnitude.

Good agreement between the experimental and calculated data shows that the algorithm is suitable for the simulating of the transient processes in hydraulic turbines. Since the main influence on the unsteady flow processes has the flow in the draft tube, the flow in the guide vanes and the spiral case...
can in neglected in the simplest approach, and only the runner and the draft tube can be considered. In addition, detached eddy simulation is suitable for the correct resolving of the dominant vortices under the runner. Comparison of the calculated and experimental unsteady behaviors of the velocity in the monitoring points in the draft tube has shown that the calculation is in very good agreement with experiment both in amplitude and frequency fluctuations of velocity during start-up of the turbine.

In addition it is necessary to note one more interesting feature. In this paper, modeling of the opening of the guide device in the startup process is not carried out. Firstly, as the inlet boundary conditions time dependences of the radial and tangential velocity components, obtained from the experimental values of the opening angle of the guide vanes and flow discharge, were used. Results of the simulation in this formulation showed that the calculated profiles of axial velocity is significantly different from the experimental data. Therefore, the discharge time dependence has been adjusted. It was used the assumption that the discharge linearly increases from a minimum to a maximum value during startup period of time 8s. This implicitly agrees with the delay in measured discharge. Also it is confirmed by experimental measurements of the velocities in the monitoring points depending on time. Calculations showed that second version of the discharge dependence provides a very good agreement of the velocity profiles during the startup of the turbine.

Acknowledgements
This work was supported by Russian Foundation for Basic Research 16-48-240707.

References
[1] Doerfler P, Sick M and Coutu A 2013 Flow-Induced Pulsation and Vibration in Hydroelectric Machinery. Engineers Guidebook for Planning, Design and Troubleshooting. London: Springer
[2] Weijia Yang, Jiandong Yang, Wencheng Guo, Wei Zeng, Chao Wang, Linn Saarinen and Per Norrlund 2015 A Mathematical Model and Its Application for Hydro Power Units under Different Operating Conditions Energies 8 10260-10275; doi:10.3390/en80910260.
[3] Cherny S, Chirkov D, Bannikov D, Lapin V, Skorospelov V, Eshkunova I and Avdushenko A 2010 3D numerical simulation of transient processes in hydraulic turbines. IOP Conf. Series: Earth and Environmental Science 12 012071 doi:10.1088/1755-1315/12/1/012071
[4] Yanna Liu, Jiandong Yang, Jiebin Yang, Chao Wang and Wei Zeng 2016 Transient Simulations in Hydropower Stations Based on a Novel Turbine Boundary. Mathematical Problems in Engineering Volume 2016, Article ID 1504659, 13 pages. http://dx.doi.org/10.1155/2016/1504659
[5] ZHANG Xiao-xi, CHENG Yong-Guang, YANG Jian-Dong, XIA Lin-Sheng, LAI Xu.
[6] Menter F R 1994 Two Equation Eddy Viscosity Turbulence Models for Engineering Applications AIAA J. 32, No 8, 1598-1605
[7] Strelets M 2001 Detached Eddy Simulation of Massively Separated Flows. AIIA Paper. 2001-0879
[8] Minakov A V, Platonov D V, Dekterev A A, Sentyabov A V and Zakharov A V 2015 The numerical simulation of low frequency pressure pulsations in the high-head Francis turbine Computers and Fluids 111 pp. 197-205, DOI:10.1016/j.compfluid.2015.01.007
[9] 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 International Journal of Heat and Fluid Flow 53 pp. 183-194, DOI: 10.1016/j.ijheatfluidflow.2015.04.001
[10] Gavrilov A A, Sentyabov A V, Dekterev A A and Hanjalić K 2016 Vortical structures and pressure pulsations in draft tube of a Francis-99 turbine at part load: RANS and hybrid RANS/LES analysis International Journal of Heat and Fluid Flow (June)
doi:10.1016/j.ijheatfluidflow.2016.05.007

[11] Sentyabov A V, Gavrilov A A, Dekterev A A and Minakov A V 2014 Numerical investigation of the vortex core precession in a model hydro turbine with the aid of hybrid methods for computation of turbulent flows Thermophysics and Aeromechanics 21(6) pp. 707 – 717.