INVESTIGATION OF FLOW STRUCTURE AND PRESSURE PULSATION IN THE FRANCIS-99 TURBINE IN TRANSIENT REGIMES

[1,2]Minakov A., [1,2]Sentyabov A., [1,2]Platonov D.
[1]Siberian Federal University, Krasnoyarsk, Russia
[2]Institute of Thermophysics SB RAS, Novosibirsk, Russia

Abstract. *We performed numerical simulation of flow in a laboratory model of a Francis hydroturbine at a startup regimes. Numerical technique for calculating of low frequency pressure pulsations in a water turbine is based on the use of DES (k–ω Shear Stress Transport) turbulence model and the approach of "frozen rotor". The structure of the flow behind the runner of turbine analyzed. Shows the effect of flow structure on the frequency and intensity of non-stationary processes in the flow path. Comparison of calculation results with the experimental data was carried out. A good agreement between theory and experiment was obtained.*

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.

2. The numerical model

The computations here reported were performed with the ANSYS-FLUENT code using the DDES method (based on the $k\text{–}\omega$ 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):

$$\frac{\partial u_i}{\partial x_i} = 0$$

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

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial}{\partial x_j}\left(\rho u_j^r u_i\right) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}(\tau_{ij}^{\mathrm{m}} + \tau_{ij}^{\mathrm{t}}) - \rho \varepsilon_{ijk} \Omega_j u_k$$

Where: u_i – absolute velocity components, u_j^r – relative velocity components, τ_{ij}^{m} – viscous stress tensor, τ_{ij}^{t} – turbulent stress tensor, Ω – angular velocity of runner rotation, p – static pressure, ρ – density, ε_{ijk} – Levi-Civita symbol.

Components of the viscous stress tensor are defined as:

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

where μ is dynamic molecular viscosity. In constructing two-equation models of turbulence for defining the components of Reynolds' stress tensor τ_{ij}^{t}, Boussinesq's hypothesis of isotropic turbulent viscosity is used:

$$\tau_{ij}^{\mathrm{t}} = \mu_{\mathrm{t}}\left[\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \frac{2}{3}\delta_{ij}\rho k\right]$$

In this paper we use the DES approach based on k-ω SST model [2]. The dissipation term of the turbulent kinetic energy is modified for the DES turbulence model as described in [3]. In this model dissipative term in k-equation is modified by means of switcher F_{DES}:

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

$$F_{\mathrm{DES}}=\max\left(\frac{l_{\mathrm{t}}}{C_{\mathrm{DES}}\Delta},1\right),\ l_{\mathrm{t}}=\frac{k^{1/2}}{\beta^{*}\omega},\qquad C_{\mathrm{DES}}=0.61$$

where: l_{t} – turbulent length scale, C_{DES} – empirical constant, and Δ 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 [4 – 6].

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 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.

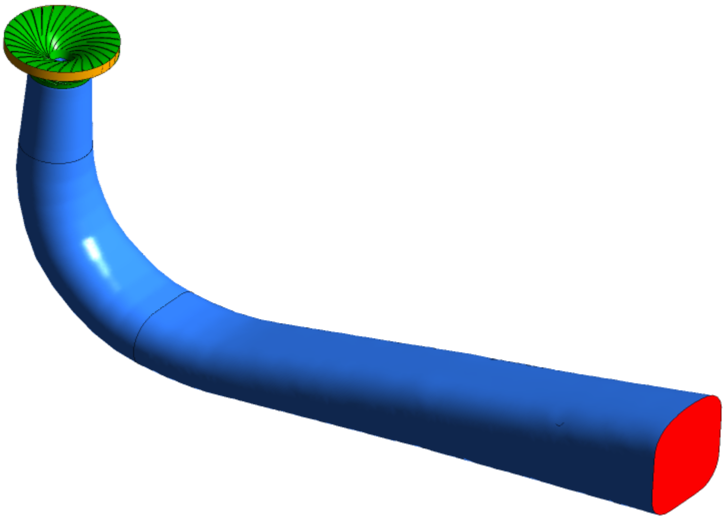


Fig. 1. The computational domain.

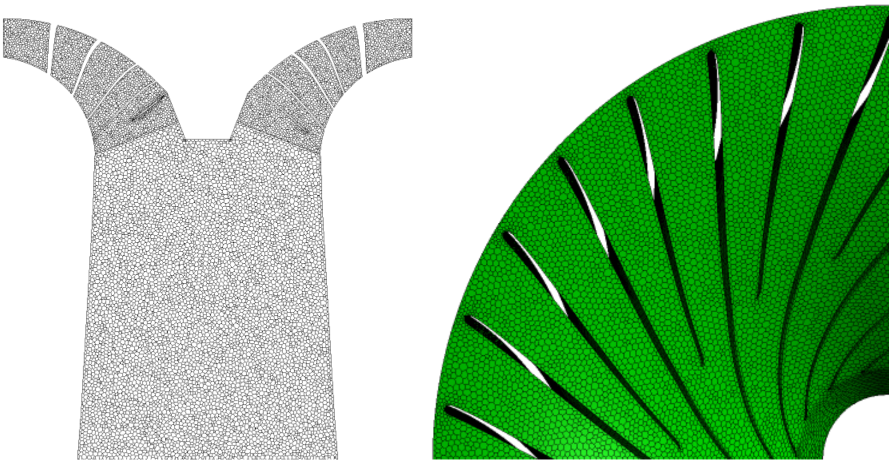


Fig. 2. Views of the computational grid with 7 million nodes in the turbine mode 1 (the draft tube and the runner).

The inlet boundary conditions were set at the runner inlet by means of time-dependent functions of the radial and tangential velocities (Fig. 3). The radial velocity component was obtained from the approximated time dependence of the experimental value of discharge (Fig. 4), and the tangential velocity component was obtained from the radial velocity and experimental value of the guide vanes angle (Fig. 5).

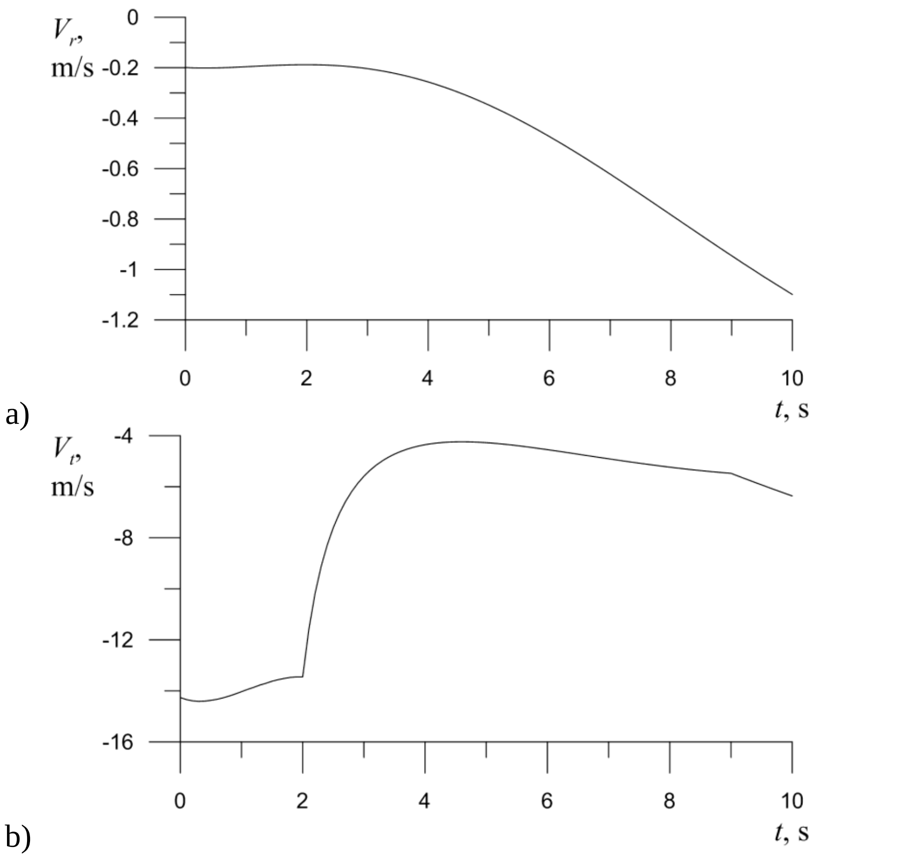


Fig. 3. Velocity components on the inlet: a) radial velocity, b) tangential velocity

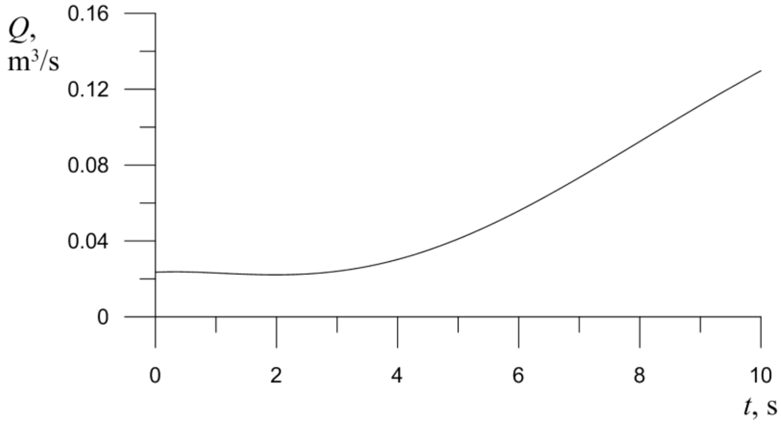


Fig. 4. Approximated time dependence of the discharge

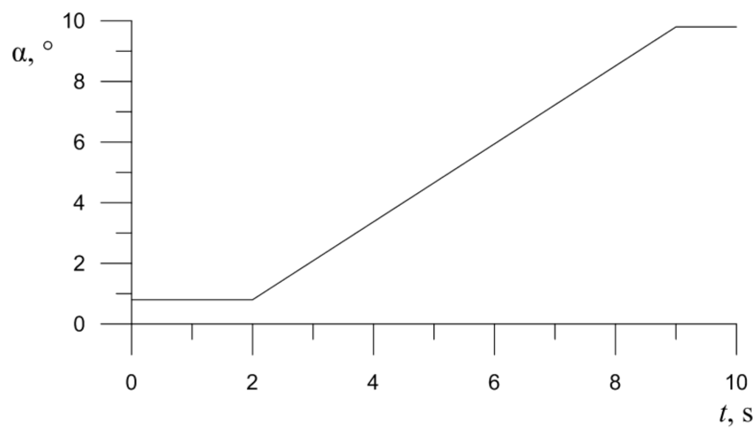


Fig. 5. Approximated time dependence of the guide vanes angle

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). The vortex rope precession induces intensive low-frequency pressure pulsations in the draft tube at $f \approx 1.5$ Hz which are clearly seen (Fig. 7 – 10). Calculated pressure pulsations closely agree with the experimental data (Fig. 7 – 10). 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 [7].

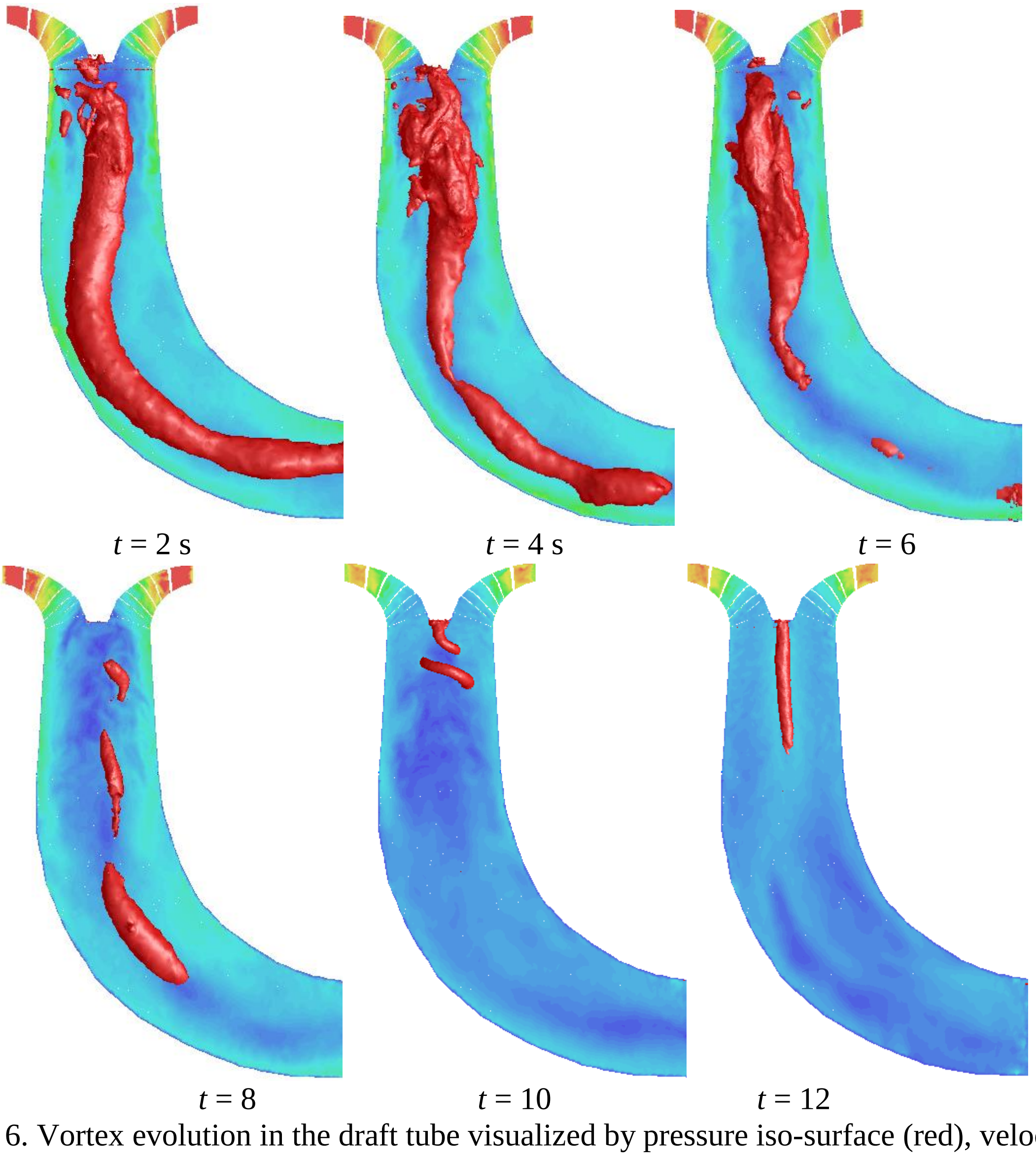


Fig. 6. Vortex evolution in the draft tube visualized by pressure iso-surface (red), velocity magnitude in the central longitudinal cross-section

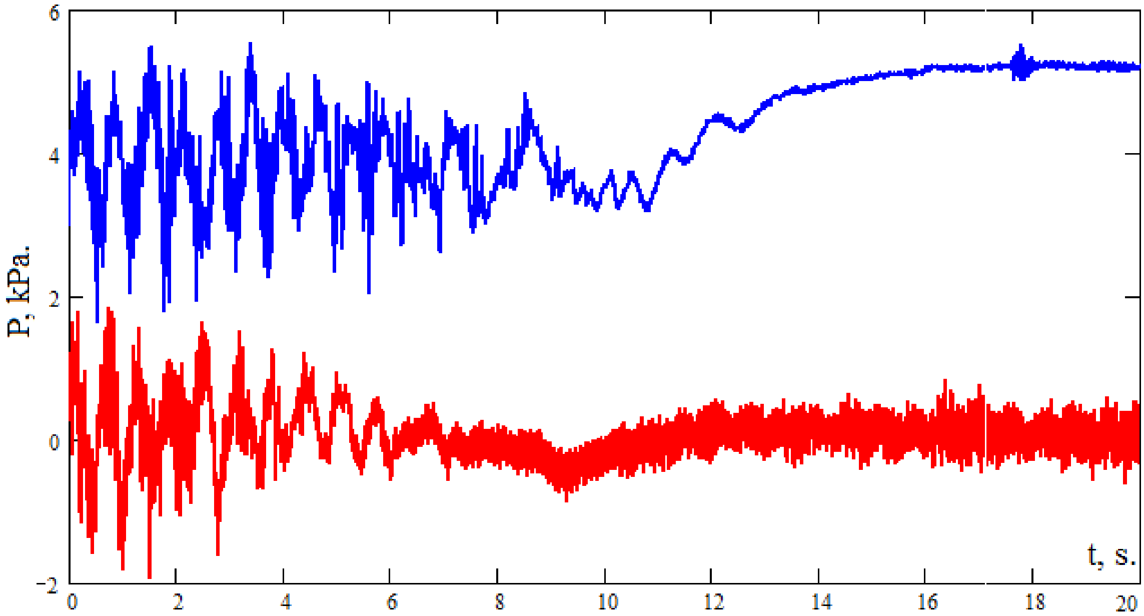
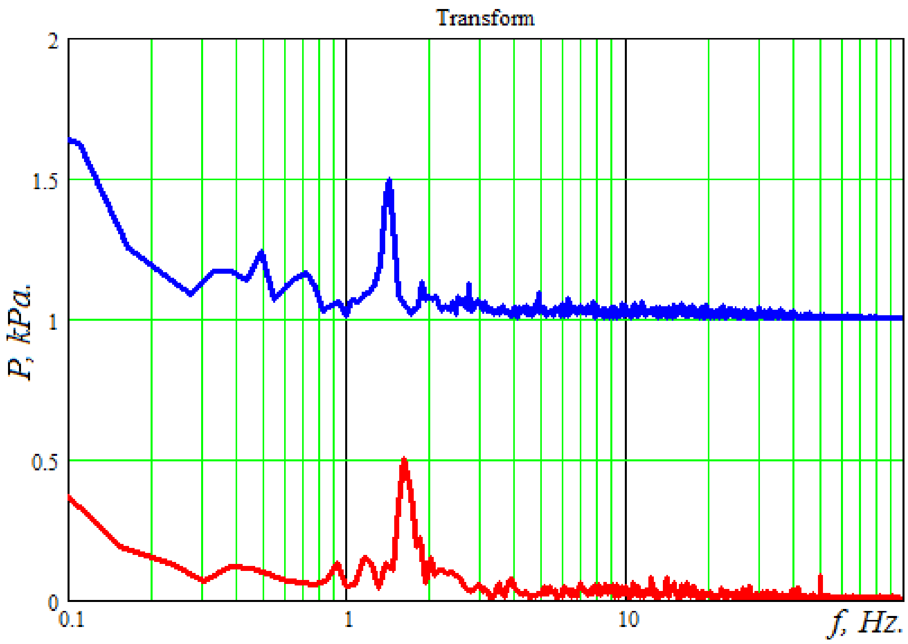


Fig. 7. Pressure pulsations at the point DT5 (blue line – simulation, red line - experiment)



DT5

Fig. 8. Spectra of the pressure pulsations at the point DT5 (blue line – simulation, red line - experiment)

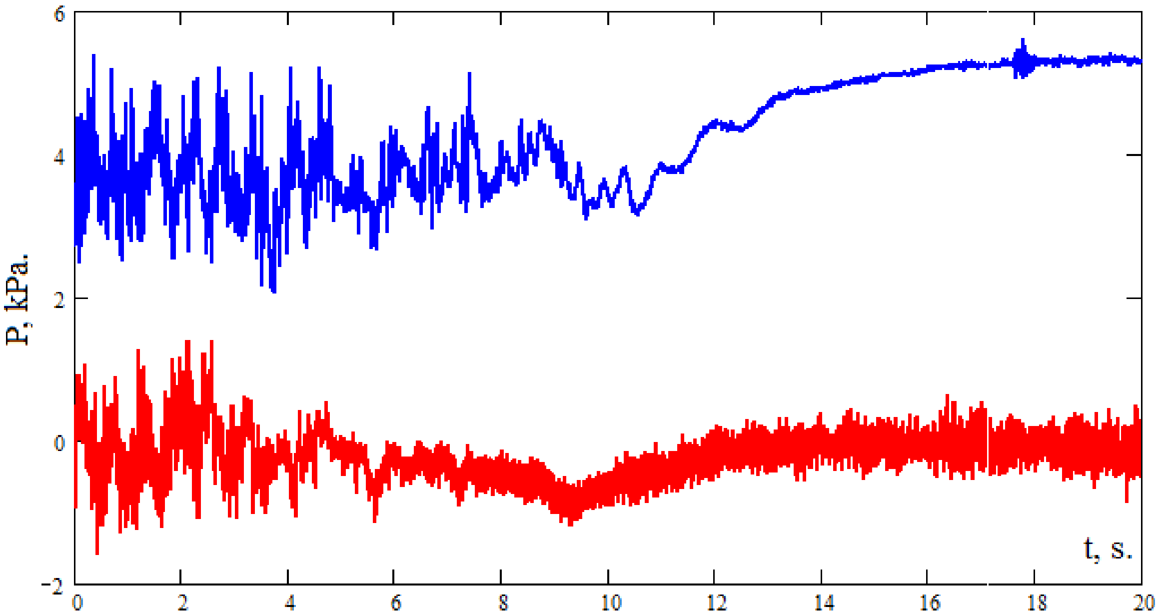


Fig. 9. Pressure pulsations at the point DT6 (blue line – simulation, red line - experiment)

DT6

Fig. 10. Spectra of the pressure pulsations at the point DT7 (blue line – simulation, red line - experiment)

Comparison of the velocity profiles in the draft tube are shown in Fig. 11. Satisfactory agreement of the calculated and experimental results can be seen. 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, recessed vortex rope in the draft tube.

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. 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, 9).

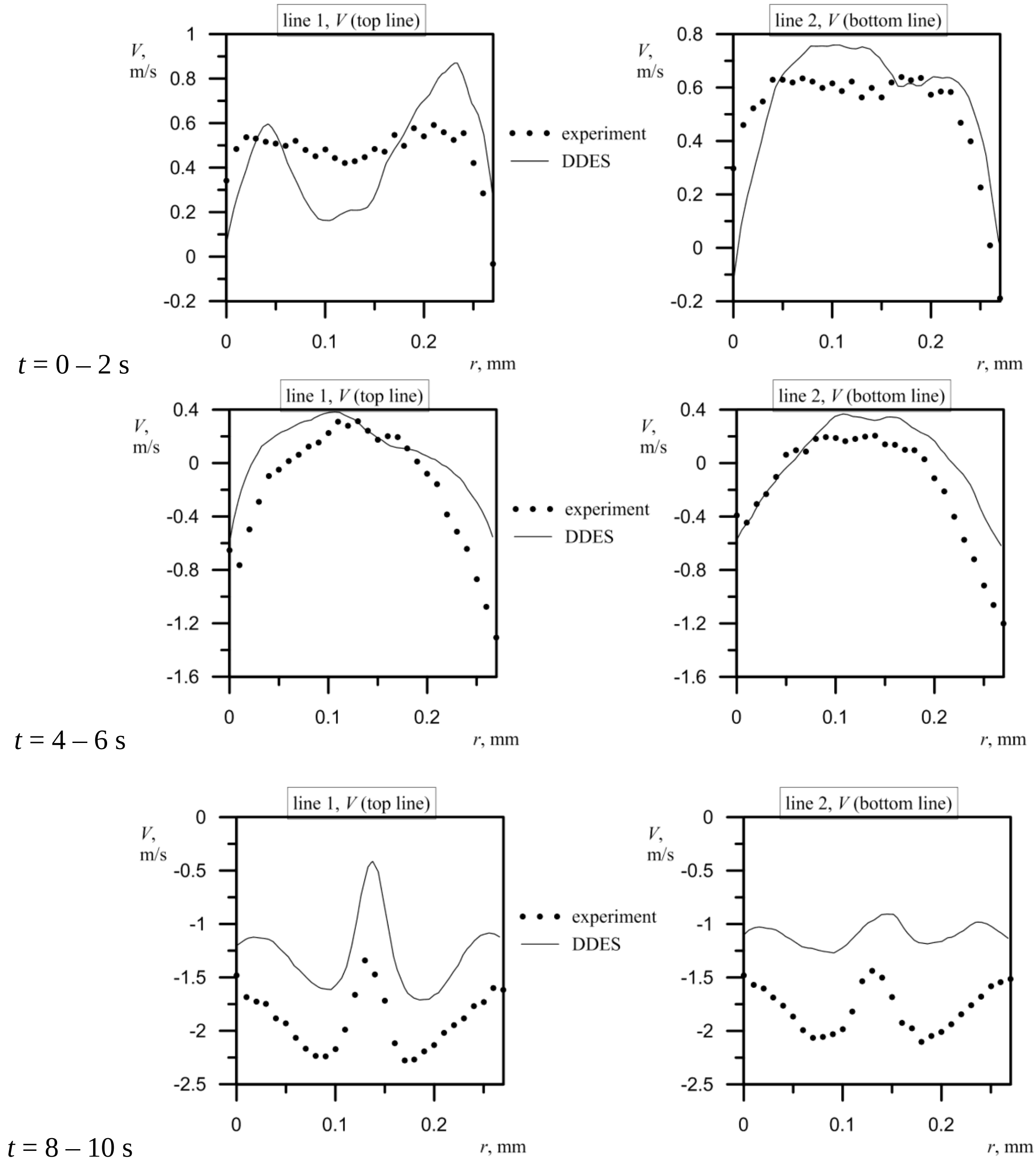


Fig. 11. Averaged velocity components under the runner

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.

Acknowledgements

This work was supported by Russian Science Foundation (project No. 16-19-00138) and the Russian Foundation for Basic Research 16-48-240707.

References

[1] Doerfler P., Sick M. and Coutu A. Flow-Induced Pulsation and Vibration in Hydroelectric Machinery. Engineers Guidebook for Planning, Design and Troubleshooting. London: Springer, 2013,

[2] Menter F R 1994 Two Equation Eddy Viscosity Turbulence Models for Engineering Applications *AIAA J.* **32**, № 8, 1598-1605

[3] Strelets M 2001 Detached Eddy Simulation of Massively Separated Flows. *AIIA Paper.* 2001-0879

[4] Minakov A.V., Platonov D.V., Dekterev A.A., Sentyabov A.V., Zakharov A.V. The numerical simulation of low frequency pressure pulsations in the high-head Francis turbine // Computers and Fluids, 2015, Vol. 111, pp. 197-205, DOI:10.1016/j.compfluid.2015.01.007

[5] Minakov A.V., Platonov D.V., Dekterev A.A., Sentyabov A.V., Zakharov A.V. The analysis of unsteady flow structure and low frequency pressure pulsations in the high-head Francis turbines // International Journal of Heat and Fluid Flow, 2015, Vol. 53, pp. 183-194, DOI: 10.1016/j.ijheatfluidflow.2015.04.001

[6] Gavrilov A.A., Sentyabov A. V., Dekterev A.A. and Hanjalić K., 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. 2016 (June) doi:10.1016/j.ijheatfluidflow.2016.05.007

[7] Sentyabov A.V., Gavrilov A.A., Dekterev A.A. and Minakov A.V., 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, 2014, Vol. 21, No 6. pp. 707 – 717.