Numerical investigation of flow structure in hydraulic turbines of high-head power development

Dmitry Platonov, Andrey Minakov, Andrey Sentyabov, Dmitry Dekterev

Department of Thermal Physics
Siberian Federal University Krasnoyarsk,
Svobodny pr., 79
Russia

Department of Physical Bases of Energy Technologies
Institute of Thermophysics, Novosibirsk
Acad. Lavrentyev ave., 1
Russia

platonov-08@yandex.ru http://www.sfu-kras.ru

Abstract: - Numerical technique for calculating pressure pulsations in a high-head hydro power plant (HHPP) is presented in the paper. The results of application of the numerical technique for description of unsteady turbulent flow in a flow path of a water turbine are presented. Adaptation of numerical method performed on the results of the simulation of turbulent unsteady flow in the flow path of the high-head hydro power plant is carried out. The structure of the flow behind the runner and its influence on the intensity of non-stationary processes was analyzed. Comparison of calculation results with the prototype experimental data was carried out. A good agreement between theory and experiment was obtained.

Key-Words: Francis turbine, numerical simulation, pressure pulsation, the precession of the vortex rope, turbulence, CFD

1 Introduction

An important task for HHPP is regulation of power in the energy system. During load changing hydraulic units repeatedly undergo off-design modes of operation. Under these flow conditions a significant part of swirl, after passing through the water turbine runner, is remained. With the instability of swirling flow is related the emergency of intense low-frequency hydrodynamic fluctuations that threaten the reliability of turbine design [1-2].

Vortex rope precession is a serious danger for the hydraulic turbine equipment in relation with the powerful flow pulsations that lead to strong vibrations of hydraulic turbine construction and in the case of resonance can lead to destruction of the equipment. Pressure pulsations generated by the vortex rope precession, may also affect cavitations processes, enhancing cavitations erosion. To predict the resonance phenomena and search methods of suppressing instability requires detailed information about the characteristics of the pulsation modes and flow structure. It should be noted that the developed approaches must meet the requirements of minimizing energy losses (increase efficiency of water turbine), which also can be realized only through in-depth understanding of the hydrodynamic processes occurring in the flow of hydraulic turbine parts.

The development of computer technology has made it possible to use modern methods of computational fluid dynamics to the description of turbulent flows in geometrically complex spatial objects, such as hydraulic turbine flow path. Recently there appeared a large number of papers in which three-dimensional formulation in the calculations of various processes in hydraulic turbines were made [3-6].

2 Mathematical model

Describing the flows in the flow path of water turbines one have to face several problems. The first challenge is related to modeling turbulence in channels of complex geometric shapes and strong swirl flow. At the same time for engineering calculations turbulence models, quite accurate describing not only required average fields, but also large-scale flow pulsations. 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 either to modify existing URANS turbulence models or use techniques which resolve large-scale turbulent eddies (LES, DES) [7,8].

Swirling flow in hydraulic turbines can be accompanied by the precession of the vortex core.

For modeling such phenomenon it is necessary to use non stationary, in particular, large-eddy simulation methods such as the method of large-eddy simulation (Large Eddy Simulation – LES). However, its use requires a very detailed grid, especially near the walls. At the same time RANS models are quite economical and have a good description of the boundary layers. To combine the advantages of these approaches in operation the method of detached eddy simulation (DES) was proposed. The first DES version was based on the model of Spalart-Allmaras. Later DES method has been used with other models of turbulence, there have been various modifications.

In the simulation of hydraulic turbines it is necessary to consider the rotation of the runner and the rotor-stator interaction. There are many approaches for modeling flows with rotating bodies, they include: dynamic grid, moving mesh method based on the transition in a moving coordinate system. The most common and simple way to model the runner rotation is to use a rotating coordinate system. The transition to a rotating coordinate system allows to simulate flow in an approximation in which the runner is fixed and swirl flows pass through it. This formulation is often referred to as "frozen rotor". In this paper, modeling of the runner rotation was performed in a form of frozen rotor. Conducted numerous test calculations show the correctness of this approach, both in the description of integral characteristics of flow and pulsation.

Below the basic equations of mathematical models expressing the conservation, laws in the rotating coordinate system are presented.

The continuity equation (conservation of mass):

$$\frac{\partial \rho}{\partial t} + \nabla\left(\rho \mathbf{v}\right) = 0$$

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

$$\frac{\partial \rho \mathbf{v}}{\partial t} + \nabla\left(\rho \mathbf{v} \cdot \mathbf{v}\right) = -\nabla p + \nabla(\boldsymbol{\tau}^m + \boldsymbol{\tau}^t) + (\rho - \rho_0)\mathbf{g} - \rho(2\boldsymbol{\Omega} \times \mathbf{v} + \boldsymbol{\Omega} \times (\boldsymbol{\Omega} \times \mathbf{r}))$$

where: $\mathbf{v}$ - fluid velocity vector, $\boldsymbol{\tau}$ - viscous stress tensor, Ω- angular velocity of runner rotation, p - static pressure , ρ - density. In the transition to a rotating coordinate system on the right side of the equation of conservation of momentum Coriolis' force and the centrifugal force are written.

Components of the viscous stress tensor is defined as:

$$\tau_{ij}^m = \mu\left[\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \frac{2}{3}\delta_{ij}\frac{\partial u_k}{\partial x_k} \right]$$

where: μ_t - dynamic (molecular) viscosity, u_i - components of velocity vector, δ_{ij} - Kronecker's delta.

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) [5,6].

Earlier test calculations have shown that this technique is acceptable for calculation period of time can reliably consider large-scale turbulent fluctuations throughout the flow path of the water turbine and estimate their frequency response characteristics.

A few words about the numerical approach. Discretization of transport equation was carried out by the control volume method on unstructured grids. Relation of velocity and pressure fields for incompressible fluid was realized using the procedure implemented by SIMPLE-C. For the approximation of the convective terms in the equation for momentum components scheme Quick (Leonardo scheme) was used. For the approximation of convective terms in the equations for turbulent characteristics up-wind scheme of the second order was used. Unsteady terms are approximated by the implicit scheme of the 2nd accuracy order. The diffusion terms are approximated by the scheme of the 2nd order.

3 Francis turbine

The section presents the results of simulation of the flow in turbine Francis-99. Three operation point proposed for the Francis-99 workshop are calculated. Geometry data provided by the workshop committee were used [9]. Computational domain includes the runner and the draft tube. The mesh consists 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 corresponds 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 [10]. Fig. 1 shows velocity magnitude in central cross-section for three operation points. Flow of HL and BEP modes are similar. Narrow concentrated vortex is formed under the runner tip. On the other hand, at PL 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.

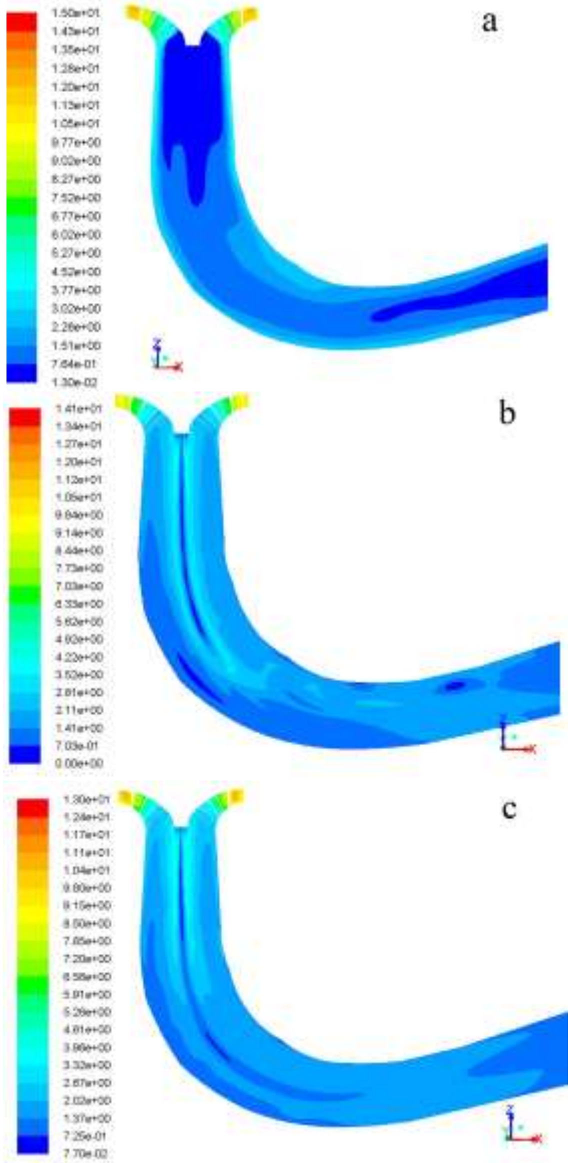
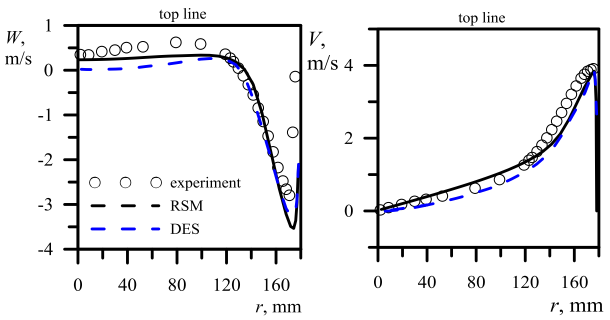
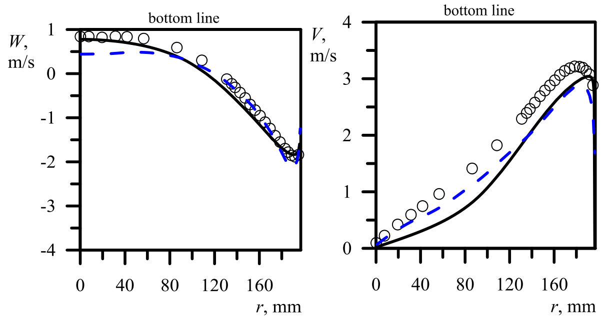


Fig. 1 Velocity magnitude in central plane; *a*) part load, *b*) best efficiency point, *c*) high load



a



b

Fig. 2 Axial and swirl velocity components in the draft tube, part load; *a*) top line, *b*) bottom line

4 Aerodynamic stand

The section presents the results of simulation of the flow in the aerodynamic turbine.

The aerodynamic contour of the stand contains all the basic functionality of the hydro turbine units: spiral case, guide vane, runner and draft tube.

For the rotation of the impeller used a computer-controlled actuator that allows measurements on different modes of hydro turbines. To create and regulate the incoming pressure of a centrifugal pump.

The calculations used unstructured grids of varying detail.

Fig. 3 shows the structure of the flow in the space runner by a pressure iso-surface.

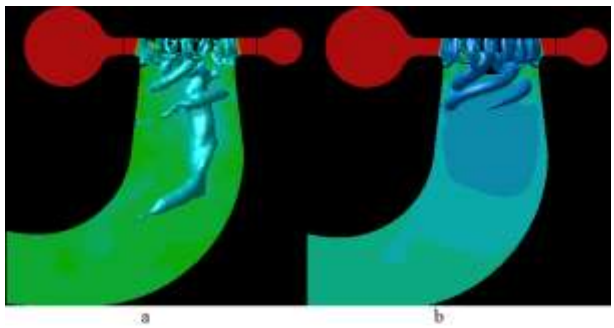


Fig. 3 Vortex flow structure in aerodynamic turbine a) DES, b) RSM

Fig. 4-5 show velocity components in the draft tube. The results of the calculations are in close agreement with experimental data for Reynolds stress model and DES method.

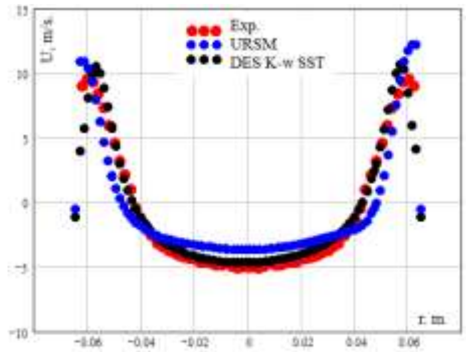


Fig. 4 Axial velocity in the draft tube in aerodynamic turbine

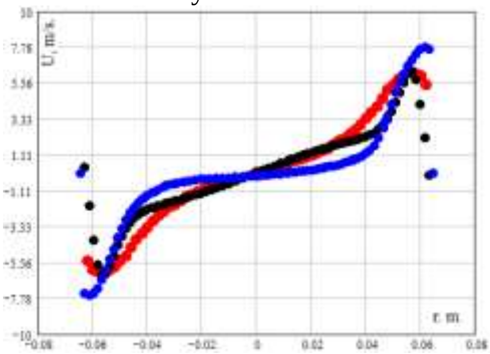


Fig. 5 Tangential velocity in the draft tube in aerodynamic turbine

5 Conclusion

The paper proposed and tested a numerical method for the simulation of flow in hydraulic turbines.

Designed and assembled a unique aerodynamic stand, which is a miniature copy of the actual hydroelectric power. At the booth, a series of experiments simulating different modes of hydroelectric works. We measure the velocity components and the pressure pulsation in the draft tube diffuser.

Using numerical simulation technique of carrying current in the aerodynamic stand. The comparison of the calculated and experimental data. Good coordination of the calculated and experimental data suggests the possibility of the use of numerical methods for the application of engineering problems. Moreover, use the aerodynamic stand for construct the structural elements Designing and testing of numerical methods.

Acknowledgements

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.

[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] 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

[4] 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.

[5] A.V. Minakov, A.V. Sentyabov, D.V. Platonov, A.A. Dekterev, A.V. Zakharov The numerical simulation of low frequency pressure pulsations in the high-head Francis turbine, Computer and fluids, Vol. pp. 197-205

[6] A.V. Minakov, A.V. Sentyabov, D.V. Platonov, A.A. Dekterev, A.V. Zakharov, The analysis of unsteady flow structure and low frequency pressure pulsations in the high-head Francis turbines, International Journal of Heat and Fluid Flow, Vol. 53, pp. 183–194

[7] 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.

[8] 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.

[9] 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.

[10] 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.