Computers & Fluids 111 (2015) 197–205

Contents lists available at ScienceDirect

Computers & Fluids

journal homepage: www.elsevier.com/locate/compfluid

The numerical simulation of low frequency pressure pulsations in the high-head Francis turbine

A.V. Minakov [a,b,*], D.V. Platonov [a], A.A. Dekterev [a,b], A.V. Sentyabov [a,b], A.V. Zakharov [c]

[a] Siberian Federal University, Krasnoyarsk, Russia
[b] Institute of Thermophysics SB RAS, Novosibirsk, Russia
[c] OJSC "Power Machines" (LMZ), St. Petersburg, Russia

ARTICLE INFO

Article history:
Received 14 July 2014
Received in revised form 22 December 2014
Accepted 17 January 2015
Available online 7 February 2015

Keywords:
Francis turbine
Pressure pulsation
Precessing vortex rope
Turbulence
Mathematical model
Computational fluid dynamics

ABSTRACT

Numerical technique for calculating pressure pulsations in the high-head hydro turbines is presented in the paper. The numerical technique is based on the use of DES turbulence model and the approach of "frozen rotor". This technique is applied for simulating unsteady turbulent flow in a flow path of a high head hydropower plant. The flow structure downstream the runner and its influence on the intensity of non-stationary processes is analyzed. It is shown that the low-frequency pressure pulsations in water turbines are mainly caused by the precessing vortex rope downstream the runner. The vortex dynamics basically determines the pulsational characteristics of turbine operation. It is found that the averaged velocity components of flow downstream the runner can serve as indicators, which characterize the intensity of transient phenomena in the draft tube of water turbine. The comparison of simulation results with the experimental data shows reasonable agreement in the integral characteristics (efficiency, flow rate, power) as well as pulsational characteristics (intensity and frequency) of the flow and turbine operation.

1. Introduction

An important task of hydropower plant is the regulation of power in the energy system. During load changing, hydraulic units repeatedly undergo off-design modes of operation. In these modes, the flow remains essentially swirling after passing through the water turbine runner. The instability of swirling flow results in the appearance of intense low-frequency hydrodynamic pulsations, which threaten the safety and reliability of turbine structure.

The improvement of hydraulic machines efficiency and failure tolerance is impossible without studying the physical mechanisms of hydrodynamic processes, among which an important role is played by the transient phenomena associated with the formation of large-scale vortex structures. One of the mechanisms responsible for generating flow pulsations is the precessing vortex rope. It is formed downstream the runner under part-load or over-load conditions when the flow has a large residual swirl after passing through the runner [1,2].

The precession of vortex rope can be a serious danger for hydraulic equipment due to powerful flow pulsations that lead to strong vibrations of turbine structure. It the case of resonance, it can result in the turbine structural failure. Pressure pulsations generated by precessing vortex rope may also affect the cavitation process and enhance cavitation erosion. To predict the resonance phenomena and find the ways of suppressing instability, one needs the detailed information about the characteristics of pulsation regimes and flow structure. It should be noted that the working approaches must meet the requirement of minimizing energy losses (increasing the efficiency of water turbine). It can be realized only through in-depth understanding of hydrodynamic processes occurring in the flow path of a water turbine.

The precession of vortex rope has been studied for quite a long time. In the early years of the XX century, power fluctuations on hydraulic power plants were observed. The studies of model plants revealed the transient behavior of vortex flow in the draft tube. It was shown that the flow pulsations in the draft tube are responsible for low frequency variations of hydraulic unit characteristics.

The development of computer technology has made it possible to apply modern methods of computational fluid dynamics to the description of turbulent flows in geometrically complex spatial objects, such as water turbine flow path. In recent years, there appeared a large number of works devoted to three-dimensional simulation of various processes in water turbines [2–9,13,14]. Here one can mention the papers of Doerfler [2], Ruprecht [4], and

* Corresponding author at: Siberian Federal University, Krasnoyarsk, Russia. Tel.: +7 902 9436822; fax: +7 391 2913012.
E-mail address: tov-andrey@yandex.ru (A.V. Minakov).

http://dx.doi.org/10.1016/j.compfluid.2015.01.007
0045-7930/

Avellan [6], who made a significant contribution to the development of computational modeling techniques with application to water turbines. The researchers from the Institute of Thermophysics SB RAS carried out experimental and numerical studies of turbulent swirling flow structure in a model draft tube [7]. A good agreement between the calculated and experimental data was obtained. It was found that a strong non-uniform flow downstream the tube elbow greatly reduces the capacity of the draft tube. Numerical simulation of flow in the Turbine-99 draft tube on the basis of different turbulence models was performed in [8]. It was shown that the widely used standard k–ε and k–ε Chen models did not correctly reproduce the pressure minimum in the draft tube elbow. The best results were demonstrated by the k–ω SST model, which provided correct pressure distribution and good description of flow in the initial section. The simulation of unsteady three-dimensional flow in a flow path of similar high-head hydro turbine (head of 215 m) was carried out in [9]. The comparison between calculations and experimental data showed a reasonable agreement in both the local flow parameters (velocity profiles and pressure distribution on the walls) and integral turbine characteristics (efficiency and flow rate).

Now it becomes clear that the numerical simulation can serve not only as an alternative approach to experiment, with which it is possible to calculate the integral characteristics of the water turbine, but it can also provide fundamentally new possibilities for analyzing the local flow structure and predicting the pulsation characteristics. At the same time, there is always a question of the adequacy of such numerical results to physical reality. The present work has two objectives. The first one is to test different numerical approaches by comparing the calculated results with the experimental data and find the best technique for simulating low-frequency pulsations in water turbines. The second objective is to analyze the local structure of the flow in a hydraulic unit and its impact on the intensity of transient processes.

2. Mathematical model and numerical method

When describing the flows in a flow path of water turbines, one has to face several problems. The first challenge is related to modeling turbulence in channels of complex geometry and strong swirling flow. The engineering calculations require turbulence models, which can accurately describe not only the mean fields, but also large-scale flow fluctuations. The k–ε and k–ω turbulence models widely spread in engineering calculations poorly describe such flows. To improve the modeling of turbulent swirling flows, researchers are trying either to modify the existing URANS turbulence models or to use techniques, which resolve large-scale turbulent eddies (LES, DES).

Swirling flow in a water turbine can be accompanied by the precession of vortex core. For modeling this phenomenon, it is necessary to use non-stationary eddy resolving simulation techniques such as the method of large eddy simulation (LES). However, its use requires a very fine grid, especially near the walls. Meanwhile, RANS models are rather time-saving and accurately describe the boundary layers. To combine the advantages of these approaches, the method of detached eddy simulation (DES) was proposed in [10]. The first DES version was based on the Spalart–Allmaras model. Later on, the DES method was used with other models of turbulence and its various modifications appeared.

In the simulation of water 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, such as dynamic grid, moving grid, and transition to a moving frame of reference. The most common and simple way to model the runner rotation is to use a rotating frame of reference. The transition to a rotating coordinate system allows one to simulate the swirling flow, which passes through a fixed runner. This formulation often referred to as ''frozen rotor'' will be used in the present study. Numerous test calculations showed the correctness of this approach for describing integral and pulsational characteristics of the flow [7–9].

Let us formulate the basic equations, which express the conservation laws in a rotating frame of reference. The continuity equation, which follows from the conservation of mass, has the form

$$\frac{\partial p}{\partial t} + \Delta(\rho \mathbf{v} = 0)$$

The momentum equation expressing the conservation of momentum in a rotating frame of reference is given by

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

where $\mathbf{v}$ is the fluid relative velocity vector, $\boldsymbol{\tau}$ is the viscous stress tensor, Ω is the angular velocity of runner rotation, p is the static pressure, and ρ is the fluid density. In a rotating frame of reference, the Coriolis force and centrifugal force must be added to the right-hand side of the momentum equation.

The components of viscous stress tensor are defined by

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

where μ_t is the dynamic (molecular) viscosity, μ_i are the components of velocity vector, δ_{ij} is the Kronecker symbol.

When constructing two-parametric turbulence models, the Boussinesq hypothesis of isotropic turbulent viscosity is used for defining the components of Reynolds stress tensor $\boldsymbol{\tau}^t$:

$$-\rho\overline{\mathbf{v}'\cdot\mathbf{v}'} = \mu_t\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \frac{2}{3}\left(\rho k + \mu_t\frac{\partial u_i}{\partial x_i}\right)\delta_{ij}$$

where μ_t is the turbulent viscosity, k is the turbulent kinetic energy.

In this paper, we use the DES method based on k–ω SST Menter's model [11]. The shear-stress transport (SST) k–ω model was developed by Menter to effectively blend the robust and accurate formulation of k–ω model in the near-wall region with the k–ε model in free stream region. The standard equations of k–ω SST Menter's model have the form [11]:

$$\frac{\partial \rho k}{\partial t} + \nabla(\rho k v) = \nabla(\Gamma_k \nabla k) + G_k - Y_k,$$
$$\frac{\partial \rho \omega}{\partial t} + \nabla(\rho \omega v) = \nabla(\Gamma_\omega \nabla \omega) + G_\omega - Y_\omega + D_\omega.$$

Here k is the turbulence kinetic energy, ω is the specific dissipation rate, the term G_k represents the production of turbulence kinetic energy, G_ω represents the generation of ω, Γ_k and Γ_ω are the effective diffusivities of k and ω, respectively, Y_k and Y_ω correspond to the dissipation of k and ω due to turbulence, respectively, and D_ω is the cross-diffusion term.

The dissipation term of turbulent kinetic energy is modified for the DES turbulence model as described in the reference [12]. In this model, the dissipative term in the equation for k is modified by means of a switcher F_{DES}:

$$\frac{\partial \rho k}{\partial t} + \nabla(\rho k \mathbf{v}) = \nabla(\Gamma_k \nabla k) + G_k - Y_k \cdot F_{\mathrm{DES}},$$

$$F_{\mathrm{DES}} = \max\left(\frac{L_t}{C_{DES}\Delta}, 1\right), \quad L_t = \frac{k^{1/2}}{\beta\omega}, \quad C_{\mathrm{DES}} = 0,61,$$

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