Computers & Fluids 88 (2013) 11–18

Contents lists available at ScienceDirect

Computers & Fluids

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

Numerical simulation and rotor dynamic stability analysis on a large hydraulic turbine

Ru-Zhi Gong, Hong-Jie Wang *, Quan-Zhong Liu, Ling-Feng Shu, Feng-Chen Li

School of Energy Science and Engineering, Harbin Institute of Technology, Harbin 150001, China

ARTICLE INFO

Article history:
Received 18 November 2011
Received in revised form 23 February 2013
Accepted 30 August 2013
Available online 7 September 2013

Keywords:
Fluid machinery
Rotor dynamics
Stability analysis
Hydraulic turbine

ABSTRACT

The rotor dynamic stability, which is a vital problem in large hydraulic turbine, is usually not taken into consideration when a hydraulic turbine is designed. But the rotor dynamic character will influence the performance (efficiency, noise, vibration, etc.), life-span and even the safety of the turbine. The method for analyzing the rotor stability of hydraulic turbine has been studied in this paper. Stability of the rotor system of a real turbine was studied with the theory of rotor dynamics in this paper. The way how each factor affects the dynamic character of labyrinth system was also investigated. It can be concluded that the research methods can be widely used in the dynamic analysis of the rotor system with gap flow.

1. Introduction

It has been discussed a lot about the application of computational fluid dynamics (CFD) method on the flow simulation of hydraulic turbines in the published literatures so far. Iwatsubo [1] evaluated the instability forces of labyrinth seals in turbines in 1980. Diewald and Nordmann [2] analyzed the fluid-mechanical interaction in fluid machinery. Nilsson and Davidson [3] compared the computational and the experimental results in a Francis turbine and proved that CFD method is reasonable to simulate the flow field in fluid machinery. And there are other works that studied the application of CFD method in the detection of behaviors of hydroturbines [4,5]. There are also works studying the stability of rotor systems. Alford [6] is the first person who found self-exited whirl in the turbo machinery in 1965. Hashimoto et al. [7,8] examined the effects of wear on steady-state and dynamic character of the theoretical and experimental methods under operating conditions including turbulence. Ren et al. [9] analyzed dynamic stability of a rotor system with labyrinth seals. However, there was few works related to rotor dynamic stability of hydraulic turbines.

In the present paper, the flow field of the whole passage in the turbine is at first simulated and forces on the journal are then obtained through the simulation of the flow in the labyrinth seals clearance. The Muszyska model for rotor system analysis is adopted to describe the flow induced force in the labyrinth seal. The parameters in rotor dynamics equation for the rotor system of the turbine is fitted with the least square fitting method. With the equation obtained, dynamic performance of the turbine rotor system is analyzed and NewMark direct integral method is used. To detect how the structure of the labyrinth seals affect the dynamics of the turbine rotor system, the length, thickness and pressure drop of the labyrinth clearance are changed for the discussions.

2. Case description

For the present study, a real hydraulic turbine for BaiHeTan hydropower station in China is adopted. The detailed geometrical parameters of the turbine are provided by Harbin Institute of Large Electrical Machinery, China. The 3D turbine for flow simulation is shown in Fig. 1. The turbine has 15 runner blades, 23 stay vanes including a special blade and 24 guide vanes with a runner diameter of $D_{ref} = 8700$ mm (runner radius $R_{ref} = 4350$ m). The designed head of the turbine is 120 m, the power is 1000 MW. And the rotational speed at designed condition is 74.5 rpm. Table 1 shows the geometrical parameters of the turbine. It is unusual to have labyrinth seal in hydraulic turbine; however, labyrinth seal is adopted in the selected turbine. The structure of the turbine is shown in Fig. 2 and the labyrinth seal is marked with red[1] circle in the figure.

3. Numerical simulation considerations

The flow path in the turbine is from the spiral case inlet to the draft tube outlet. The water was considered to be incompressible. A

* Corresponding author. Tel.: +86 13836034807.
E-mail address: Wanghongjie@hit.edu.cn (H.-J. Wang).

[1] For interpretation of color in Fig. 2, the reader is referred to the web version of this article.

0045-7930/$ - see front matter
http://dx.doi.org/10.1016/j.compfluid.2013.08.014

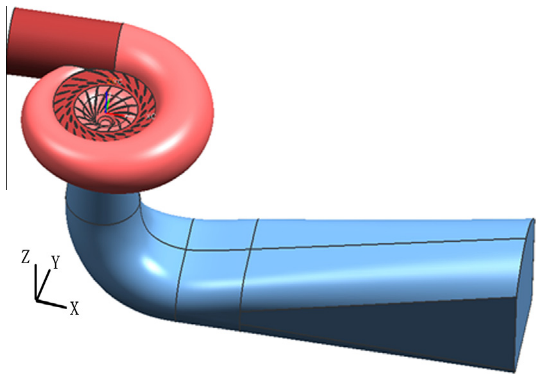

Fig. 1. Model of the hydraulic turbine.

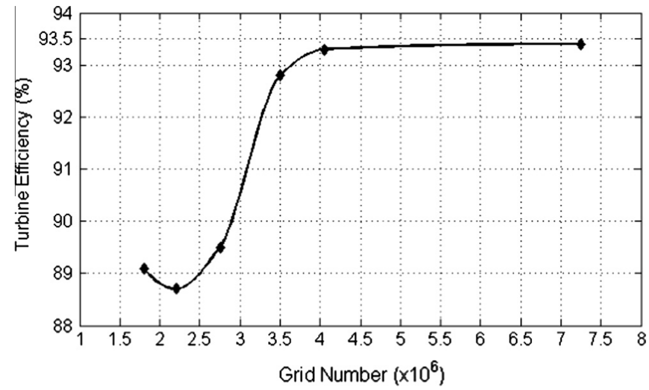

Fig. 3. Grid validation result.

Table 1
Parameters of the hydraulic turbine studied presently.

Parameters	Value
Diameter of the runner, D_1 (mm)	8700
Amount of blades, Z	15
Distribution radius of the guide vanes, D_0 (mm)	$1.145D_1 = 9960$
Amount of guide vanes, Z_0	24
Height of guide vanes, B_0 (mm)	1588
Amount of stay vanes, Z_s	23

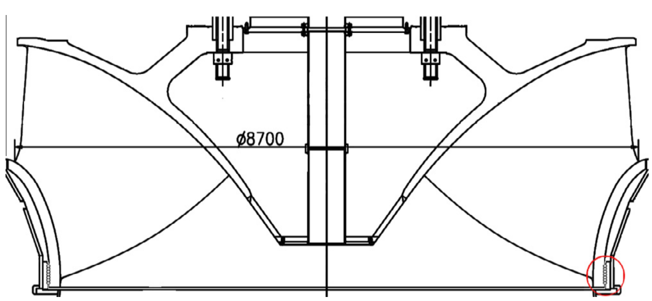

Fig. 2. Structure of the labyrinth seal.

time-dependent Reynolds average Navier–Stokes (RANS) model was chosen for the governing equations of the flow and numerical simulation was performed in order to obtain the flow field in the turbine flow path. The continuity equation and the momentum equations for the flow in RANS-model style are as follows:

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = \frac{1}{\rho}\frac{\partial}{\partial x_j}\left[-p\delta_{ij} + \mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\delta_{ij}\frac{\partial u_k}{\partial x_k}\right) + (-\overline{\rho u_i' u_j'})\right] \tag{2}$$

The flow in hydraulic turbine and the labyrinth seal is very complex. It is impossible to find accurate or approximate analytical solutions. Herein, the above equations are solved by the commercial CFD code FLUENT (Version 12.1.2).

The streamlines in the turbine have large curvature. Fortunately, the re-normalization group (RNG) k–ε turbulence model in FLUENT simulates rotational and swirl flow in the average flow better through improved calculation of the turbulent viscosity. The influence of turbulence in the hydraulic turbine is thus considered with RNG k–ε turbulence model in the present study. To close the governing equation, RNG k–ε turbulence model provides two additional equations:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u_i k)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\alpha_k(\mu + \mu_t)\frac{\partial k}{\partial x_j}\right] + \rho(p_r - \varepsilon) \tag{3}$$

$$\frac{\partial(\rho \varepsilon)}{\partial t} + \frac{\partial(\rho u_i \varepsilon)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\alpha_\varepsilon(\mu + \mu_t)\frac{\partial \varepsilon}{\partial x_j}\right] + \frac{\rho}{k}(C_{1\varepsilon}\varepsilon p_r - C_{2\varepsilon}\varepsilon^2) \tag{4}$$

where,

$$\mu_t = \rho C_u \frac{k^2}{\varepsilon} \tag{5}$$

α_k, α_ε , $C_{\varepsilon 1}$ and $C_{\varepsilon 2}$ are constants and can be determined by experiments and statistical theory. These constants are herein chosen as $\alpha_k = \alpha_\varepsilon = 1.39$, $C_{\varepsilon 1} = 1.42$ and $C_{\varepsilon 2} = 1.68$.

The mesh for simulation was generated by ICEM grid generator. And grid independence is validated, as shown in Fig. 3. The whole path was discretized with an unstructured hybrid mesh of tetrahedron cells. Normally, the fluid flow simulation needs finer grid than structure analysis. So the mesh size was determined according to the recommendation by Ma and Zhou [10] to meet the requirement for fluid flow simulation. It is acceptable when there is no evident change with the increasing number of grid nodes. And the final mesh has 4.05×10^6 nodes.

The interaction between the rotor and stator is simulated with interface. The kind of interface between the rotor and stator is sliding surface. And the hydraulic conversation across the interface is just about 0.0002%. It can be completely neglected. Multi Reference Frame (MRF) model is used for the part of runner. The pressure drop between the inlet and the outlet of the seal is obtained from the results of the flow field of the turbine, which is about 55,000 Pa.

Then the flow field in the seal is simulated with the rotational speed of the shaft and pressure drop obtained as boundary conditions. In the simulation, different rotating speed of the shaft and different shaft offset are taken into consideration. The pressure distributions in the labyrinth seal are acquired. Fig. 4 shows the pressure distribution profile along the circumference under the same rotational speed but different shaft offset. And the eccentricity direction is X coordinate, which means circumference angle is zero (or 2π).

When the offset of the shaft is different, the pressure distribution is different from each other. Because of the eccentricity, the pressure is higher on the side of the shaft deviation, and there is some phase difference, always before the eccentricity direction. The pressure inhomogeneity is increasing with the increase of the offset (x_e) of the shaft.