Advances in Engineering Software 90 (2015) 152–158

Contents lists available at ScienceDirect

Advances in Engineering Software

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

Transient hydrodynamic analysis of the transition process of bulb hydraulic turbine

Yiming Li, Guiqiu Song*, Yunlei Yan

School of Mechanical Engineering & Automation, Northeastern University, 110819, China

ARTICLE INFO

Article history:
Received 4 June 2015
Revised 22 August 2015
Accepted 23 August 2015
Available online 14 September 2015

Keywords:
Rotating machinery
Dynamic mesh
Transition process
Transient flow
Dynamic characteristics

ABSTRACT

In this paper, a semi-implicit method for solving pressure coupled equations used to calculate the flow field, and dynamic mesh were employed to simulate the three-dimensional dynamic turbulent flow of whole flow channels, in start transition process and runaway transition process of bulb hydraulic turbine. In above processes, the transition process of pressure field and flow field were realistically modeled and simulated, which reflects the influence of water flow state, after passing through guide vanes on the inflow of the runner blades, and reveals the flow field variations in the transition process. The numerical results show that: during the start transition process, with the increase of guide vane opening, the flow circulation produced by water flowing through the guide vanes becomes smaller; therefore, the rotational acceleration of the runner decreases and the rotational speed value reduces. In the runaway transition process, the hydraulic turbine operating with load has a sudden load rejection and then, the hydraulic turbine rotational speed is rising with the guide vane closing, leading to pressure pulsations and severe vortex phenomenon, which will cause significant vibrations of swings.

1. Introduction

Bulb hydraulic turbine is a kind of rotating machinery converting hydro energy to kinetic energy. Although bulb hydraulic turbine is used more widely in the development and utilization of hydropower, and is changed to the high-capacity, high specific speed and high efficiency direction, meanwhile its stability remains a problem to be solved.

In the research of hydraulic turbine, fluid dynamics technology has two main applications: the first application is the calculation of the steady flow field in the flow components of the hydraulic turbine. It is mainly used to analyze the flow regimes of each flow channel in the flow field and study the hydraulic losses of each parts, the pressure distribution on the blades surface and its optimization design; secondly, it is used to calculate the unsteady flow field inside the hydraulic turbine, i.e. the instantaneous flow regime of each flow channel, the hydraulic vibrations and the pressure pulsations caused by various factors of instability in each flow channel. Malcevic and Ghattas [1] presented a novel method for simulation of flows with dynamic interfaces based on a Lagrangian flow formulation and dynamic finite element meshes. In the paper by Liu et al. [2], a three-dimensional unsteady turbulent flow is simulated through the whole flow channel of three Gorge Francis hydraulic turbine. The pressure pulsations are predicted at positions in the draft tube, in front of runner, wicket gate, stay vane, and at the inlet of spiral case. Based on CFD code, Zhao et al. [3] used *RNG* k-ε turbulence model to study the inner flow in labyrinth sealing of hydraulic turbine. The flow structure in the seal is obtained, which shows the property of resistance enhancement of labyrinth sealing by turbulence. In the paper by Gao et al. [4], 3-D numerical calculations are performed for flow within the integral flow path of the prototype hydraulic turbine of the Three Gorges Project. The numerical method is proposed based on Reynolds-averaged Navier–Stokes equations. The *RNG* k-ε turbulence model with finite volume algorithm is employed in curvilinear body-fitted coordinate systems. In the paper by Zhang et al. [5], a three-dimensional unsteady turbulent flow in axial-flow pumps was simulated based on Navier–Stoke solver embedded with k-ε *RNG* turbulence model and SIMPLEC algorithm. In the paper by Qian et al. [6], three-dimensional unsteady turbulent flow in the entire flow channel of a model Francis turbine was investigated by a k-ε turbulence model. The validity of the mathematical model is verified by experimental data. Li et al. [7] established a full three-dimensional inverse problem calculation mode of the bulb turbine unit blades and the guide vanes matched with the blades. The calculation mode established the matching relationship between the blades and guide vanes by being given the velocity moment distribution at the guide vanes outlet side and the blades inlet side. In the paper by Zhang et al. [8], CFD methods for unsteady flow simulation are reviewed,

---

* Corresponding author.
E-mail address: song1892@sina.com (G. Song).

http://dx.doi.org/10.1016/j.advengsoft.2015.08.006
0965-9978/

including moving grid generation techniques and unsteady numerical methods. Based on the topology of computational grids, two types of unsteady numerical methods are discussed, e.g. the methods based on structured grids and unstructured/hybrid grids, respectively. Their advantages and drawbacks are analyzed with regard to efficiency, robustness for complex geometries and other features. In the paper by Pope et al. [9], unsteady 3-D CFD simulations was conducted on a multi-bladed vertical axis rotor. As a flow instability, rotating stall was conducted on a pump-turbine by Widmer et al. [10] Wang et al. [11] simulated the internal flow fields and external characteristics at turbine braking condition by the SST k-ω model and SIMPLEC algorithms. Compared with the experimental data, the simulation results were examined. Pang et al. [12] based on the CFD numerical model established, quantitative analysis on the vibration performance of Karman vortex shedding is achieved by using the software FLUENT. In the paper by McTavish et al. [13], an assessment of the performance of a novel vertical axis wind turbine was conducted using RANS CFD simulations. Huang et al. [14] established the coupling computation model between the flow channel transient flow and the three-dimensional flow inside the unit, and successfully realized the calculation of the power station load rejection with this model. In the paper by Trivedi et al. [15], investigations of a Francis model during an emergency shutdown with transition into total load rejection and steady-state measurements under runaway operating conditions were performed.

The transition process of bulb hydraulic turbine refers to the transient process that the bulb hydraulic turbine changes from one working state to another working state, including start transition, stop transition, sudden load increasing transition, sudden load shedding transition, sudden load rejection transition, emergency stop transition, runaway transition, runaway exit transition, generator phase modulation transition, and etc. [16] . A transition process lasts shortly. However, due to substantial changes of working parameters, the dynamic load caused by flow inertia and hydraulic turbine movement greatly affects the safe and stable operation of the entire power system. Moreover, as the courses of the transition processes are different, the hydraulic turbine dynamics problems that may occur are not the same. The course analysis of the transition process of bulb hydraulic turbine is particularly important.

Based on previous studies, this paper mainly analyzed the start transition process and the runaway transition process of the bulb hydraulic turbine. A fine mathematical model of the whole flow channels in the bulb hydraulic turbine was established and analyzed in details with dynamic mesh.

2. Numerical analysis method and key technologies of the flow field

2.1. Flow control equation

In the transition process of bulb hydraulic turbine, algebraic difference between hydro-dynamic moment acting on rotating wheel (M_H) and motor torque (M_f), determines the movement state and law of the rotating parts of the hydraulic unit.

$$M_\mathrm{H} - M_\mathrm{f} = J\frac{\mathrm{d}\omega_\mathrm{H}}{\mathrm{d}t} \tag{1}$$

where, J is moment of inertial of the rotational system; ω is angular velocity.

Hydro-dynamic moment equation of bulb hydraulic turbine:

$$M_\mathrm{H} = \rho Q_\mathrm{H}(v_{u\mathrm{H}1}r_1 - v_{u\mathrm{H}2}r_2) - \iiint\limits_{\Omega} \frac{\partial v_{u\mathrm{H}}r}{\partial t}\mathrm{d}W \tag{2}$$

where, ρ is density; Q_H is dynamic flow; v_{uH1} and v_{uH2} are the circle of dynamic absolute velocity of inlet and outlet on the rotating wheel; r_1 and r_2 are the radius of inlet and outlet on the rotating wheel; $v_{\mathrm{uH}}r$ is dynamic velocity moment; t is time; Ω is body area in rotating wheel; dW is volume of the micro unit of water.

Dynamic axial hydraulic thrust equation of bulb hydraulic turbine:

$$S_\mathrm{ZH} = (p_\mathrm{H1} - p_\mathrm{H2})F - \rho\iiint\limits_{\Omega} \frac{\partial v_\mathrm{ZH}r}{\partial t}\mathrm{d}W \tag{3}$$

where, p_H1 and p_H2 are hydrodynamic pressures at inlet and outlet of the rotating wheel;v_ZH is dynamic absolute velocity in y direction.

2.2. Pressure correction theory

Pressure correction theory is the semi-implicit method for solving the pressure coupled equations. This equation was proposed by Patankar and Spalding in 1972 [17], and is one of the main numerical methods for solving incompressible flow field. The fundamental principle of the equation is based on the fact that, in the process from guessing to correcting, the calculation of the pressure field, based on staggered grid, is used to solve the momentum equations. First, a pressure field to solve the momentum equation is assumed, so as to get the velocity and the pressure correction equation, from which a pressure correction value will be derived. Second, according to the corrected pressure field, a new velocity field will be solved. Finally, a velocity field convergence checking scheme will be performed. If it converges, the obtained corrected pressure value will be used to form a new pressure field. Otherwise the calculation will be repeated until a convergence is achieved.

2.3. Dynamic mesh technique

Dynamic mesh model is used to simulate the fluid flow, where the shape of computational domain shape changes with time due to the movement of computational domain boundary. If the boundary motion law and motion state are known, it is an active dynamic mesh. Otherwise the movement is determined by the flow characteristics, when it is a passive dynamic mesh.

Unstructured dynamic mesh technique is the key to solve the rotary coupling problem under the unstructured mesh framework. According to the rigor degree and the deformation amount of the boundary movement, unstructured mesh can be achieved by method of mesh deformation, method combining layer advancing and mesh deformation partition or deformation plus local or whole mesh reconstruction method [18]. Here, the unstructured mesh reconstruction (static-regrinding) generation technique, means regenerating the mesh at each time step, which has a simple concept and can be applied for large deformation or large displacement problems. Because of the interpolation in each step to reconstruct the mesh, an interpolation error is inevitably introduced, and the interpolation relation between time layers needs to be regained. Thus, the amount of computation time increases. Grvber and Carstens [19] used this technique to solve the forced vibration problem of a turbine engine blades. Schulze [20] applied it to deal with the air elastic problem of the wing movement.

In this paper, in the analysis of the unsteady flow field in the runaway transition process of the bulb hydraulic turbine, the active mesh method was applied as the boundary motion state is known, in the analysis of unsteady flow field in the start transition process of the bulb hydraulic turbine, as the boundary motion law is unknown, the active mesh method, the unstructured mesh reconstruction generation technique, were employed.

3. Mesh model of the bulb hydraulic turbine

The bulb hydraulic turbine is composed of four parts: bulb turbine body, tube-type seat, guide vanes and runner. The computational domain of flow field analysis in the transition process includes the