Renewable Energy 65 (2014) 227–235

Contents lists available at ScienceDirect

Renewable Energy

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

Transient numerical analysis of rotor–stator interaction in a Francis turbine

Anup KC [a], Bhola Thapa [b], Young-Ho Lee [c,*]

[a] Flow Informatics Lab, Department of Mechanical & Energy System Engineering, Korea Maritime & Ocean University, Busan, South Korea
[b] Department of Mechanical Engineering, School of Engineering, Kathmandu University, Dhulikhel, Nepal
[c] Department of Mechanical and Energy System Engineering, Korea Maritime & Ocean University, 606-791 Busan, South Korea

ARTICLE INFO

Article history:
Received 20 February 2013
Accepted 12 September 2013
Available online 18 October 2013

Keywords:
Francis
Unsteady flow
Rotor–stator interaction
Vortex rope
CFD

ABSTRACT

Pressure fluctuation due to rotor–stator interaction and occurrence of vortex rope in draft tube at partial load operation are obvious phenomena in Francis type reaction hydro turbines. These hydrodynamic effects are important issues and should be addressed during the design of hydraulic machines. A 3-dimensional transient state turbulent flow simulation in the entire flow passage of a 70 kW-Francis turbine having specific speed of 203.1 is conducted to investigate the rotor–stator interaction by adopting $k\omega$ based SST turbulence model. The commercial 3D Navier–Stokes CFD solver Ansys-CFX is utilized to study the flow through this vertical shaft Francis turbine in its stationary and transient passages, at 100% optimum load and 72% of part load. The investigated turbine consists of a spiral casing with 16 guide vanes, 8 stay vanes, a runner with 13 blades and a draft tube. With a time step of 2° of a rotational period of the runner for 10 full rotations, the time dependent pressure and torque variations are monitored at the selected locations during the unsteady state calculation. A periodical behavior is observed for the pressure distribution in guide vanes, runner blades and torque in the runner blades. The pressure distribution curve in runner blades reveals the two dominating frequencies – the lower peaks due to runner speed and the upper peaks corresponding to the number of guide vanes interacting with the flow. The flow acceleration toward inside of the runner is depicted by the expanding wakes behind the stay vanes. Vortex rope is observed in draft tube, downstream the runner, at part-load operation.

1. Introduction

The ever escalating energy demand, quest for green energy and iterative development in the hydropower sector have pushed the generation of hydropower towards a lost-cost and more flexible system that can operate in wider hydrodynamic regimes without compromising its performance. With robust and compact designs operating in wide regimes, new Francis hydro turbines are required to handle effects like cavitation, pressure fluctuations, rotor–stator interactions and vortex shedding effectively. An unsteady state analysis of hydro turbine can be useful in predicting and analyzing instability in turbine due to unsteady flow field and to develop mitigating techniques to minimize the effects of these phenomena. With the rapid development of computational technology and its versatility in computing fluid dynamics, computational fluid dynamics (CFD) has emerged as a reliable and economic tool in analyzing the performance of the turbines and revitalizing them. With its higher accuracy and flexibility in simulating diverse flow conditions and ability to analyze complex three-dimensional phenomena occurring in fluid machineries, it has now become a standard to evaluate the feasibility of hydro turbine and its components minimizing much of the time and cost-intensive experimental investigations. Over the past few years, CFD has been used routinely within the R&D process of hydraulic machinery [1].

In hydro machineries, a small improvement in the geometry of rotating components can have a large positive effect from the point of operation and maintenance cost. To identify such room for improvement, it is a good idea to consider the interaction between the components of turbine. Since turbines are tailor-made to suit specific site conditions, the flow behavior in turbine must be well predicted to establish its safe operation range and to develop its reliability of performance. Among all hydraulic turbines employed for hydro energy conversion, wide operating range of Francis turbine enables it to be used for varying range of small to large

---

* Corresponding author. Tel.: +82 51 410 4293; fax: +82 51 403 0381.
E-mail address: lyh@kmou.ac.kr (Y.-H. Lee).

0960-1481/$ – see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.renene.2013.09.013

hydropower plant. With reaction turbines, the components of Francis turbine experience strong three-dimensional rotational flow due to the change in flow configuration from radial to axial direction. The flow is basically turbulent and unsteady due to rotor–stator interaction where higher-order effects and secondary flows pose a dominant influence on the machine and its performance characteristics. At part-load operation of the turbine, swirling flow appears downstream the runner which is absorbed by draft tube resulting in genesis of vortex rope. The consequences of vortex so developed are pressure pulsation, axial and radial forces, torque fluctuation, structure vibration – the unpleasant features exhibited by Francis turbine that occur due to the associated unsteadiness in the flow [2].

Flow in the different components of turbine is interrelated and reacts mutually; especially the components like guide vanes, runner and draft tube have strong influence on one another due to the dynamic forces and resulting vibration. The prediction and understanding of the flow behavior in casing, runner and draft tube region hold key importance in redefining the flow and developing better flow techniques to overcome the flow instabilities and the detrimental interaction between the components. While a steady-state analysis can predict turbine performance parameters like efficiency, cavitation, and hydraulic losses, the analysis of dynamic forces demands calculation of unsteady flow with advanced turbulence model to achieve accurate results. Shear stress model (SST) [1], realizable k–ε [3], standard k–ε including hybrid Kato–Launder correction [4] are of good choices for turbulence model to analyze rotor–stator interaction and pressure pulsation but more sophisticated turbulence models like Renormalization Group (RNG k–ε) [5,6], extended k–ε [7], large eddy simulation (LES), scale-adaptive simulation (SAS-SST), Reynolds stress models (RSM) [8] are opted for capturing draft tube vortex rope more accurately. These turbulence models and the numerical simulation as a whole require finer grid, extended computational effort and CPU time. In transient-state flow analysis, various hydrodynamic parameters are the function of time and several unsteady phenomena transpire due to additional inertial forces. This paper presents the method and results obtained from the simulation of the unsteady flow field resulting from the rotor–stator interaction by considering the full fluid passage of Francis turbine designed for a small scale hydropower plant. The interaction between the stationary and rotating domains and the influence of the fluid flow in various components are studied in this time-dependent analysis.

2. Numerical procedure

The commercial 3D Reynolds averaged Navier–Stokes CFD-solver Ansys-CFX v13.0 is utilized to investigate the time varying unsteady flow through a vertical shaft 70 kW Francis turbine in its stationary and transient passages at 100% load (0.5 m^3/s) and 72% load (0.36 m^3/s) where the transient flow fields in casing, runner and draft tube are simulated. The efficiency of the turbine at 0.36 m^3/s is evaluated as 58.48% from steady state analysis.

The hydraulic parameters of the investigated turbine are: head $(H) = 18$ m, rotational speed $(N) = 900$ rpm, and rotational frequency $(f_n) = 15$ Hz. Owing to non-uniform inflow from spiral casing and unequal pitching of guide vanes and runner, the computational grid of entire turbine with all channels of runner and of tandem cascade is considered, without applying any periodicity in any components, as shown in Fig. 1. This arrangement represents the most general approach in predicting the rotor–stator interaction. For calculation, the computation domain is divided into 3 components – a spiral casing with 16 guide vanes, 8 stay vanes and a head cover, a runner of 13 blades with crown and band and a draft tube. The 3D models of the components are generated in Unigraphics NX6. Independent computational grid for casing, runner and draft tube are generated with unstructured tetrahedral elements along with prism layers at the solid boundary to capture the boundary layer separation. The whole computational grid is comprised of 3.6 million grid nodes. The entire fluid domain of the turbine is formed by combining the components with an interface between casing and runner and runner and draft tube, each, using general grid interface (GGI) method for mesh connection. Spiral casing with guide vanes and stay vanes, and draft tube are stationary component while runner is the rotating component. The hydraulic head in terms of relative total pressure at the inlet of spiral casing and a free outflow of total mass flow rate at the outlet of the draft tube are given as inlet boundary condition and outlet boundary condition respectively.

For transient analysis, the time step of 2° rotation of runner is taken for 10 full rotations of the runner. So the time step is 0.00037037 s, corresponding to 1/180 of the runner rotational period and the total computational time is 0.667 s of the runner. A second order backward Euler is used as a transient scheme with a high-resolution advection scheme. The maximum loop coefficient is taken as 3. The k–ω based shear stress transport (SST) model of Menter is applied for turbulence treatment to study the rotor–stator interaction [9]. The transport equations for the SST model are expressed below where the turbulent kinetic energy 'k' and turbulent frequency 'ω' are computed by using the following relations:

For turbulence kinetic energy,

$$\frac{\partial k}{\partial t} + \nabla \cdot (uk) = P_k - \beta^* k\omega + \nabla \cdot [(\nu + \sigma_k \nu_T)\nabla k] \tag{1}$$

where P_k is the production limiter.

For specific dissipation rate

$$\frac{\partial \omega}{\partial t} + \nabla \cdot (u\omega) = \alpha S^2 - \beta\omega^2 + \nabla \cdot [(\nu + \sigma_\omega \nu_T)\nabla\omega] + 2(1 - F_1)\sigma_{\omega 2}\frac{1}{\omega}\nabla k \cdot \nabla\omega \tag{2}$$

The recommended values for the constants in the above equations are:

$$\alpha_1 = 5/9,\ \alpha_2 = 0.44,\ \beta_1 = 0.075,\ \beta_2 = 0.0828,$$
$$\beta' = \frac{9}{100},\ \sigma_{k1} = 1.176,\ \sigma_{k2} = 1,\ \sigma_{\omega 1} = 2,\ \sigma_{\omega 2} = 1.168$$

The governing equations are discretized by finite volume method in spatial direction and by finite difference method in temporal direction. The SST model is recommended for high accuracy boundary layer simulation and flow separation. SST model has merit for its good behavior in adverse pressure gradients and separating flow. Since this study is more focused in understanding the interaction between the components and the associated flow features rather than accurate prediction of vortex breakdown, k–ω based SST turbulence treatment renders prominent turbulence levels in the region with large normal strains, like stagnation regions and regions with strong flow acceleration. This tendency is much lesser addressed by normal k–ε model. The solution obtained from 3D steady state flow with SST treated turbulence model and a frozen rotor approach is specified as an initial guess for the transient calculation to give it a smooth start-up.

3. Numerical results

The overall results of the analysis are presented for full load and partial load operations. Figs. 2 and 3 depict the velocity vector plot