PDF-XChange Viewer
Click to buy NOW!
www.docu-track.com

A Full Three Dimensional N.S. Numerical Simulation of Flow Field Inside a Power Plant Kaplan Turbine Using Some Model Test Turbine Hill Chart Points

S.M. Hosseinalipour[a] A. Raja[b] and S. Hajikhani[c]

[a]*School of Mechanical Engineering ,IUST, Tehran, Iran*
[b]*School of Mechanical Engineering ,IUST, CAEC, Tehran, Iran*
[c]*School of Mechanical Engineering , shahid Rajaee teacher training university (srttu), Tehran, Iran*

Abstract. A full three dimensional Navier – Stokes numerical simulation has been performed for performance analysis of a Kaplan turbine which is installed in one of the Irans south dams. No simplifications have been enforced in the simulation. The numerical results have been evaluated using some integral parameters such as the turbine efficiency via comparing the results with existing experimental data from the prototype Hill chart. In part of this study the numerical simulations were performed in order to calculate the prototype turbine efficiencies in some specific points which comes from the scaling up of the model efficiency that are available in the model experimental Hill chart. The results are very promising which shows the good ability of the numerical techniques for resolving the flow characteristics in these kind of complex geometries. A parametric study regarding the evaluation of turbine performance in three different runner angles of the prototype is also performed and the results are cited in this paper.

Keywords: Kaplan turbine, Turbulent flow simulation, Efficiency, Scale up, Computational Fluid Dynamics, Hill chart.
PACS: 47.85.Dh

INTRODUCTION

Due to the huge size and specific conditions, the water turbines are not mass producted. It is very difficult and impossible, under many circumstances, to change the geometry of a water turbine, because any change may encounter with so many limitations and large expenses. Therefore it is very vital to have a complete and exact picture of a water turbine performance in advance. To achieve this, it is necessary to build and test a similar model of the prototype turbine. The prototype and the model have geometrically similar parts, such as the spiral case, stay vanes, guide vanes, runner and the draft tube.

The experimental tests are run on the model and the characteristics of the model are summarized on a Hill chart. Then the scaling relations are used in order to find the prototype Hill chart. The recent advances in computing hardware and numerical techniques have provided enough abilities for studying the operation of prototype water turbines in full scale. During the past decade the computational fluid dynamics technique has been used for simulating the fluid flow in water turbines. This technique may use as a completing or competing tool besides the experimental techniques.

The performance of Kaplan turbines has been under intense investigations during the past two decades. Balint et al. [1] have simulated the whole hydraulic circuit of a Kaplan turbine in order to generate detailed information about the flow from inlet to the outlet of the turbine assembly. Nilsson and Davidson [2] used the control volume method for simulating the flow field inside a Francis and a Kaplan turbine. They compared their numerical results with existing measured ones at different parts of the two turbines. Their investigation was focused on the runner parts.

In 2006 Nilsson [3] has conducted steady and unsteady numerical simulations for water flow inside the draft tube and turbine runner. Balint et al [4] calculated the steady tangential and radial velocity components at the runner inlet, which are important information for design or optimization of the runner blades. They included the distributer and spiral casing in their simulation.