Performance Investigation of a Kaplan Hydraulic Turbine Draft Tube using CFD Techniques

Samaneh Hajikhani
School of Mechanical Engineering
Shahid Rajaee Teacher Training University (SRTTU)
Tehran, Iran
Samaneh.hajikhani@gmail.com

S. Mostafa Hosseinalipour
School of Mechanical Engineering
Iran University of Science & Technology (IUST)
Tehran, Iran
Alipour@iust.ac.ir

Amir Raja
School of Mechanical Engineering
Iran University of Science & Technology, CAEC
Tehran, Iran
Amir.Raja@mecheng.iust.ac.ir

Kamran Mobini
School of Mechanical Engineering
Shahid Rajaee Teacher Training University (SRTTU)
Tehran, Iran
kamobini@yahoo.com

Abstract—**The draft tube has an important role in the overall efficiency of water turbines. The enhancement of draft tube performance has been under considerable attentions during the past decade. As the flow field characteristics inside the draft tube can reveals the possible trends for enhancing its performance therefore the computational fluid dynamics techniques have been used extensively in this regard. A full 3 – D steady turbulent flow numerical simulation has been conducted for a prototype hydraulic Kaplan turbine. The effect of some key parameters such as guide vane opening and runner blade angles on the draft tube performance are studied. The results are presented as the variation of draft tube pressure recovery factor in terms of these parameters. Some features of draft tube flow field are also reported.**

Keywords-component; kaplan turbines; turbulent flow simulation; CFD; draft tube; Hill chart

I. Introduction

Draft tube is one of the most important parts of the reaction water turbines which connect the runner exit to the tail water basin. It provides the possibility of mounting the turbine in a level higher than the tail water level with the minimum head loss. The main purpose of the draft tube is to convert the kinetic energy of the flow leaving the runner into pressure energy by an increase of the area perpendicular to the main flow direction, and thereby increase the efficiency of the turbine. The efficiency of hydraulic reaction turbines considerably depends on the draft tube design. The role of an appropriate design for the draft tube becomes more significant for low head and large mass flow rate conditions, as at these cases the draft tube losses can grow considerably (up to 50%) [1]. To recover as much energy as possible in the draft tube, the goal is to increase the outlet area as possible before large scale separation occurs and best performance is actually considered to take place when there is some transitional separation. The amount of flow swirl entering the draft tube has a considerable effect on its performance. This amount usually depends on the flow conditions which leaves the runner blades. There are several wake regions: after the runner blades, the runner hub and possibly the guide vanes, that all will contribute to the energy distribution that enters the draft tube [2]. Therefore the flow conditions inside the runner and at the exit have a crucial role on the draft tube performance. Besides the inlet flow conditions, the draft tube different parts geometries also play important roles in the draft tube performance. Different performance indices have been introduced by researchers in order to specify the draft tube performance among which the pressure recovery factor is the one which has been under consideration in our study.

According to the above discussion, the draft tube performance has been under considerable numerical and experimental investigations during the past decade. A couple of these studies and their results are summarized in the following.

Prasad V. et al. [3] carried out the numerical flow simulation for 3-D viscous turbulent flow in elbow draft tube by varying its parameters like length and height at different mass flow rate using Ansys CFX code. The draft tube efficiencies and losses are computed from pressure and velocity distributions and presented graphically to study the effect of geometrical parameters on draft tube performance. According to the simulation results, they claimed that both height and length of draft tube have significant effect on performance of elbow draft tube. Kuppinger K. [4] performed a numerical analysis for draft tube flow with the intention to increase the performance of two existing hydroelectric power plants. He showed that installing guide plates in the S – shaped draft tube, helps to suppress the separations. He also reported that the design of the cone at the beginning of the draft tube (outlet of the turbine runner) plays a role which should not be disregarded.