INVESTIGATIONS OF PARAMETERS AFFECTING DRAFT TUBE PERFORMANCE

A THESIS

Submitted in partial fulfilment of the requirements for the award of the degree

of

DOCTOR OF PHILOSOPHY

in

ALTERNATE HYDRO ENERGY CENTRE

by

ALI ABBAS

ALTERNATE HYDRO ENERGY CENTRE
INDIAN INSTITUTE OF TECHNOLOGY ROORKEE
ROORKEE – 247 667 (INDIA)
JUNE, 2018

ABSTRACT

Hydropower is one of the important and proven renewable energy sources to meet the growing needs of energy. Hydraulic turbine is one of the important components for harnessing hydro power. Modern hydraulic turbines are being designed by using modern tools the higher efficiency even up to 95% are being achieved. However, old power plants are to be renovated and upgraded for achieving the modern efficiency from 80%-85% to 95%. Enhancement of the efficiency in hydraulic turbine is a challenging task, which is greatly affected by the performance of the draft tubes in old hydropower plants. The straight conical shape of the draft tube has a good hydraulic characteristic, generally used only at the small and medium diameters of the runner, because of the high construction cost of the long vertical draft tubes. While the elbow draft tube is used at the hydraulic turbine with the large diameter of the runner to reduce the excavation depth of the draft tube. Traditionally, the design of the draft tubes has been carried out using the experience of the designers and the laboratory model tests. The efficiency of a reaction turbine is significantly affected by performance of its draft tube. At low heads and high flow rates, draft tube losses are considerably large. Draft tube serves two purposes (a) to recover kinetic energy leaving the runner into pressure energy, which would be otherwise lost without the draft tube and (b) to install the turbine runner below tail water level to avoid cavitation, without affecting the net head acting on the turbine.

The main shape of the draft tube is essentially a diffuser. The draft tube is one of the most challenging parts to describe from a fluid flow perspective, due to the interaction of many complex flow features such as unsteadiness, turbulence, separation, curvature streamline, secondary flow, swirl, and vortex breakdown.

For rehabilitation projects, runner replacement by a new design improves the flow velocity field at the draft tube inlet. However, there is a limitation to improve the performance of turbine by replacement of runner itself. If the flow velocity inside the elbow is too high or leading edge of pier nose is not well adapted or the diffuser divergence is too high, even with a new runner design it is not possible to avoid flow separation and high losses inside the draft tube. In such case, the hydraulic profiles of draft tube are analyzed and optimized to improve the flow with the runner. Special

attention need to be paid for the draft tube design in case of unit rehabilitation and uprating since such an increase may bring a problem that did not exist before rehabilitation.

Detailed study regarding uncertainty analysis of different flow and performance parameters of hydraulic turbine have not extensively reported in the literature. Further, transient simulations for determination of three dimensional velocity profiles with fluctuations at draft tube inlet under different operating conditions have not been reported. In optimization of draft tube geometry accurate boundary condition at draft tube inlet have not been adapted comprehensively in studies reported in literature. Very few studies have been conducted for redesigning of the existing draft tube.

The objectives of the study were formulated on the basis of literature gaps:

(i) In situ calibration of measuring instruments and uncertainty analysis

(ii) To validate the numerical results of turbine performance with results obtained from model test.

(iii) To investigate the performance of existing draft tube at different operating conditions using CFD and validate with experimental results

(iv) To obtain three dimensional velocity profiles at the best efficiency point and apply it as inlet boundary condition in process of optimization of draft tube geometry

(v) To investigate Effect of different geometrical parameters on draft tube performance using CFD simulation

(vi) Optimization of draft tube geometry for two cases using response surface methodology coupled with design of experiment approach

(vii) Financial study of optimized draft tubes

In order to achieve the objectives, research work was carried out using following methods:

1. Fabrication of experimental set up for hydraulic turbine model testing and in situ calibration of measuring instruments i.e. measuring tank load cells, calibrator tank load cell, torque transducer, friction toque load cell, differential pressure transmitter and speed sensor were performed. Calibration equations

were obtained from calibration curves along with regression error curves and used to calibrate the raw electrical signals during efficiency measurements.

2. Uncertainty analysis of flow and performance parameters i.e. discharge, torque, net head, speed and efficiency was performed as per IEC 60193 (1999) and ISO 4185 (1980) under different turbine operating conditions.
3. Model turbine test was conducted in the test rig and efficiency hill chart was obtained.
4. Three different operating conditions i.e. part load, best efficiency point (BEP) and overload were selected for numerical validation of flow and performance parameters i.e. torque, net head and efficiency.
5. Experimental set up were developed to measure the static wall pressure recovery across draft tube. Experiments were conducted under three different operating conditions.
6. Transient CFD simulation were performed with 0.5 degree time step using K-ω Shear stress transport turbulent model and performance parameters i.e. torque, net head, efficiency static pressure at inlet of draft tube were monitored at every time step. CFD results were validated with experiments under three different operating conditions.
7. Validation of wall static pressure recovery obtained from CFD results were performed with experiments under three different operating conditions.
8. Three dimensional velocity components (axial, circumferential and radial) along with standard deviation were investigated through transient CFD simulations under three different operating conditions.
9. At BEP, velocity profiles, turbulent kinetic energy and eddy dissipation rate were obtained from CFD at draft tube inlet and were exported and used as inlet boundary condition in optimization process.
10. Proposed draft tube geometry has been optimized for two different optimization goals i.e. for rehabilitation and for new draft tube design.
11. Eight different geometrical parameters were selected and using design of experiment approach, 81 design points were generated for both cases separately. RSM tool has developed different response surfaces. MOGA method coupled with RSM was adapted to optimize the proposed draft tube geometry for achieving the desired optimization goal.

12. Financial analysis of prototype draft tube for two optimized cases were carried out

A model on scaled down 1:12.52 of a prototype Francis turbine having specifications (head as 27.432 m, power as 45 MW, runner diameter as 3.988 m, discharge as 141.58 m^3/s and speed as 112.5 rpm) have been selected for the present investigations at the Hydraulic Turbine R&D Laboratory, Alternate Hydro Energy Centre, Indian Institute of Technology Roorkee, Roorkee, India.

In situ calibration of different measuring instruments used in the turbine model testing viz. flow meter, measuring tank load cells, calibrator tank load cell, shaft torque transducer, friction torque load cell and speed transducer were performed and calibration equations were derived from their calibration curves. For flow calibration, gravimetric approach using flying start and stop method was adopted. Systematic and random error of weighing balance and flow diverter has been evaluated as per ISO 4185 (1980). Performance test were conducted as per IEC-60193 (1999). Efficiency of the turbine as well as others flow and performance parameters viz. discharge, head, speed and torque were obtained at different operating points including finding out random error in efficiency measurement within 95% confidence limit. At each operating point different errors such as regression error, random error and instrument error at the nearest calibration point were calculated in order to find out total uncertainty of flow and performance parameters. Efficiency and total uncertainty in efficiency, discharge, head, speed and torque are evaluated at different operating points. It was observed that at BEP, uncertainty in efficiency measurement is found as minimum as ±0.1411%. An interesting trend was observed that at each energy coefficient ($E_{\omega d}$), uncertainty in efficiency measurement at part load points was more compared to overload point. A correlation with ±9% error is also developed to predict uncertainty in efficiency measurement in operating range of turbine.

Passage modeling approach was adopted for computational model consisting of casing, 2/10 of distributor, 3/15 of runner and draft tube. Grid indecency test has been conducted to find optimal grid points and was about 11 million for entire flow passage. Structured mesh using 'O' grid approach was made in ICEM CFD with acceptable mesh quality. Courant study has been further carried out to determine the optimum time step and was found as 0.5 degree of runner rotation. Transient simulation of five revolution of runner with 0.5 degree time step was performed using K-ω SST turbulent model in CFX solver under three different operating conditions i.e. part load, BEP and overload.

Transient simulation has been performed and flow parameters i.e. net pressure across turbine, torque and efficiency were obtained and results were validated with experimental results. At the BEP deviation in efficiency, torque and head are found as 0.19%, 2.75% and 2.3% respectively. Non dimensional velocity components i.e. radial circumferential and axial, with their standard deviation along radial direction have been obtained from transient CFD simulation at three different operating conditions. Wall static pressure recovery (C_{pw}) values have been obtained from experiment and are compared with CFD results. At part load condition deviation in wall static pressure recovery obtained from experiment are about 21.74% although at BEP, value obtained from CFD were 11.3% higher than value obtained from experiments. Swirl number in elbow section was maximum for all three operating conditions. It is also be concluded that the shape of elbow section need to be optimized in order to reduce swirl magnitude in elbow section and loss coefficient.

Proposed draft tube geometry has been optimized for two different optimization goals i.e. for rehabilitation and for new draft tube design. Eight different geometrical parameters were selected and using DOE approach, 81 design points were generated for both cases separately. Two different objective functions, pressure recovery (C_p) and loss factor (ζ) were considered in optimization process. Optimizations were performed on proposed draft tube at BEP where three dimensional velocity profiles along with turbulent kinetic energy and eddy dissipation rate were used as inlet boundary condition. RSM tool has developed different response surfaces. MOGA method coupled with RSM was adapted to optimize the proposed draft tube geometry for achieving the desired optimization goal. For case 1, optimum values of Cp and ζ value were for found as 0.8290 and 0.1158 respectively. Value of static pressure recovery was enhanced by 7.0% at BEP. For case 2, optimum value of Cp was found out as 0.851 which is 10.0% higher compared with performance of existing draft tube. Gain in efficiency of turbine ($\Delta\eta$) was also evaluated for two optimized cases i.e. case 1 and case 2 and found as 0.69% and 0.94% respectively.

Further financial analyses of modified prototype draft tube for two cases have been carried out and costs involved in both cases have been presented. Expected performance of prototype draft tube for two optimized cases were calculated and % gain in mechanical power, $(\Delta MP)_p$ for two optimised draft tube cases i.e. case 1 and case 2 were calculated as 1.14% and 1.57% respectively. Expected gain in energy production of prototype turbine with modified draft tube for two optimized cases i.e. case 1 and case 2 is calculated as 42.26 MU and 53.39 MU respectively and expected capital gain for two optimized draft tube cases were calculated as INR 160.9 Million and 221.6 Million respectively. The power plants going rehabilitation woks, it is recommended for power plant manager to focus on optimizing the shape of existing draft tube in addition to the improved turbine runner profile.