STUDY OF CAVITATION IN FRANCIS TURBINE FOR SMALL HYDRO POWER PLANTS

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

GOHIL PANKAJKUMAR PARAGBHAI

**ALTERNATE HYDRO ENERGY CENTRE
INDIAN INSTITUTE OF TECHNOLOGY ROORKEE
ROORKEE-247667 (INDIA)**

AUGUST, 2015

ABSTRACT

Hydropower is considered one of the major renewable energy sources worldwide which can play a dominant role for the development of a country. However, large hydro power plants have problems of long gestation period, and submergence of valuable forest and useful land. On the other hand, Small Hydro Power (SHP) projects are free from these problems and are most suitable to meet the power requirements of remote areas. Plant capacity wise definition of small hydro power is different in different countries. In India, hydro power plants having capacity up to 25MW are classified as SHP plants. The management of the large and small hydro power plants for achieving maximum efficiency with time is an important factor. However, the plant components like turbines show the decline performance after few years of operation as the turbine components under water get damaged due to silt erosion, cavitation, corrosion and fatigue. One of the significant reasons of turbine components failure is erosion due to cavitation.

Cavitation is a complex multiphase phenomenon that involves the formation and activity of bubbles inside a liquid medium. It causes negative implications on hydro turbines such as noise, vibration, surface erosion and performance loss. Inception and development of cavitation in hydro turbines depend upon different parameters such as atmospheric pressure, suction head, velocity of flow, temperature, surface tension, gas content in the liquid and operating hours of the turbine. Cavitation is difficult to be eliminated completely and cannot be avoided under off design conditions. However, attempts should be made to minimize it within acceptable limits.

Reaction turbines like Francis and Kaplan are more prone to cavitation as pressure of water flow varies inside the turbines. Based on literature survey, it is revealed that application of Francis turbines has the highest percentile in all continents and are more affected by cavitation in comparison to other hydro turbines. It may be owing to having fixed runner vanes arrangement of Francis turbine and it operates under a larger range of pressure head than Kaplan turbine.

Traditionally, the studies on cavitation completely rely on model testing, which is usually difficult, time consuming and costly method in case of small hydro power plants. Now a days, the rapid development in the Computational Fluid Dynamics (CFD) plays an important role in conducting inner flow field analysis in the turbo machines. Many studies reported that it is possible to use solely CFD tools to predict time evolution of cavitation erosion, including final extent and magnitude, with good accuracy. Furthermore, it allows engineers to test systems in a virtual environment.

It is also revealed from the literature that researchers have conducted many studies on cavitation to predict qualitatively the location of cavity on runner and evaluate performance drop in hydraulic machines. The quantum of erosion was calculated in terms of the total energy release on hydrofoil and size of vapor cavity. Results obtained during model testing of turbines, are generally considered to establish the studies on cavitation intensity and rate of erosion progress for different material properties. Further, the nature of the cavitation phenomenon, erosion rate on different materials and monitoring of cavitation using vibration and acoustic emissions were studied by various researchers. For installing a turbine in hydro power plants, cavitation factor known as Thoma cavitation coefficient (σ) is considered and is determined during model testing. Based on the experimentally determined results, the critical value of cavitation factor (σ_{cr}) is taken for siting of the turbine by assuming standard values of other parameters. For the given values of net head, vapor pressure and atmospheric pressure, the maximum value of suction head is determined in order to avoid the cavitation. However, at some sites the assumed values of these parameters may vary under actual site conditions. Under these situations the values of suction head may be different than the considered values which will lead the turbine under cavitation. It is therefore, there is a need to investigate the cavitation in hydro turbines under different operating conditions.

Keeping this in view, the present study was planned with the following objectives:

i. To develop 3D model of a Francis turbine prototype to stimulate the model for without cavitation and with cavitation conditions.
ii. To design and fabricate the experimental set up to compare qualitatively the numerical results with experimental results.

iii. To investigate the effect of cavitation parameters such as, suction head, water temperature and flow velocity on flow properties and output of the turbine.

iv. To develop correlations for cavitation rate and efficiency loss of Francis turbine as function of cavitation parameters.

In order to investigate the cavitation in Francis turbine, an extensive numerical study has been carried out to discuss flow properties and performance of Francis turbine under cavitation. A prototype of Francis turbine having a capacity of 200 kW has been considered to carry out Computational Fluid Dynamics analysis in order to investigate the flow under cavitation and without cavitation conditions. The specifications of the turbine considered are given in Table 1. In this work more attention was given for meshing and physical modeling. Hybrid-unstructured meshing has been generated.

Table 1: Specification of Francis turbine considered under the present study

Sr. No.	Parameter	Value
1	Power [kW]	200
2	Rated flow rate [m^3/s]	0.85
3	Rated head [m]	24.25
4	Rotational speed [rpm]	750
5	Number of runner blade	15
6	Number of guide vane	12
7	Number of stay vane	12
8	Diameter of runner [m]	0.465

The range of parameters has been considered based on the operating conditions of the Francis turbine installed at a Billing Small Hydro Power plant in Keylong District of Himachal Pradesh state in India. The range of parameters investigated is as given in Table 2. The numerical simulation has been performed for different positions of guide vanes (α) for different load conditions i.e., (i) $\alpha = 20^0$ at 100% load (v =3.35 m/s, rated load), (ii) $\alpha = 15^0$ at 80% load (v =2.64 m/s, upper part load), (iii) $\alpha = 6^0$ at 40% load (v =1.24 m/s, lower part load) and (iv) $\alpha = 36^0$ at 130% load (v =4.57 m/s, over load).

Simulations were performed with ANSYS-CFX solver and using Shear Stress Transport (SST) turbulence model. The turbulence was modeled using incompressible

URANS approach and homogeneous model was used for cavitating flow regimes. In the homogeneous model Rayleigh Plesset, mass transfer models have been used in order to consider mass transfer rate due to cavitation. The calibrated mass transfer models were applied for the numerical predictions of cavitating flow through Francis turbine and a good level of accuracy was ensured.

Table 2: Range of parameters investigated

Sr. No.	Parameters	Range
1.	Temperature, T	10^0C - 35^0C
2.	Suction head , Hs	0.1 m - 3.7 m
3.	Flow velocity, v at different load conditions	1.24 m/s - 4.57 m/s

An experimental set up was designed and fabricated for visual qualitatively validation of the simulation results. The test set up comprised with a Francis turbine model, service pump with motor, main inlet valve, high pressure upstream tank, low pressure downstream tank, air compressor, vacuum pump and instrumentations. In order to validate qualitatively the simulation results, the flow patterns have been visualized on the cone of draft tube by taking the images with the help of a camera. Under the turbine test run, the flow field images were taken for three different guide vane openings. It has been found that the similar trend of streamlines of fluid and vortex rope was obtained under experimental and numerical investigations. Locations of the cavitation region are also compared by vapor volume fraction and it was found that simulation results are similar to the results reported in earlier studies. The values of efficiency, predicted numerically under different operating conditions have also been compared with the model test results obtained from the Hill chart provided by the turbine manufacturer.

After visual qualitatively validation of flow field results, the numerical analysis on cavitation has been carried out to investigate; the effect of (i) Temperature, (ii) Suction head and (iii) Flow velocity. The parameters investigated are; velocity and pressure distribution, blade loading, pressure pulsation at draft tube, head loss coefficient, vapor volume fraction, vortex rope, cavitation rate and normalized efficiency loss.

In order to study the flow field, the velocity vectors are observed and found to be moving randomly in the form of reverse flow at higher temperature. It has been observed that with an increase of suction head for a given flow velocity, the main flow in downstream is deflected towards the wall and a reverse flow was occurred in the central region of the runner at part and over load conditions of turbine operation. Under such conditions, there is a possibility of occurrence of instability and vortex formation in the flow.

The rope shape phenomena i.e. low pressure zone has also been developed in the flow field. It is observed that the shape of the rope gets elongated with the increase in temperature and suction head for a given flow velocity. The low pressure zone area has been found to be shrunk on the suction side of the runner vanes at higher temperature. This may be due to the occurrence of reverse flow under high pressure in this zone.

Further, the blade loading i.e. pressure on the runner vane, has been found to be minimum in the vicinity of trailing edge on suction side for higher values of flow velocity. The pressure distribution on the suction and pressure side has been found to be decreased gradually with the increase in temperature and suction head. Minimum blade loading is more susceptible for vapor formation on the surface of runner blade which may lead to cavity formation.

The pressure pulsation has been investigated in time domain and frequency domain to discuss hydraulic turbine instability for different values of temperature, suction head and flow velocity. As per IEC-60193 (1999), two pressure points are considered at 0.3D and 1D from the runner exit or the inlet of the draft tube, where D is the diameter of the runner. The values of rotational frequency and blade passing frequency of the turbine are found to be as 12.5 Hz and 187.5 Hz respectively. The sampling rate of 375 Hz is used in the simulation in order to cover the entire range of frequency considered under the present study. For higher values of flow velocity, the maximum amplitude has been obtained at blade passing frequency. The effect of temperature and suction head on dominating frequency has been found negligible, however, the amplitude increases marginally with temperature.

Transient behavior has also been observed under the part load and over load conditions. Moreover, high amplitude under low frequency range has been found at part load which may cause fatigue damage to the turbine over a time of turbine operation. Further, head loss inside runner and draft tube of the turbine has been estimated. The head loss coefficient has been found to be increased in case of cavitation and it increases with temperature for all considered values of velocity and suction head. It is found that it increases monotonously inside the runner and draft tube with the increase in suction head.

In order to evaluate the shape and the extent of the cavity, the flow pattern has been observed under different cavitation parameters. A straight vortex rope has been observed at high flow velocity. The rope length has been found to be increased with the increase of temperature and suction head. The cavitation region has been found to have smaller area at higher temperature and the intensity of cavitation was increased with increase of temperature. Moreover, the cavity surface area and its intensity are found to be increased with increase of suction head, which shows that the suction head plays the main role in cavitation development. Leading edge cavitation has been observed at part load and overload conditions. The cavity surface area has been found in the vicinity of trailing edge of the runner for higher values of flow velocity at overload conditions. The cavitation region has also been found at inlet of the draft tube for all values of flow velocity considered.

Further, in order to determine the efficiency loss due to cavitation, the hydraulic efficiency of turbine is computed using the expression; $\eta_h = [T_r \times \omega / \{(P_{ti}-P_{to}) \times Q\}]$ where P_{ti} and P_{to} are the total pressure at inlet of casing and at draft tube exit respectively, ω is angular speed of runner, T_r is the torque produced by runner, and Q is the discharge. The efficiency loss of turbine has been estimated based on the simulation results under cavitation and without cavitation conditions. The efficiency loss was normalized with model test efficiency. Further, the cavitation rate has been calculated over the runner. The cavitation rate is computed by estimating the inter-phase mass transfer rate per unit volume from liquid to the vapor phase. The inter-phase mass transfer rate of the Rayleigh-Plesset model was obtained from simulation results.

It has been found that the cavitation rate increases with the increase of temperature, suction head and flow velocity. However, efficiency loss initially decreases as flow

velocity increases from 1.24 m/s to 2.64 m/s, attains a minima and then increases as flow velocity increases beyond 2.64 m/s. It increases with the increase in temperature and suction head.

Finally, based on the results obtained during numerical investigation, correlations for cavitation rate and normalized efficiency loss have been developed as function of the cavitation parameters of Francis turbine. The developed correlations are expressed as;

(i)   Correlation for cavitation rate, [kg/s.m^3],

$$\dot{m}_{cav} = 0.02149 \times T^{0.85} \times H_s^{0.43} \times v^{4.61} \tag{1}$$

(ii)   Correlation for normalized efficiency loss, [%],

$$\eta_{loss\_cav} = 1.5314 \times T^{0.3225} \times H_s^{0.1772} \times v^{-7.9274} \, e^{[4.357 \times (\ln v)^2]} \tag{2}$$

The values of cavitation rate and normalised efficiency loss predicted by correlations under different conditions are compared with those obtained by simulation and a good agreement is found.

Summarizing, based on the numerical investigation carried out under the present study, it is concluded that cavitation rate in Francis turbine increases with suction head, temperature and flow velocity. However, efficiency loss increases with flow velocity at over load and part load operating conditions of turbine. The effect of suction head on cavitation has been predicted prominent at higher values of temperature for a given value of flow velocity.

The computational results obtained under the present study may be useful to predict the pressure variation in flow field over Francis runner which is responsible of cavitation inception. The developed correlations may be useful for SHP plant developers and turbine designers in order to predict the efficiency loss of Francis turbine and possibility of damage to the runner surface due to cavitation for given site conditions.