Unsteady simulation of cavitating turbulent flow for low head Francis turbine

Pankaj P Gohil1, R P Saini2
1Mechanical Engineering Department, Sarvajanik College of Engineering and Technology, Surat.
2Alternate Hydro Energy Centre, Indian Institute of Technology Roorkee.
1pankaj.gohil@scet.ac.in

Abstract. Cavitation is undesirable phenomenon and it becomes more severe under off-design conditions and lower tail race level. Transient simulations has been performed using Shear Stress Transport (SST) turbulence model in CFX solver. The Rayleigh Plesset mass transfer model is applied for the numerical predictions of cavitating flow through Francis turbine. An attempt has been carried out to analysis of cavitating flow at low head Francis turbine under different operating conditions having varying suction heads. Summarizing, it has been found that the performance loss and cavitation rate are found maximum under over load conditions. A high amplitude of pressure at low frequency has been found at part load operating conditions of the turbine which may cause fatigue damage to the turbine over time. Cavitation intensity, performance loss and instability increase with increase of suction head.

1. Introduction
Hydropower is considered one of the major renewable energy sources worldwide which can play a dominant role for solving the problem of energy crisis. 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. Cavitation erosion is a major concern in all reaction turbines for operation and maintenance of hydro power plants. The St. Anthony Falls laboratory surveyed 729 installed hydro turbines having unit capacity more than 20 MW. The study found that more than 90% of these turbines are experiencing cavitation erosion [1].

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.
In advancement of CFD, CFX code has used Rayleigh-Plesset (R-P) equation for cavitation analysis. It is widely used for the numerical modeling of complex real cavitating flows by assuming that the liquid carries cavitation nuclei whose growth is controlled by the R-P equation. This is a simple way to account the production of vapor in a cavitating flow by pressure reduction. The corresponding source term is introduced in the continuity equation for the vapor phase which is solved numerically together with the Navier-Stokes equation for the liquid/vapor mixture.

Many of researchers have been carried out researches on cavitation in different sections of hydrofoils and model & prototype of hydro turbines using numerical and experimental methods. Cavitation numerical model i.e. R-P equation was studied and validated for the flow around a NACA 009 hydrofoil and hemispherical fore-body [2, 3]. Different aspects of cavitation phenomena over hydro turbines viz; erosion damage, cavitation acoustics and rotating cavitation [4, 5]. Moreover, Simulation of cavitating flow in prototype Francis turbine has been carried out at different operating conditions and found the cavity shape and position having good agreement with experimental study using flow visualization method [2, 6].

The paper presents that unsteady cavitating flow study has been carried out in Francis turbines having capacity of 200 kW. The simulation has been performed in operating range from over load (130% of rated load) to part load (40% of rated load) with transient condition and viscous flow turbulence Shear Stress Transport (SST) model along R-P model for cavitating flow using CFX code. In order to predict the efficiency loss, cavitation rate and instability of a prototype Francis turbine under the present investigation. It is a low head medium specific speed turbine and the specifications of the turbine as given in Table 1.

| Sr. No. | Parameter               | Value |
|---------|-------------------------|-------|
| 1       | Rated power [kW]        | 200   |
| 2       | Rated flow rate [m³/s]  | 0.850 |
| 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 [mm] | 465   |

2. Simulation methodology
Based on the detailed design drawings, the geometry of prototype Francis turbine using modeling software of Autodesk Inventor has been made. Hybrid-unstructured meshing has been generated using ICEM tool in ANSYS for all tight mesh quality. In order to minimize the influence of grid number on the computational results, a grid independency test has been carried out with three different meshes. In order to maintain mesh independent simulation results, the minimum element size and distribution are kept for each part of the model. It is used to predict the uncertainty (error) in the results. The efficiency value obtained has been found nearly same as the model test value for the medium grid number (=15.67 million of elements) among others considered grid numbers meshes. Therefore, the medium grid (=15.67 million of elements) number has been selected for entire simulation under different operating conditions. The quality mesh generated and orthogonal quality (≥ 0.4), aspect ratio (≤ 80), skewness (≥ 0.85) are found to within the permissible limits. The y⁺ value is a non-dimensional distances from the wall to the first cell node and it has been obtained in the logarithmic region range of 1.2 to 97.

All the components are attached by domain interface. The optimum time step has been considered as 0.00266667 second for 12 degree of revolution. The inlet mass flow rate and outlet pressure are specified as widely accepted boundary conditions for the simulation of hydraulic turbo machines.
The mass flow rate type boundary condition is considered at the spiral casing inlet and another identical pressure type boundary condition is considered at the draft tube outlet. The outlet of the draft tube is submerged and water level varies as per the operating conditions of hydro turbine in the power house. SST turbulence model along R-P model are considered for present investigation. The Rayleigh Plesset cavitation model validation is used as presented by various researchers [7-12].

The range of parameters has been considered based on the possible operating conditions of the turbine installed at Billing SHP plant having capacity 200 kW. The range of different parameters considered is given in Table 2.

The numerical simulation has been performed for different values of flow velocity correspond to different positions of guide vanes ($\alpha$) at different load conditions on turbine i.e., (i) $v = 3.35$ m/s ($\alpha = 20^\circ$ at 100% load i.e. rated load turbine operation), (ii) $v = 2.64$ m/s ($\alpha = 15^\circ$ at 80% load i.e. upper part load turbine operation), (iii) $v = 1.24$ m/s ($\alpha = 6^\circ$ at 40% load i.e. lower part load turbine operation) and (iv) $v = 4.57$ m/s ($\alpha = 36^\circ$ at 130% load i.e. over load turbine operation) for covering the whole range of operating conditions.

Table 2. Range of different parameters considered under the present study.

| Sr. No. | Parameters       | Range         |
|---------|------------------|---------------|
| 1       | Temperature, $T$ | $25^\circ C$  |
| 2       | Suction head, $H_s$ | 0.1 m - 3.7 m |
| 3       | Flow velocity, $v$ | 1.24 m/s - 4.57 m/s |

3. Numerical investigation and validation of flow in hydraulic turbines

An experimentation was carried out in the Hydraulic turbine testing laboratory for visual qualitative validation of the simulation results. The details of experimental facility employed for data collection with respect to above mentioned objective have been given in an earlier publication of author's [7]. It was found that similar trend of streamlines of fluid and vortex rope phenomena are visualized under experimental and numerical investigations. Straight vortex rope has been observed in both the cases of experimental and numerical analysis for full opening of the guide vane corresponding to minimum value of cavitation number as shown in Figure 1. The shape and extent of cavity region are found to be very similar in both cases of experimental and numerical investigations.

![Experimental result](image1)

(a) Experimental result  

![Numerical result](image2)

(b) Numerical result

Figure 1. Observation of vortex rope for full opening of guide vane.

Further, 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. Moreover the efficiency loss of turbine has been estimated based on the simulations results under cavitation and without cavitation conditions.
The numerically obtained results of efficiency are compared with the model test results determined from Hill chart and the same is shown in Figure 2. A good agreement between numerical analysis results and model test results is found.

![Figure 2. Comparison between prediction and model test efficiency.](image)

4. Performance analysis

The performance analysis was carried out with respect of normalized efficiency loss. The normalized efficiency loss values is computed using the expression given in earlier publication of author's [7]. Figure 3 illustrates the effect of flow velocity on efficiency loss for different values of suction head. It has been found to be as 1.68%, 0.29%, 0.63% and 2.4% correspond to flow velocity of 1.24 m/s, 2.64 m/s, 3.34 m/s and 4.57 m/s respectively corresponding to high suction head (H_s = 3.7 m).

Efficiency loss has been found marginally correspond to flow velocity of 2.64 m/s and 3.35 m/s. However, prominent efficiency loss has been found prominent for the flow velocity of 1.24 m/s and 4.57 m/s as suction head varies from 2.5 m to 3.7 m. The normalized efficiency loss has been found as 0.95%, 1.39% and 2.4% correspond to suction head of 0.1 m, 2.5 m and 3.7 m respectively. It has been predicted that the efficiency loss is maximum at high value of suction head (low cavitation number).

![Figure 3. Effect of flow velocity on efficiency loss for different values of suction head](image)

5. Cavitation rate

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. Cavitation rate has been calculated over the runner. In order to determine the effect of flow velocity on cavitation rate, plots are prepared for different values of suction head and temperature of 25°C as shown in Figure 4. The cavitation rate value is found to be as 4.5 kg/m^3.s,
63 kg/m³.s, 121 kg/m³.s and 424 kg/m³.s correspond to flow velocity of 1.24 m/s, 2.65 m/s, 3.35 m/s and 4.57 m/s respectively corresponding to value of suction head is 3.7 m.

The cavitation rate has been found as 23 kg/m³.s, 74 kg/m³.s and 121 kg/m³.s correspond to different values of suction head as 0.1 m, 2.5 m and 3.7 m respectively. The cavitation rate has been found to be more prominent correspond to suction head of 3.7 m. Initially, the cavitation rate is found to be increased linearly at small rate. However, it increases prominently at higher rate beyond flow velocity of 3.34 m/s.

![Figure 4](image-url)

**Figure 4.** Effect of flow velocity on cavitation rate for different values of suction head.

6. Pressure pulsation

Pressure pulsation study has been intended to predict the system stability aspects. The pressure points $P_1$ and $P_2$ 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 according to IEC-60193 (1999) [13].

Comparison of amplitude of pressure fluctuation with time domain and frequency spectrum of pressure at points DT₁ (=0.3D) and DT₂ (=1.0D) for different values of suction head of 0.1 m, 2.5 m and 3.7 m under with cavitation and without cavitation are presented in Figure 5 & Figure 6 respectively. The relative amplitude is found to be increased by 36.81% correspond to low flow velocity of 1.24 m/s under cavitation. Moreover, the dominating peak is observed at low frequency of 2.65 Hz (0.21 times of rotational frequency) which may be related to the Rheingan frequency. Pressure pulsation in time domain and FFT response have led to conclude that shows more stability at flow velocity of 3.35 m/s and 2.64 m/s compared to the flow velocity of 1.24 m/s and flow velocity 4.57 m/s.

![Figure 5](image-url)

(a) Pressure variation with time.  
(b) Frequency spectrum of pressure.

**Figure 5.** Pressure pulsation at point DT₁ (=0.3D) for suction head of 3.7 m at different values of flow velocity.
Figure 6. Pressure pulsation at point DT2 (=1.0D) for suction head of 3.7 m at different values of flow velocity.

7. Conclusion
On the basis of numerical investigation, the shape and extent of cavity region are found to be very similar in both cases of experimental and numerical investigations. Further, the obtained simulation results of efficiency are predicted well with model test data. It has been predicted that the efficiency loss is maximum at high value of suction head (low cavitation number). Summarizing that intensity of cavitation increases with condition of cavitation. Moreover, the magnitude of pressure pulsation is found to be more under the condition of cavitation and it is found to be increased with suction head. High amplitude under low frequency has been found at part load may cause fatigue damage to the turbine over a period of time operation of turbine.

8. References
[1] Arndt R E A, Voigt Jr R L, Sinclair J P, Roderique P and Ferreria A 1989 Cavitation erosion in hydro turbines Journal of Hydraulic Engineering 115(10) pp 1297-1315
[2] Bernad Sandor, Sebastian Muntean, Romeo F Susan-Resiga and Ioan Anton 2004 Numerical Simulation of Two-Phase Cavitating Flow in Turbomachines International Conference on Hydraulic Machinery and Hydrodynamics 4(22) pp 439–46
[3] Bernad Sandor and R Susan-Resiga 2006 Numerical Analysis of the Cavitating Flows Proceedings of the Rom Academay Ser A 7(1) pp 1–13
[4] Avellan F 2004 Introduction to cavitation in hydraulic machinery The 6th International Conference on Hydraulic Machinery and Hydrodynamics Timisoara Romania pp 11-22
[5] Escaler X, Egusquiza E, Farhat M, Avellan F and Coussirat M 2006 Detection of cavitation in hydraulic turbines Mech Syst Signal Process 20 pp 983–1007
[6] Gohil Pankaj P and Saini R P 2016 Numerical Study of Cavitation in Francis Turbine of a Small Hydro Power Plant Journal of Applied Fluid Mechanics 9(2) pp 357-365
[7] Gohil Pankaj P and Saini R P 2015 Effect of Temperature, Suction Head and Flow Velocity on Cavitation in a Francis Turbine of Small Hydro Power Plant International Journal of Energy 93 pp 613-624
[8] Bakir F, Rey R, Gerber AG, Belamri T and Hutchinson B 2004 Numerical and Experimental Investigations of the Cavitating Behaviour of an Inducer Int J Rotating Machinery 10 pp 15-25
[9] Zhang R and Hong-xun C 2013 Numerical Analysis of Cavitation within Slanted Axial-Flow Pump Journal of Hydrodynamics Ser B 25(5) pp 663–72
[10] Liu S, Zhang L, Nishi M and Wu Y 2009 Cavitating Turbulent Flow Simulation in a Francis Turbine Based on Mixture Model J Fluids Eng 131(5) Article ID: 051302 pp 1-6
[11] Jost D and Liqe A 2011 Numerical Prediction of Non-Cavitating and Cavitating Vortex Rope in a Francis Turbine Draft Tube Strojniški Vestn J Mech Eng 57 pp 445–456
[12] Liu JT, Wu Y L and Liu S H 2013 Study of unsteady cavitation flow of a pump-turbine at pump
mode IOP Conf Ser Mater Sci Eng 52 Article ID: 062021 pp 1-8

[13] IEC 60193 1999 Hydraulic turbines, storage pumps and pump-turbines-model acceptance tests
International Electrotechnical commission Geneva