Performance analysis of micro-class Francis hydro turbine by CFD

Enkhtaivan Batmunkh and Young Ho Lee
Department of Mechanical Engineering, Graduate School, Korea Maritime and Ocean University, 727- Taejong-ro, Yeongdo-Gu, Busan 49112, South Korea

lyh@kmou.ac.kr

Abstract. Hydropower is the largest source of renewable energy. Francis hydro turbines are the world’s most widely used hydro turbine. In this paper, CFD software used to simulate for the performance analysis of micro-class Francis hydro turbine. A draft tube is one of the most important parts of Francis turbine which connects the runner exit to the tailrace where the water is being finally discharged at atmospheric pressure from the reaction turbine. There are several issues in the draft tube like a pressure pulsation, effect of cavitation of draft tube performance, vortex rope study etc. At part-load condition vortex form, it hits structure constantly which affect the performance of the turbine. This study focused on prediction of vortex behavior at the draft tube and numerical results obtained the hydraulic performance of 3KW micro Francis turbine with the inlet pipe, a spiral casing with 12 guide vanes, 6 stay vanes and the runner 13 blades and a draft tube. Additionally, results from the CFD simulation and experimental results will be compared to 3KW micro-Francis turbine with misaligned guide vane.

1. Introduction

Today, Global warm has become one of the biggest issues in the world which is concerns dictate greater attention to renewable energy. Water has been playing an important role in human’s life. Antiquity people have been harnessing water to perform work for thousands of years. Throughout ancient history, French engineer Benoit Fourneyron had developed successfully water turbine in the first half of 19th century. Shortly after, British-American engineer James Francis invented the first modern turbine the Francis turbine which is inward flow reaction turbine and the most widely used water turbine in the world today. [1] Francis turbine operates water head from 10 to 650 meters. Most of the hydropower plant uses Francis turbine on their application. A draft tube is the main part of Francis turbine where is the turbine can reduce pressure to higher extent without fear of back flow but there are several issues like vortex rope when turbine operates in part-load condition. Vortex rope has been critical issues when turbine operates under part load condition, especially for the Francis turbine. The rotation of the vortex rope - it usually rotates with 20-40% of the turbine speed - causes severe rotating pressure fluctuations. [2] They are generated by unsteady vortex behavior as the incoming swirling flow decelerates in the diffuser cone, a hydrodynamic instability arises that looks like a rope swirling in the draft tube, the so-called vortex rope. [3,4] This phenomenon has affected efficiency reduction. Due to the vortex rope high-pressure unsteady fluctuations on the walls of draft tube that might be lead structural vibration and fatigue damage. Francis turbine experiences cyclic stresses, asymmetric forces on the runner, and wear and tear, all of which reduce the operating life of
the components [5]. For the vortex problem, many researchers explain about the formation of the vortex rope, vortex rope breakdown and furthermore mitigation of the vortex rope using control technique. Computational fluid dynamics (CFD) is the very helpful technique which is can predict the complex flow field in the hydraulic machines. It is used different industries for different analytical capabilities. It’s complicated to measure the vortex rope when turbine working in actual operating. So CFD is possible to measure that we can’t predict during real operating the frequency, pressure pulsation amplitude etc. The primary goal of this study to get the accurate result from numerical analysis and experimental analysis. In particular, this work aims to understand the fundamental physical processes governing the formation of the vortex rope.

2. Numerical method

2.1. Modelling

Francis turbine is designed with 3 main components; spiral casing with 12 guide vane, 6 stay vane, runner, and draft tube are considered as per original dimensions to be manufactured. A turbine was designed following condition $H = 20\text{m}$, $N = 1800\text{min}^{-1}$ and $Q = 0.02\text{m}^3/\text{s}$. Unigraphics NX 8.5 software is used for the modelling. The computational domain is extracted from the structured model as shown in figure 1. The main specification of turbine summarized in table 1.

![Figure 1. Computational domain of Francis turbine](image)

**Table 1.** Model turbine specification

| Parameter                | Value (mm) |
|--------------------------|------------|
| 1 Runner inlet diameter D | 150        |
| 2 Runner blade number Z   | 13         |
| 3 Guide vane number       | 12         |
| 4 Stay vane number        | 6          |
| 5 Guide vane opening      | 26.3       |

2.2. Grid convergence

The fluid domain discretization can be accomplished by multiple means, but the most often adopted in three-dimensional CFD are based on either tetrahedral or hexahedral volumes. A mesh that consists of
mainly tetrahedral elements is referred to as unstructured mesh while a structured mesh is comprised of hexahedral elements. Ansys ICEM used for numerical discretization of domains. In CFD simulation there was always big question how fine mesh is needed for certain level of CFD result. The mesh dependence was conducted for full domain CFD analysis the efficiency normalized by selected efficiency as shown in figure 2.

![Figure 2. Grid independence on performance of model turbine](image)

3.4 million Number of nodes and the unstructured tetrahedral mesh selected for all domain therefore nearby wall require special treatment the prism layer is generated. (figure 3)

![Figure 3. 3D unstructured grids generation by parts for the computational domain.](image)

2.3. Boundary condition
The simulation has been conducted by performing steady state Reynolds averaged Navier-Stokes 3D calculation at different operating conditions using Ansys-CFX 13 commercial solver. All CFD
problems are defined in terms of initial and boundary condition. Mass flow rate at the inlet of the casing as inlet boundary and pressure at the outlet of the draft tube as outlet condition. All solid walls have nonslip boundary condition. The rotational speed of runner 1800rpm. The entire model of the turbine is formed by combining the components with frozen rotor interface, each between casing and runner and runner and draft tube using General Grid Interface (GGI) method for mesh connection. Shear Stress Transport (SST) model is used for turbulence treatment.

The second order for simulation method based on multiphase transient flow in operating range from part load to over load and steady state results used for initial condition. Reference Pressure=0 Pa, Vapor pressure = 3169 Pa (temperature at 25˚C). For the transient analysis the time step was set as0.0001833s corresponding to a runner rotating angle 2˚ per time step. Total computational time was0.18s for 10 rotational periods of the runner.

3. Result and discussion

3.1. Hydraulic efficiency and power Characteristic

The hydraulic efficiency is calculated using the relation which can be defined as;

\[ \eta = \frac{T \times \omega}{(P_1 - P_o) \times \rho \times g} \]  

(1)

\(P_1, P_2\) - Total pressure at inlet and outlet [Pa]
\(\omega\) - Angular speed of runner [rad/s]
\(T\) - Torque produced by runner [Nm]
\(g\) - Gravitational acceleration [m/s²]
\(\rho\) - Density of water [kg/m³]

Available power is calculated using the relation which can be defined as;

\[ P = \rho \times g \times Q \times H \]  

(2)

\(g\) - Gravitational acceleration [m/s²]
\(\rho\) - Density of water [kg/m³]
\(Q\) - Flow rate [m³/s]
\(H\) - Head [m]

The hydraulic efficiency and power have been computed by using equation (1, 2). Evolution by varying discharge the best efficiency point achieved at full load condition. Table 3 and figure 4 show result of the all performance at different load condition. Best efficiency point /BEP/ indicated 3.32kW Power, 91.67% Efficiency at 0.02m³/s discharge. Power and efficiency drops at part loads due to loss in Head in guide vane, runner and draft tube.

**Table 2. Evaluation characteristics at different mass flow rate**

| Cases | Discharge | Head | Shaft power | Efficiency |
|-------|-----------|------|-------------|------------|
| 1     | 0.024     | 24.06| 5.12        | 90.74      |
| 2     | 0.022     | 21.09| 4.16        | 91.11      |
| 3     | 0.02      | 18.52| 3.32        | 91.67      |
| 4     | 0.017     | 15.55| 2.28        | 88.37      |
| 5     | 0.015     | 13.35| 1.66        | 85.02      |
| 6     | 0.012     | 10.44| 0.86        | 70.55      |
3.2. Flow feature, pressure distribution and torque distribution at full load transient condition

Pressure differences from the pressure and suction sides are higher at higher loads. For all operation conditions pressure distributions in the pressure side across the height of the runner inlet are non-uniform. High pressure zones could be observed at the stay vane and guide vane regions in this while magnitude of pressure decreased towards the inner radius. It means flow moves inward towards the runner. At the top of the runner the pressure is quite high whereas at the bottom the pressure is low.

The pressure contour revealed a gradual decrease in pressure from inlet to outlet region on both pressure and suction side. With the magnitude of pressure in pressure side being more than that on suction side. The higher the operating load, the more will be the blade loading more pressure difference between pressure side and suction side. Pressure distribution and velocity distribution in the central plane as shown in figure 5, 6.

Figure 4. Evolution characteristics at different mass flow rate.

Figure 5. Pressure field at mid height of the distributor

Figure 6. Velocity field at mid height of the distributor

Figure 7 shows the torque distribution of 2 runner blades. Periodic distribution of torque as time step increases. All blades have different phase and ultimately, sum of forces remain constant. Fluctuation caused by guide vanes is small and wake regions vanish quickly due to accelerated flow.
3.3. Part load operation of the turbine
The numerically obtained shape of draft tube vortex rope and accuracy of its prediction largely depend
on the choice of the turbulence model used for the simulation. There are several papers, which have
complimented the unsteady flow analysis numerically and experimentally. Various authors have
reported that standard k-\( \varepsilon \) model and obtained draft tube rope but RNG-k-\( \varepsilon \) and SST model showed
better result. In this study, SST model was used to predict the occurrence and nature of vortex rope
which is an ensemble averaged Navier-Stoke equations more responsible to streamline curvature and
higher strain rates than other models. A mature of vortex rope as obtained with taken time step and
turbulence model at 10 times rotation of runner. At the different instance of runner rotation is shown
following picture. The low pressure regions which represent the vortex center revealed the rotation of
the vortex causing pressure fluctuations. In the draft tube, 2 locations were chosen and recorded. Fast
Fourier Transformation (FFT) carried out to gather pressure signals at the selected point as shown in
figure 8.

The amplitude spectra of pressure pulsations at draft tube presented in figure 9.
As shown in figure 10 the distribution of pressure in the mid section of draft tube at a different instance of runner rotation.

3.4. Influence of MGV /Misaligned Guide Vane/
Pressure pulsation is the primary reason for an unstable operation of any hydro machinery. Misaligned guide vane has been keenly incorporated in pump turbine system to improve flow stability and minimize pressure pulsation. The characteristic of the dominant unsteady flow frequencies in the entire flow passage of the Francis turbine for various misaligned guide vane openings.
The guide vane angle of turbine under study is \( \alpha_1 = 26.5 \). The 3 different opening angle of 25°, 28° and 24° were chosen for analysis as shown figure 11. For this calculation, structured and unstructured meshes were selected for spiral casing runner and guide vane. The time dependent numerical analysis was conducted followed by steady state simulation with the same setting 24° of guide vane angle carried out for the transient analysis, the time step 2° of runner was taken 10 full rotations. So time step was 0.0000185s, corresponding to 1/80th of the runner rotation period. Figure 12 shows the result of calculation.

![Figure 12. Variation efficiency, discharge and head at different guide vane angle](image)

4. Conclusion
In this study, a micro-class Francis hydro turbine of 3kW output was designed and its flow dynamics was numerically analysed using CFX software. Optimization was performed on CFD that signifies reducing simulation and cost of design. Time dependent analysis figure out pull load, part load condition with misaligned guide vane. In numerical simulation, different sets of operating points were
selected to get performance characteristics of the turbine and best efficiency point indicated 91.67% efficiency at 0.02m³/s, power output 3.32kW. Performance of the turbine is improving as moving towards BEP from part load due to reduction of head loss in guide vane, draft tube and runner while in case of spiral casing head loss increases. Vortex shedding is a main operational problem in Francis turbine. Several researchers have considered to reducing the vortex shedding using different control techniques. With small size turbine, the level of vortex control techniques also showed moderate effect in swirl control. For misaligned guide vanes, more options can be tried out by misaligning more guide vanes, at higher angles. Additionally, based on the numerical analysis of a Francis turbine, the results for efficiency obtained from simulation are found to good agreement with the model results obtained from manufacturer.

References
[1] Trivedi, C., Gandhi, B. K., and Cervantes, M., 2013, “Effect of Transients on Francis Turbine Runner Life: A Review,” J. Hydraul. Res., 51(2), pp. 121–132.
[2] Skotak, A., “Draft tube swirl flow modelling”, IAHR WG "The Behaviour of Hydraulic Machinery under Steady Oscillatory Conditions", Brno, 1999.
[3] Y X Xiao, J Zhang, Y Y Luo, Z W Wang and H H Xu “Numerical analysis of misaligned guide vanes effect pressure oscillations in a prototype pump turbine” Materials Science and Engineering 52 (2013) 052026
[4] Z D Qian, B Zheng, W X Huai, and Y H Lee “Analysis of pressure oscillations in a Francis hydraulic turbine with misaligned guide vanes”
[5] Yan J, Koutnik J, Seidel U and Huebner B. Compressible simulation of rotor-stator interaction in pump-turbines. Proceedings of 25th IAHR Symposium on Hydraulic Machinery and Systems (Timisoara, Romania), IOP Conference Series: Earth and Environmental Science. IOP Publishing, 2010, 12(1) 012008