Comparative numerical analysis between two designs of Francis runner blades in sediment affected conditions

R Lama¹, S Gautam¹, S Chitrakar¹, H P Neopane¹ and B S Thapa¹
¹Department of Mechanical Engineering, Kathmandu University, Dhuslikh, Nepal
*Corresponding author (hari@ku.edu.np)

Abstract. Sediment erosion of hydraulic turbine is the major problem from the perspective of operation and maintenance in power plants of Himalayan and Andes region. The effects of sediment erosion on turbine components need to be predicted in advance during the design phase so that the best design with better sediment handling can be installed in real power plants. In this paper, comparison of performance and erosion of two different designs of Francis turbine i.e. design I and design II are carried out. Full turbine steady state numerical simulations are carried out for 8 different guide vane openings using shear stress transport (SST) k-ω turbulence model. Sediment erosion analysis is carried out using Tabakoff erosion model for both the designs, numerical hill charts are constructed using Nsd and Qsd values for all the operating conditions. Comparison of pressure distribution along pressure and suction side of GV surface between design I and a similar measurement in a previous experiment is carried out, which shows a good agreement between numerical and experimental results. Hydraulic efficiency and the sediment erosion rate density on the runner blades of design I for all operating conditions is higher than that of design II. The difference in efficiency is less than 1.85% for all operating conditions while sediment erosion rate density is much less in design II, which shows that design II is a better option for this turbine.

Keywords: Francis Turbine, Efficiency, Hill Chart, Sediment Erosion

1. Introduction
Francis turbine is a type of reaction turbine widely used in many hydropower projects due to its flexibility and efficiency. In these types of turbines, extraction of energy takes from runner blades with water at suitable reaction ratio. The extracted energy from water is transferred to the turbine shaft in form of torque and rotation. The speed of this turbine can be set constant by adjusting guide vane angle to operate these turbines in varying operating conditions thereby ensuring its wider operating range. Despite its advantages, there are several design, operational and maintenance challenges as well as difficulties due to erosion when used in sediment laden water. There are various reasons that cause such erosion in Francis turbine components. Several researchers have studied the nature, cause and effect of erosion on different components of turbines. The particle moving with highest relative velocity causes erosion on outlet of Francis runner[1]. Highest absolute velocity and acceleration; and incorrect pressure distribution between pressure side and suction side at inlet of runner results severe local erosion at runner inlet[2]. Since the draft tube is also connected with runner, the effects of erosion can be seen in this component too. Also, secondary flow from spiral casing causes erosion on stay vanes while high absolute velocity and acceleration in case of high head Francis cause guide vane erosion. Eroded Francis turbine components of Jhimruk hydropower plant of Nepal is shown in Figure 1, which demonstrates the severity of erosion on different components, especially guide vanes and runner. Prediction of such
erosion on components of turbine is a complex problem. CFD techniques is a powerful tool, which has been used to design and study the flow conditions inside hydraulic turbine and pumps over the past three decades[1][3][4][5]. Turbine design requires continuous design optimization by numerical simulations and verification of optimized turbine performance by model testing[6]. In addition, these simulated CFD results should be validated in order to ensure the correct design for the given design parameters for specific sites.

![Figure 1. Erosion on guide vanes and runner outlet at Jhimruk HPP](image)

Design of hydraulic turbine is site specific and for proposing a new design, the model needs to be verified. Model testing of turbine is one method, which is implemented by the manufacturers to investigate the performance of the turbine before fabricating the final prototype. Optimized designs of Francis runner in terms of higher efficiency and better sediment handling have been proposed by various researchers[8][9][4]. These designs will be model tested in Turbine Testing Lab, Kathmandu University in cooperation with Norwegian University of Science and Technology. However, the model test does not consider the erosion due to the impracticality of inducing sediments in the flow during laboratory conditions. Hence, this study is aimed for developing a numerical model of the turbine, which will be validated with the results from the experiment. The validated numerical model will then be used to predict and compare the erosion between different designs of the runner.

2. Performance prediction method of Francis turbine

The numerical generation of Hill chart is the ultimate goal of predicting the performance of turbine. Hill chart is obtained by keeping guide vane opening angle constant and varying the speed. The volume flow rate and torque on runner blades are obtained from CFD post and the net head ($H_n$), efficiency ($\eta_h$) and the dimensionless speed ($N_{ed}$) and discharge ($Q_{ed}$) factors are calculated by using equations (1), (2), (3) and (4) respectively as per IEC 60193 guidelines [11].

$$H_n = \frac{(P_1 - P_2)}{\rho g} + \frac{(V_1^2 - V_2^2)}{2g} + (Z_1 - Z_2)$$  \hspace{1cm} (1)

Where, $P_1$ and $P_2$ are static pressure at inlet of spiral casing and outlet of draft tube respectively. $V_1$ and $V_2$ are velocity at inlet of spiral casing and outlet of draft tube. $Z_1$ and $Z_2$ are elevation of points at inlet and outlet from reference plane.

The efficiency for the turbine is calculated by dividing the power output by the available water power.

$$\eta_h = \frac{T \cdot \omega}{\rho \cdot Q \cdot g \cdot H_n}$$  \hspace{1cm} (2)

Where, $T$ is summation of torque acting on runner blades, hub and shroud in Nm, $\omega$ is the angular velocity in rad/s, $\rho$ is the density of water in $kg/m^3$, $Q$ is the volume flow rate in $m^3/s$, $g$ is the acceleration due to gravitation and $H_n$ is net head in meter. The losses in other components of turbine are not incorporated during hydraulic efficiency calculation.
The process of finding efficiency and head is repeated with 8 different guide vane openings. As a rule of thumb, the rotational speed is varied in the range of ±20% from best efficiency point and guide vane opening angle, α at axis of guide vane is varied in the range of ±40% while conducting experiments [8]. The same approach is adopted for the creation of numerical hill chart.

\[
N_{ed} = \frac{\omega \cdot D_2}{\sqrt{g \cdot H_n}} \\
Q_{ed} = \frac{Q}{D_2^2 \cdot \sqrt{g \cdot H_n}}
\]

Where, angular velocity \( \omega = \frac{2\pi N}{60} \), in which \( N \) is the rotational speed of the turbine in revolutions per minute and \( D_2 \) is the runner outlet diameter in meter.

3. Blade angle distribution of Francis Turbine

Once the main dimensions of the runner are known, runner blades can be designed by developing the shape of the runner in axial view and radial view. Inlet and outlet blade angles are fixed and the distribution of the blade angles can be determined using two methods: 1) by specifying the blade angle distribution 2) by specifying the energy distribution. Choosing the β angle distribution and calculating the energy distribution \( U, C_u \) gives the designer full control of design outcome, which is widely used method for design of Francis runner[4]. Therefore, this method of blade angle distribution is adopted to generate the blade shapes in this study. The selection of blade angle distribution causes different energy conversion at each section of the blades form inlet to outlet. In this study, the turbines with different shapes of runner blades named as design I and design II are used for modeling and numerical analysis. The dimension of blades of design I and design II are same except the beta angle distribution. Design I for this case has linear blade angle distribution, whereas design II has nonlinear blade angle distribution. Design I and II are termed as shape 3 and 1 respectively in previous studies[4][1]. The conversion of hydraulic energy into mechanical energy takes place linearly in case of design I and most of the energy conversion takes form halfway through the runner to trailing edge in design II. Figure 2 shows the five different shapes of the relative blade angle distribution. Based on this five shapes of beta distributions, shape 3(design I) and shape 1(design II) runners are used in this study.

The turbine studied in this paper is scaled model (1:2.17) of reference turbine made for 12 MW capacity Jhimruk hydropower plant of Nepal which has three units of horizontal Francis turbines. The prototype turbine has 17 runner blades and 24 guide vanes. Hydraulic designs of the design I and design II are done by using in-house design software, “Khoj” and these designs have 17 runner blades and 20 guide vanes. The numerical analysis on design I and design II have some constraints because these two
turbines are already manufactured and available at Turbine testing lab. Table 1 shows the design parameters used for the designs.

| S.N. | Parameters                          | Symbol | Unit | Values  |
|------|------------------------------------|--------|------|---------|
| 1    | Head                               | H      | m    | 44      |
| 2    | Discharge                          | Q      | m³/s | 0.237   |
| 3    | Outlet Diameter                    | D₂     | mm   | 251.02  |
| 4    | Number of pole pairs in generator  | t      | -    | 3       |
| 5    | Reduced peripheral velocity at inlet| U₂     | -    | 0.72    |
| 6    | Outlet angle                       | β₂     | deg  | 20      |
| 7    | Acc. of flow through runner         | Acc.   | %    | 35      |
| 8    | Inlet height                       | B₁     | mm   | 42.1054 |
| 9    | Number of blades                   | Z Blades| -   | 17      |
| 10   | Rotational speed                   | N      | RPM  | 1000    |
| 11   | Speed Number                       | Ω      | -    | 0.32    |
|      | Thickness at Leading edge          | t_LE   | mm   | 4.5080  |
|      | Thickness at Trailing edge         | t_TE   | mm   | 2       |

4. Numerical Model

4.1 Geometry
The complete 3D turbine model was prepared in Creo parametric 3.0 for design I and design II turbine. The geometry includes the spiral casing, 20 stay vanes, 20 guide vanes, 17 runner blades and draft tube. The trailing edge of runners is modified in order to make the model simple. The labyrinth seals were not included in the model because of its higher computational power demands during numerical investigation. NACA 0012 guide vane profiles are used for both designs for simulations.

4.2 Meshing
Figure 3 shows the hexahedral mesh in turbine components which is prepared independently for all turbine components for design I and design II turbine by applying three-dimensional blocking technique in ANSYS ICEM CFD. The same quality of mesh, angle, edge parameters and node spacing were maintained for all guide vane openings.

Figure 3. Hexahedral mesh for the Francis Turbine; (top left): Spiral casing; (top middle): Runner; (top right): Guide vanes; (bottom): Draft tube
The mesh sensitivity study was carried out for full turbine passages including spiral casing, stay vane, guide vane, runner and draft tube. Three different sizes of mesh having number of elements: coarse (0.8M), medium (1.27 M) and fine (5.05 M) were created using grid refinement factor (r) by 2X. Numerical uncertainties due to mesh size was evaluated using Grid Convergence Index (GCI)[12]. Hydraulic efficiency of turbine was chosen as the monitored value and uncertainty in the efficiencies associated with three different meshes were observed. Table 2 shows the uncertainties and extrapolated values of the efficiency. The numerical uncertainty of the efficiency for the medium mesh was calculated to be 0.14%. In order to reduce the computational time, the medium mesh with 5.05M elements was selected for complete numerical simulations of turbine at different operating conditions.

Table 2. Discretization in error in numerical simulation

| Efficiency as monitored variable for fine, medium and coarse mesh |
|------------------|------------------|------------------|
| \( N_1, N_2 & N_3 \) | \( 1.27M, 5.05M & 15.05M \) | \( r_{21} \) |
| \( r_{32} \) | 1.4433 |
| \( \phi_1 \) | 1.5797 |
| \( \phi_2 \) | 86.87 |
| \( \phi_8 \) | 87.84 |
| \( \phi_8 \) | 87.78 |
| \( P \) | 6.5861 |
| \( \varepsilon_{r21} \) | 0.0600 |
| \( \Phi_{c21} \) | 1.1700 |
| \( GCI_{med}^{32} \) | 0.14% |

4.3 Operating range, boundary conditions and simulation setup
The entire simulations were performed in 8 different guide vane opening angles including BEP. Out of them, five operating points were at part load (PL), 2 operating points were at full load (FL) and 1 at BEP. These operating points were obtained by rotating the guide vane by 1degree at 2/3 of chord length of guide vane form the trailing edge in both PL and FL. Rotational speed of the runner was varied in the range of 750 to 1200 RPM with interval of 50. In order to obtain constant head, the total pressure corresponds to net head \( H_n \) was specified at spiral casing inlet and average static pressure of 0 Pa is prescribed at draft tube outlet. In this case the mass flow is entirely dependent on the operating conditions of the turbine. That means mass flow is different for different operating conditions as well as rotational speed at PL, BEP and FL. For numerical simulation of sediment erosion, 17,000 particle numbers of quartz having density of 2650 kg/m^3 and 100 micron diameter was used at the inlet of spiral casing. The mass flow rate of sediment was taken as 1.2 kg/s. The Tabakoff erosion model was used for the entire simulations. The computational domain for steady analysis consists of full turbine with spiral casing, stay vanes, 17 runner blades, 20 guide vanes and draft rube. 3D- Reynolds’s Average Navier Stokes was used to solve the governing equations for an incompressible and isothermal flow. Steady state numerical simulations were performed in CFD solver ANSYS-CFX-18.1. All the simulations used high resolution discretization in advection scheme and first order scheme in turbulence equations. The convergence criteria of Root Mean Square (RMS) residuals less than 1E-6 and maximum iterations of 500 were used in convergence control.

5. Results and Discussion
5.1 Comparison of pressure distribution with a previous study
Figure 4 shows the pressure distribution at BEP along the surface of guide vanes at end span. All the pressure values were normalized with higher pressure at leading edge, which is termed as \( C_P \) in the GV loading graph. The normalized pressure was plotted against the circumferential length of GV from leading edge to trailing edge. The pressure distribution curve shows that the difference in the pressure between pressure side and suction side is minimum towards the leading edge and increases as fluid flows towards the downstream. The flow from pressure side has negative pressure gradient and flow from suction side has positive pressure gradient at trailing edge of GV. The numerically obtained pattern of
pressure distribution for this turbine is compared to the pressure distribution obtained experimentally [11], which shows a good agreement between the numerical and experimental results for the pressure distribution along GV surface.

![Figure 4. Pressure distribution along GV surface](image)

5.2 Velocity distribution along runner blades from inlet to outlet
Figure 5 shows the numerical velocity distribution along runner blades for PL, BEP and FL respectively. The relative velocity increases for all operating conditions form inlet to outlet. At BEP, it is observed that the pattern of change in relative velocity form inlet to outlet is almost linear for design I while it is nonlinear for design II. The pattern of relative velocity distribution along streamwise locations is almost similar to the pattern of beta distribution in design I and design II for BEP. However, the pattern of relative velocity distribution for PL and FL does not matched with beta angle distribution for design I and design II.

![Figure 5. Comparison of velocity distribution along runner blades at different operating conditions](image)

5.3 Compassion of numerical hill chart
Figure 6 shows numerical hill chart of design I and design II Francis turbine. The numerical hill chart contains 80 operating points, 8 angular position of guide vanes and 10 values of runner angular speed for each angular position of guide vanes. The maximum hydraulic efficiency for design I is 88.16% at BEP, $N_{ed} = 1.27$ and $Q_{ed} = 0.10$. The maximum hydraulic efficiency for design II turbine is 87.42% at BEP, $N_{ed} = 1.27$ and $Q_{ed} = 0.11$. The efficiencies shown in these contours seems less than actual efficiencies because these are made from averaged values of efficiency rather than specific values. At best efficiency point, guide vane angle at axis is 16.42°. The discharge values for design I were 0.065, 0.136, and 0.206 m$^3$.s$^{-1}$ for PL, BEP and FL, respectively. The discharge values for design II were 0.065, 0.137, and 0.209 m$^3$.s$^{-1}$ for PL, BEP and FL, respectively.
Comparison of the results for design I and design II is shown in Figure 5. There is a good agreement for the general shape of the hill charts on the left half and right half of both the hill charts. On the other hand, for a constant guide vane opening the operating range for design I runner is larger (flatter) than design II runner (circular) in middle portion of the hill charts. This implies that design I has a wider range of BEP operation than for design II.

**Figure 6.** Numerical Hill Chart of turbines (D = 0.251 m, H = 44m); (a) Design I (b) Design II

5.4 Comparison of hydraulic efficiency and sediment erosion rate density

Figure 7 shows the comparison of efficiencies for design I and design II at 8 different guide vane openings. The efficiency curves for both the designs showed similar trends for all operating conditions. The efficiency difference for design I and design II is 0.83% in BEP and in the range of 0.92-1.85% and 0.37-0.55% in PL and FL respectively. The sediment measuring parameter termed as sediment erosion rate density (SERD) on runner blades of both design case are compared. The effect of sediment erosion is less in the case of design II in all operating conditions than that of design I. The design II is suitable for all operating conditions from erosion’s perspective.

**Figure 7.** (a) Efficiency comparison (b) Sediment erosion rate density comparison

5.5 Erosion pattern on runner outlet

Figure 8(a) and 8(b) show the erosion on runner outlet of design I and design II. Design II with nonlinear beta angle distribution has less erosion on runner outlets due to lower relative velocity at its outlets.
However, the efficiency has to be compromised to achieve less erosion in this case. Figure 8(c) shows the erosion on runner outlet of actual turbine of Jhimruk Hydropower plant of Nepal. The behaviour of erosion is almost similar for numerical analysis and actual case on runner of Jhimruk hydropower plant, showing good agreement between erosion pattern in CFD and actual runner.

![Figure 8. (a) Erosion at runner outlet: Design I  (b) Erosion at runner outlet: Design II  (c) Erosion at runner outlet of Jhimruk HPP corresponding to design I](image)

6. Conclusions

Numerical hill chart for two different designs of turbine were constructed. The hydraulic efficiency for both the turbines were compared in eight different guide vane openings. The maximum hydraulic efficiency for design I turbine was 88.16% at BEP with $N_{ed} = 1.27$ and $Q_{ed} = 0.10$. Whereas, the maximum hydraulic efficiency for design II turbine was 87.42% at BEP with $N_{ed} = 1.27$ and $Q_{ed} = 0.11$. The drop in efficiency for design II was found 0.83% in BEP, in the range of 0.92-1.85%, and 0.37-0.55% in PL and FL respectively compared to design I. The pressure distribution along guide vane surface was also compared with a previous experiments and found close agreement between the numerical and experimental results. The pattern of relative velocity distribution and beta angle distribution form inlet to outlet of runner were compared and found to be similar. Similarly, comparison of the effects of sediments on runner blades for both cases were carried out which shows that design II has better sediment handling capacity than design I for all operating conditions. Design II could be an alternative choice for design I if turbine operates in sediment laden water. However sacrifice in efficiency has to be made which is in the range of 0.37-1.85% in all operating range.

References

[1] Thapa B S, Thapa B, Eltvik M, Gjosater K, and Dahlhaug O G, 2012, Optimizing runner blade profile of Francis turbine to minimize sediment erosion IOP Conf. Ser. Earth Environ. Sci.15 32052
[2] Thapa B, 2004, Sediment erosion in hydraulic machinery, PhD [dissertation] Norwegian University of Science and Technology
[3] Lama R, Dahal D R, Gautam S, Acharya N, Nepane H and Thapa B S , 2018, Numerical investigation on performance and sediment erosion of Francis runner with different guide vane profiles, J. Phys. Conf. Ser.10421
[4] Gjøsæter K, 2012, Hydraulic Design of Francis Turbine Exposed to Sediment Erosion, Master Thesis Norwegian University of Science and Technology
[5] Keck H and Sick Mt, 2008, Thirty years of numerical flow simulation in hydraulic turbomachines, Acta Mech 201pp 211–229
[6] Kurosawa S, Lim S M, and Enomoto Y, 2010, Virtual model test for a Francis turbine, IOP Conf. Ser. Earth Environ. Sci 12, p 012063
[7] Kavurmaci B, Celebioglu K, Aradag S and Tascioglu Y, 2017, Model Testing of Francis-Type Hydraulic Turbines, Meas. Control 50 3 pp 70–73
[8] Nakamura K and Kurosawa S, 2011, Design Optimization of a High Specific Speed Francis
Turbine Using Multi-Objective Genetic Algorithm, Int. J. Fluid Mach. Syst. 2 2 pp 102–109

[9] Chirkov D V, Turuk P A, Lyutov A E, Cherny S G and Skorospelov V A, 2015, Coupled Multipoint Shape Optimization of Runner and Draft Tube of Hydraulic Turbines, J. Fluids Eng. 137 11 p 111302

[10] Chitrakar S, Thapa B S, Dahlhaug O G and Neopane H P, 2017, Numerical and experimental study of the leakage flow in guide vanes with different hydrofoils, Journal of Computational Design and Engineering, 4, pp 218–230

[11] International Electrotechnical Commission 60193, 1999, Hydraulic turbines storage pumps and pump turbines model acceptance tests

[12] Celik I B, Ghia U, Roache P J, Freitas C J, Coleman H and Raad P E, 2008, Procedure for Estimation and Reporting of Uncertainty Due to Discretization in CFD Applications, J. Fluids Eng 130 7 p 078001

[13] Thapa B S, Dahlhaug O G, and Thapa B, 2016, Flow field measurement in guide vane cascade of a high head Francis turbine, Int.conf.on water resources and hydropower development in asia, 2016, Vientiane

Acknowledgments
This study was performed as a part of PhD degree at Kathmandu University. The program is funded by EnergizeNepal project. Authors would like to thank IGC Technology, Aatmaram Kayastha for their help support.