Numerical investigation on the effects of leakage flow from Guide vane-clearance gaps in low specific speed Francis turbines

Saroj Gautam$^{1,2}$, Ram Lama$^1$, Sailesh Chitrakar$^1$, Hari Prasad Neopane$^1$, Biraj Singh Thapa$^1$ and Baoshan Zhu$^2$

$^1$Turbine Testing Lab, Kathmandu University
$^2$Department of Energy and Power Engineering, Tsinghua University
Address to correspondence: hari@ku.edu.np

Abstract. Clearance gap in guide vanes of Francis turbine induces the leakage vortex. This vortex flow interacts with the main flow and leads to the instability in the fluid flow pattern and eventually deteriorates the performance of the turbine. In this study, the detailed numerical examination of the unsteady flow due to leakage vortex and influence of this in the performance of turbine is carried out using time dependent numerical analysis. The numerical simulation is carried out using SST turbulence model with numerical validation. The development of leakage vortex is studied within the clearance gap region. Time dependent numerical simulation is carried out to investigate the growth and vortex propagation considering five revolutions of the runner. Results shows that the leakage vortex travelling form the guide vane clearance gaps influence the performance of the runner. On analysing the leakage vortex path, it is seen that the vortex travels from the pressure side of runner blade to the suction side of adjacent blade opposite to the runner rotation. Furthermore, this leakage vortex is carried down to the draft tube where the vortex rope seems to grow up to 50% geometric progression of draft tube cone and gradually decreases before reaching the elbow. Upon investigation of resulting torque and head during runner rotation, the periodic variation of torque and head can be seen, however with different phase. This is inferred to be the influence of leakage vortex that travels along with the main flow.

Keywords: Clearance gap, Francis Turbine, Leakage Vortex, Unsteady flow

1. Introduction
The losses in a typical hydropower plant is a cumulation of the losses due to civil, geological structures, losses in a transmission line, losses within hydro-mechanical components and in electro-mechanical components [1]. Out of these electromechanical losses arises from its constituting components. These are a set of rotating and stationery components after the penstock pipe that transfers the mechanical energy to the generators. In a typical power plant with Francis turbine, single rotating runner is the heart between the stationery components: single Spiral Casing, set of Stay Vanes, set of Guide Vanes and a Draft Tube. Each component in a Francis turbine is characterized by the unique flow behaviour in its own. Spiral casing or volute casing maintains the even distribution of kinetic and potential energy of water flowing through its circular inlet conduit towards the stay vanes radially. Flow through stay vanes then direct the flow towards the guide vanes. Through the guide vanes flow is directed towards the runner vanes converting about 50% of potential energy into kinetic
energy. Runner vanes are the main rotating components of the turbine, where the fluid leaving guide vanes strike at a certain angle [2-3]. The tangential impact force in the sets of runner blade causes them to rotate. Flow leaves the runner low velocity and low swirl having leftover energy within it. This flow is then transferred to the draft tube having gradually increasing cross section area to reduce to velocity of discharged fluid and minimize the loss of kinetic energy at the outlet [4-5]. Loss in a Francis turbine is the cumulation of individual losses through each component [6]. Losses in spiral casing is mainly due to skin friction and secondary flow. Secondary flow, wakes and incidence losses are major contributors to the losses in guide vanes and stay vanes. Besides these guide vanes are also associated with leakage flow losses that significantly affects the performance of turbine. In runner the leakage flow from the guide vanes, incidence loss at the inlet and swirl at the outlet are major contributors to the losses in it [7]. Similarly, the losses in the draft tubes are the major losses through the elbow, diverging section and vortices travelling from runner to the draft tube [6].

Several studies were carried out to investigate the flow behaviour in a Francis turbine using both experimental and numerical techniques. Trivedi et. al. [8] examined the high head Francis turbine with numerical method and validated with the experimental results at run-away operating condition. In a similar study by [9-10], the start-up and shut down condition of Francis turbine and associated pressure and velocity measurements during this condition was presented. A high-head Francis-type reversible pump turbine operating in a turbine mode was investigated incorporating the transients during the closure of wicket gates by [11]. It presented the numerical modelling technique to model the transients occurring during the closure of GV and its effect in the performance of the runner. Performance of Francis turbine is highly affected by the changing flow configuration [12]. Numerical technique has played significant role in predicting the flow behaviour at different components of Francis turbines in varying operating conditions [3].

In this study a hydropower plant in Nepal with low specific speed Francis turbine is presented. The powerplant has three Francis turbines with 4.2 MW of generating capacity each. Table 1 presents the specification of a reference case presented in this study.

| Parameters                              | Symbols | Units | Value |
|-----------------------------------------|---------|-------|-------|
| Power                                   | P       | MW    | 4.2   |
| Net Head                                | H       | m     | 201.4 |
| Flow rate                               | Q       | m³/s  | 2.35  |
| Rotational speed                        | N       | rpm   | 1000  |
| Number of runner blades                 | Z_{blades} |       | 17    |
| Number of GV                            | Z_{GV}  |       | 24    |
| Inlet diameter of runner                | D₁      | mm    | 890   |
| Outlet diameter of runner               | D₂      | mm    | 540   |

Overall, this paper discusses the flow in a single of unit of Francis turbine for Jhimruk HPP. A numerical technique is used to investigate the flow behaviour in the different components of Francis turbine. The method and results of unsteady flow field due to rotor-stator interaction and leakage flow is presented in this paper considering the full passage of fluid flow. The combined effect of rotor-stator interaction and leakage flow vortices affecting on the runner and the draft tube is studied with transient blade modelling technique of numerical analysis.
2. Reference case and numerical model

The reference fluid passage was designed using the design algorithm for high head Francis turbine [13] using available head and flow condition. Each components of Francis turbine were separately designed. The designed sets of components were separately modelled in ANSYS ICEM CFD. Figure 1 shows the developed model of reference case of Jhimruk HPP. It consists of a Spiral casing, 24 Guide vanes, 17 runner blades and a draft tube.

![Figure 1. Reference Francis turbine corresponding to Jhimruk HPP](image1)

The complete fluid passage model as shown in Figure 1 was further used for numerical analysis to investigate the flow behaviour. Each component of turbine is discretized by structured hexahedral mesh using commercial ANSYS ICEM CFD. The complete domain consists of nearly 30 million hexahedral elements. Walls of each component were treated as ‘No Slip’ walls. Thus, a dense mesh was created towards the wall. Size of each elements from wall were increased by 1.2x growth factor, in order to resolve the high gradient. Each of the Guide vanes in the numerical model consists of the leakage domain with high quality hexahedral elements refined near walls of top covers and guide vane end surface. The whole fluid model was formed by combining the components of each, using General Grid Interface (GGI) technique as shown in Figure 2.

![Figure 2. Mesh generated for numerical study](image2)
Figure 3. Computational domain for the Francis turbine. The domain includes spiral casing, stay vanes, guide vanes with leakage, runner and draft tube. Inlet of the spiral casing was extended 10x the diameter of the inlet pipe of spiral casing to achieve uniform flow towards the spiral casing. An opening type boundary condition was prescribed at the outlet of draft tube with zero pascal static pressure thus allowing recirculating flow at the draft tube outlet [8]. At the inlet, mass flow rate of 2350 kg/s normal the circular inlet of spiral casing was employed. Spiral casing walls, stay vanes, guide vanes, runner blades and draft tube walls were specified walls with no-slip condition. At the inlet, 5% turbulence intensity was used.

Shear Stress Transport (SST) model was used to in this study to predict the effect of turbulence near the wall of each component. This turbulence model is widely used and robust two-equation eddy-viscosity turbulence model that combines k-ω and k-ε turbulence model, where former is used in the inner region of boundary layer and switches to the later model in the free shear flow. This equation gives good agreement between mass transfer simulations with the experimental data [14-15]. The equations are defined as,

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho u_j k)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{k}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + \frac{\varepsilon}{\rho} - \frac{\varepsilon_k}{\rho} + F \frac{\varepsilon}{\rho} \frac{\partial k}{\partial x_j} + \frac{\varepsilon}{\rho} \frac{\partial \varepsilon}{\partial x_j} + \frac{\varepsilon_k}{\rho} \frac{\partial \varepsilon_k}{\partial x_j} - \frac{\varepsilon_{k^2}}{\rho} \frac{\partial k}{\partial x_j} - \frac{\varepsilon_{k^2}}{\rho} \frac{\partial k}{\partial x_j}
\]

Both steady and unsteady numerical calculations were performed using commercial CFD solver ANSYS CFX with a high-resolution advection scheme. The convergence criteria for steady state simulation was set to 1E-4. The converged solution of steady state numerical analysis was used in second step to carry out the unsteady numerical simulation. The total simulation time was 0.3s corresponding to 5 revolutions of the runner. The time step was chosen to each degree revolution of the runner corresponding to 1.67E-4s. The unsteady analysis was carried out in this case in order to investigate the vortices travelling into the runner and draft tube through the interface whereby a steady state simulation would average out the flow field through the interface [16]. For unsteady numerical simulation the inner loop iteration for each time step was set to 10.

3. Grid Convergence Test

Grid scaling test was carried out with three different grid densities (Coarse G3, Medium G2 and Fine G1) using the widely accepted grid convergence index (GCI) method [17]. From the coarsest mesh to the fine mesh the refinement of the mesh was done by increasing the number of elements in i, j and k direction of 3-D bounding block by 1.5x such that the grid refinement factor (r) is greater than 1.3 for successive mesh scheme. The GCI value was computed as,

\[
GCI_{fine}^{21} = \frac{1.25e_{21}}{r_{21}^2 - 1}
\]

Where, \(e\) represents error in absolute value, \(r_{21}\) represents the grid refinement factor from medium mesh to fine mesh and \(p\) represents the apparent order of numerical solution.
Table 2. Discretization error for the numerical solution

| Grid Type(G) | Grid Refinement factor (r) | Measurement Parameter (φ) | GCI (fine) |
|--------------|---------------------------|---------------------------|------------|
| Coarse(G3)   | r_1 = 1.48                | Head (h)                  | 0.958      |
|              | r_2 = 1.51                | Torque (τ)                | 0.962      |
|              |                           | Efficiency (η)            | 0.975      |
| Medium(G2)   |                           | GCI = 0.3023%             |            |
| Fine(G1)     |                           | 0.981                     |
|              |                           | 0.982                     |
|              |                           | 0.987                     |
|              |                           | GCI = 0.738%              |
|              |                           | GCI = 0.5031%             |

Table 2 shows the computed parameters with different mesh scheme. In this case net head, total torque produced by runner blade and efficiency of runner were chosen parameters for sensitivity study. Each value was normalized by the corresponding values at BEP. Lower uncertainties were obtained for all parameters from medium mesh scheme to fine mesh scheme. Fine mesh with lowest uncertainty value was chosen for further investigation.

4. Numerical results

In this study, only BEP is considered for further investigation. The investigation was carried out considering the leakage flow from guide vanes as well. Leakage flow travelling from the clearance gaps of GV affects significantly to the performance of Francis turbine [16,18]. The pressure pulsations in runner due to sets of stationary guide vanes termed as rotor-stator interaction is furthermore added due to the leakage flow travelling to the runner as suggested by [18-19]. In sediment affected hydropower projects, leakage flow leads to increment to the relative velocity near the impingement region that eventually induces severe erosion as suggested by [20].

Figure 4. Flow streamlines near the hub of both GV and RV. Flow travelling from stay vanes reaches the stationary domain of GV. Near the clearance gap region, high velocity fluid can enter the clearance gap region. At clearance gap fluid experiences, the pressure difference between two adjacent sides of GV, highest from the mid span region. The region of leakage flow from clearance can be clearly observed in Figure 4. These leakage flow mixes with primary fluid flowing between the GV channel. This leakage flow can have significant influence in the performance characteristic of the runner as it travels from the guide vane towards to the runner forming the vortices that disturbs the runner inlet flow conditions [21] and furthermore carried down to the downstream from the outlet of runner.

Figure 5 (a) shows the swirling strength of vortex pattern along with velocity swirling strength at different planes between two runner blades. The three-dimensional vortex defined by swirling strength which represents the imaginary part of complex eigenvalues of velocity gradient tensor. The positive value of discriminant of velocity gradient tensor for complex eigenvalues gives positive swirling strength, which shows the existence of swirling motion around local centres. The velocity gradient tensor in Cartesian coordinates can be decomposed as,
Where, \( \lambda_i \) represents the real eigenvalue with corresponding eigenvector \( \overrightarrow{v_i} \) and the complex conjugate pair of complex eigenvalues \( \lambda_{ci} \pm \lambda_{ci}i \) with corresponding eigenvectors \( \overrightarrow{v_{ci}} \pm \overrightarrow{v_{ci}} \).

Figure 5(a) shows the vortex pattern travelling from stationary guide vane domain towards the runner after fifth complete revolution of runner. As shown in figure, the tip leakage vortex travelling from guide vane clearances gaps are divided into primary leakage vortex and secondary leakage vortex. Primary leakage vortex is originated from the mid-span of clearance gap and travels towards the rotating runner. The secondary leakage vortex is developed in the guide vane clearance gap that eventually disappears before reaching towards the runner at design flow rate. At this instant leakage patterns mainly centralizes on the primary leakage vortex. The intensity of vortex as shown in Figure 5(a) shows increasing towards the outlet that is due to the higher relative fluid velocity from inlet to outlet of the runner. Furthermore, the vortices were found to travel from pressure side of one blade to the suction side of other blade opposite to runner rotation.

![Figure 5. Vortex flow with velocity swirling strength (a) Runner blade (b) Draft tube](image)

Figure 5(b) shows the shape of vortex rope in draft tube as influenced by the leakage vortex travelling from the runner vanes. The vortex rope downstream of runner is very highly affected by flow behaviour in the runner. Though the design condition, indicating zero swirl at the outlet of the runner was modelled for current numerical study, the high intensity vortex was found to gradually grow up to 50% of DT-cone and then reduced before reaching to the DT-elbow. The turbulence model used in the numerical study has significant role in predicting the pattern of vortex flow. Six different planes were created from the inlet of draft tube towards the elbow to observe the nature of swirl flow along the draft tube. The intensity of velocity swirling strength increases from the up to third plane which is at the 50% location of DT-cone. The vortex gradually decreases and disappears. This vortex rope as major contribution to the low-frequency pressure pulsation in the draft tube as suggested by [22]. In this study, leakage vortex travelling from the guide vane clearance gap has significant contribution to draft tube vortex.

The primary tip leakage vortex that travels from the pressure side of guide vane clearance gap reaches the inlet of the runner. Before reaching the rotating runner domain, 50% of the hydraulic energy
available for the runner is converted to the kinetic energy. When the flow travels to the mid-span of runner the energy is extracted by the turbine blades from the working fluid. Since, most of the available energy is extracted by the runner blades, ideally the tangential velocity of the fluid is zero that signifies the zero swirl at the outlet of runner. However, due to the travel of primary leakage vortex and the development of secondary vortices during runner rotation, the fluid leaves runner with both tangential and meridional velocity thus resulting vortex. The secondary vortices in the runner may be developed due the influence of primary leakage vortex travelling from the guide vanes to the runner. The deviation in inlet stagnation angle due to the leakage flow from guide vane clearance gap was suggested by [23]. Thus, the vortex rope is formed in draft tube due to the decelerating flow in draft tube along with swirling component of velocity. This vortex rope formation and breakdown causes the pressure fluctuation in the turbine and deteriorates the performance of the turbine at any operating conditions.

![Figure 6](image.png)

**Figure 6.** Pressure variation in draft tube (a) Length along Dt-Cone (b) Circular section of Dt-Inlet

Fluid flow from the inlet of draft tube is decelerating. Thus, pressure gradually increases when fluid flows from inlet to outlet. Figure 6 (a) shows the variation of pressure along the line at the centre of Dt-cone. In Figure 6(a), it represents the inlet of draft tube and 0 represents the outlet end of the Dt-cone. Furthermore, pressure in this case is normalized by the maximum pressure along the line. Thus, as fluid flows from inlet to the outlet pressure gradually increases indicating decelerating flow.

In Figure 6(b) the pressure variation in the circular section at inlet of draft tube along with the velocity vector is shown. The pressure in this case is normalized by the maximum pressure as well. This variation in the pressure indicates the rotational flow along the circular section. Along 360 degrees of circular position there is two peaks of pressure oscillation. These two peaks of highest pressure indicate the lowest velocities at two sections. The circular section was divided into 25 equal points. Thus, up to mid-span i.e. semi-circular arc, pressure gradually increases, reaches maximum and drops nearly by 25%. The pressure further increases and reaches nearly 97% of maximum pressure and reduces to the same minimum value. This variation in pressure at the circular position of inlet of draft tube shows the swirl flow flowing from the outlet of runner as shown by the vector plot at that circular position. The, motion of swirl at this position is at clockwise direction corresponding to the runner rotation.
Figure 7 shows the variation of Head and torque during the runner rotation. In this case, data from each degree of runner rotation at T=5T, indicating fifth runner rotation is plotted against the torque and head value. The fluctuations in torque and head values are due to the rotor-stator interaction between guide vanes and runner. The fluctuations may also be due to the leakage vortex travelling from the guide vane clearance gaps. In Figure 7 the head and torque values are normalized by the corresponding design value at BEP. Both values were found lesser than the actual design value which may be due to the strong influence of leakage flow vortex. It also indicates the losses that may have occurred in the fluid flow at various components of the turbine. Head value of the turbine is related to the pressure in the turbine that fluctuates in the runner due to the influence of rotor-stator interaction. Furthermore, [18-21] suggests the leakage vortex also influence the pressure pulsations. The torque fluctuations appear due to the expanding wakes that appear beyond stay vanes and expand to the runner as well as leakage vortex. The phases of torque and head fluctuations are different due to introduction of leakage in the clearance gap. This leakage tends to add the dynamic force in the runner blade thus reducing the torque value of the runner unlike the case without clearance gap.

5. Conclusion
In the present work, a detailed numerical investigation of high head Francis turbine is carried out introducing the tip leakage in the guide vanes. A CFD based unsteady flow was conducted using SST turbulence model at the design flow and head condition of the turbine using full-passage modelling technique. The numerical validation was carried out using the grid convergence test. The numerical model with fine grid was used for further investigation of each components of the turbine. The flow from the guide vanes along with the clearance gaps was investigated that has strong influence on the vortex development and flow towards the runner and draft tube.

The leakage vortex was found to be formed while the fluid enters the clearance gaps of guide vanes. The leakage vortices developed from all the clearance gap regions tend to travel to the rotating runner. Primary leakage vortex with high intensity fluid velocity enters the runner whereby, secondary vortices developed between the adjacent sides of GVs disappears before reaching the runner. This, vortex travels from inlet of the runner to the outlet. At rotating runner, vortex travels from the pressure side of one runner blade to the suction side of opposite blade, opposite to the runner rotation. This vortex gradually grows before leaving the rotating runner domain, which may be due to the reducing velocity from inlet to the outlet of the runner. From the outlet of the runner, flow constituting both tangential and meridional velocity was observed indicating swirl flow. The outlet of runner was the
region with least pressure, that increases towards the geometrical span of draft tube cone due to the decelerating flow.

Furthermore, time variation of head and torque value was studied during the rotation of the runner. The variation with different phases of torque and head fluctuations was observed indicating the influence of leakage flow that tends to add the dynamic force due strong vortex flow.

6. Acknowledgments
This study was a part of MS by Research degree at Kathmandu University on ‘Francis turbine without guide vanes for sediment affected projects’ supported by EnergizeNepal Project at Turbine Testing Lab. Authors would like to thank colleagues at Turbine Testing Lab, Kathmandu University and Department of Energy and Power Engineering, Tsinghua University for the continuous support.

7. References
[1]. Jorabloo M, Abdolahpour M, Roshan R and Sarkardeh H A techno-economical view on energy losses at hydropower dams (case study of Karun III Dam and Hydropower Plant) 2011 Computational Methods in Multiphase Flow VI 253.
[2]. Kadaj E and Bosleman R 2018 Energy Recovery Devices in Membrane Desalination Processes Renewable Energy Powered Desalination Handbook.
[3]. Trivedi C, Cervantes MJ and Dahlhaug OG 2016 Numerical Techniques Applied to Hydraulic Turbines: A Perspective Review, Applied Mechanics Review, Vol. 68 / 010802-1.
[4]. Jošt D, Škerlavaj A, Morgut M, Mežnar P and Nobile E 2015 Numerical simulation of flow in a high head Francis turbine with prediction of efficiency, rotor stator interaction and vortex structures in the draft tube Journal of Physics: Conference Series 579 012006.
[5]. KC A, Lee YH and Thapa B 2016 CFD study on prediction of vortex shedding in draft tube of Francis turbine and vortex control techniques, Renewable Energy 86 1406-1421.
[6]. Iliev I, Trivedi C and Dahlhaug OG 2018 Simplified hydrodynamic analysis on the general shape of the hill charts of Francis turbines using shroud-streamline modelling IOP Conf. Series. Journal of Physics: Conf. Series 1042 012003.
[7]. Chitrakar S, Neopane HP, Dahlhaug OG 2016 Study of the Simultaneous Effects of Secondary Flow and Sediment Erosion in Francis Turbine Renewable energy 97 881-891.
[8]. Trivedi C, Cervantes MJ and Gandhi BK 2016 Investigation of a High Head Francis Turbines at Runaway Operating Conditions energies 9(149)
[9]. Goyal R, Gandhi BK and Cervantes MJ 2019 Experimental investigation on a High head Francis turbine model during shutdown operation IOP Conf. Series: Earth and Environmental Science 240 022028.
[10]. Goyal R, Cervantes M and Gandhi BK 2019 Synchronized PIV and pressure measurements on a model Francis turbine during start-up Journal of Hydraulic Research 58(2) 1-17.
[11]. Wang W, Pavesi G, Pei J and Yuan S 2020 Transient simulation on closure of wicket gates in a high-head Francis-type reversible turbine operating in pump mode Renewable Energy 145 1817-1830.
[12]. Trivedi C and Dahlhaug OG 2017 Investigation of the unsteady pressure pulsations in the prototype Francis turbines during load variation and startup Journal of Renewable and Sustainable Energy doi:10.1063/1.4994884.
[13]. Brekke H 2001 HYDRAULIC TURBINES Design, Erection and Operation NTNU, Trondheim, Norway.
[14]. Chitrakar S, Thapa BS, Dahlhaug OG and Neopane HP 2016 Numerical investigation of the flow phenomena around a low specific speed Francis turbine’s guide vane cascade IOP Conf. Series: Earth and Environmental Science 49 062016.
[15]. Chitrakar S, Thapa BS, Dahlhaug OG, and Neopane HP 2017 Numerical and experimental study of the leakage flow in guide vanes with different hydrofoils *Journal of Computational Design and Engineering* 4(3) 218-230.

[16]. Gautam S, Neopane HP, Acharya N, Thapa BS, Chitrakar S and Zhu B 2020 Sediment erosion in low specific speed Francis turbine: A case study on Effects and Causes *Wear* 203152 442-443.

[17]. 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 *Journal of Fluids Engineering* 130(7).

[18]. Gautam S, Neopane HP, Thapa BS, Chitrakar S and Zhu B 2020 Numerical investigation of the effects of leakage flow from guide vanes of Francis turbines using alternative clearance gap method *JAFM* 13(5) 1407-1419.

[19]. Gautam S, Lama R, Chitrakar S, Neopane HP, Thapa BS and Zhu B 2019 Methodology to Predict Effects of Leakage Flow from Guide Vanes of Francis Turbine *IOP Conf. Series: Journal of Physics: Conf. Series*, No. 1266:12003.

[20]. Thapa BS, Dahlhaug OG and Thapa B 2017 Effects of sediment erosion in guide vanes of Francis turbine *Wear* 390-391 15 104-112.

[21]. Thapa BS, Dahlhaug OG and Thapa B 2017 Sediment erosion induced leakage flow from guide vane clearance gap in a low specific speed Francis turbine *Renewable Energy* 107 253-261.

[22]. KC A, Thapa B and Lee YH 2014 Transient numerical analysis of rotor-stator interaction in a Francis turbine *Renewable Energy* 65 227-235.

[23]. Chitrakar S, Dalhaug OG and Neopane HP 2018 Numerical Investigation of the effect of leakage flow through erosion-induced clearance gaps of guide vanes on the performance of Francis turbines *Engineering Applications of Computational Fluid Mechanics* 12 662-678.