Numerical investigation of the flow phenomena around a low specific speed Francis turbine's guide vane cascade

Sailesh Chitrakar¹, Biraj Singh Thapa¹, Ole Gunnar Dahlhaug², Hari Prasad Neopane³

¹PhD Candidate, Waterpower Laboratory, Department of Energy and Process Engineering, Norwegian University of Science and Technology, Trondheim, Norway
²Professor, Waterpower Laboratory, Department of Energy and Process Engineering, Norwegian University of Science and Technology, Trondheim, Norway
³Associate Professor, Department of Mechanical Engineering, Kathmandu University, Dhusi, Nepal

E-mail: sailesh.chitrakar@ntnu.no, biraj.s.thapa@ntnu.no, ole.g.dahlhaug@ntnu.no, hari@ku.edu.np

Abstract. Guide vanes of Francis turbines convey a significant influence on the flow field at the inlet of the runner. This influence is in the form of pressure pulsation, caused due to rotor-stator-interaction. A guide vane cascade containing a single blade passage was developed to predict the flow field experimentally. Firstly, this paper investigates flow phenomena around the guide vane cascade through computational techniques. A numerical model is prepared with three different turbulence models. The velocity distribution obtained from these models are compared with experimental results at two circumferential midspan locations. Secondly, the influence of increasing the clearance gap on the flow is studied. Such gaps are expected to increase when the flow containing eroding particles passes through the turbine. This paper also shows that the pressure difference between the pressure and the suction side of guide vane influences the leakage flow through the gap. Hence, reduction of the pressure gradient will reduce leakages through clearance gaps, hereby condensing the subsequent effect of pressure pulsations and erosion. This study also shows that the effect of the gap is prominent in the near wall regions which are close to the gap, whereas it dissipates gradually towards the midspan.

1. Introduction
Guide vanes (GV) of Francis turbines experience several unsteady flow phenomena at high velocity and acceleration. Such flow contributes for total losses developed in turbines and moreover, aggravates erosion, if solid particles are present in the flow. Brekke[1] described the wakes from guide vanes in Francis turbine as ‘pressure shock’ waves, that cause high pressure pulsations in runner. These pulsations eventually adds up to the unnecessary vibrations, noise and fatigue problems in the turbine. The problems of pressure pulsation is more severe when the plant is running at part load conditions. An experimental investigation of a radial pump-turbine showed that the flow pattern at the space between the guide vanes and the impeller deteriorated at low discharge due to vortices and backflows[2]. These days, when Francis turbines...
are needed to operate frequently at off-designed conditions, the fatigue problems have become more vulnerable.

Literatures show that there has been a number of studies performed to predict the flow field around GV and the vaneless space between GV and runner, through both computations and experiments. Finstad[3] used TRPIV (Transient Particle Image Velocimetry) technique to study the dynamics present in wake flow from a hydrofoil at 9 m/s. Su et. al.[4] used PIV of a complete Francis model to study the Blade Passing Frequency (BPF) phenomena. This experiment used transparent GV and Stay Vanes (SV), and the flow was observed by drilling a hole on the casing. Antonsen[5] used Laser Doppler Anemometry (LDA) technique in a guide vane cascade rig to relate the pressure distribution at the inlet of the runner and the dynamic load due to Rotor Stator Interaction (RSI).

All of these experiments have shown that prediction of the unsteady flow around GV is complex and is associated with several assumptions and uncertainties. The numerical method, on the other hand, is gaining popularity, specially with the available commercial tools. However, such technique also faces questions related to the fidelity of results. Hence, a proper validation technique has to be imposed to strengthen the precision of computations.

1.1. Clearance gap and leakage flow

A small clearance gap is present in guide vane to adjust the guide vane angle, depending upon the load or flow variation. This adjustment maintains the constant speed of turbine at different operating points. However, the size of the gap increases due to head cover deflections and erosion on the surface, leading to inevitable losses due to leakage.

The concept of leakage flow through clearance gaps is shown in Figure 1. The pressure difference between the two sides of the guide vane drives the flow towards the low pressure side. This flow creates disturbances to the main flow in the suction side, which accumulates and travels further downstream into runner, creating consequent effects.

Eide[8] claims that out of 5-6% of the total losses developed in a high head Francis runner, around 1.5% is due to the leakage flow from guide vanes. He built a 2-D numerical model of guide vanes including clearance gap to show leakage flowing through the gap.

A single guide vane (GV) cascade test rig has been developed in the Waterpower Laboratory at NTNU, to predict velocity and pressure distribution till the inlet of runner. The measurements acquired from this rig help to study the dynamic behavior of the flow and validate the computational results conducted in the relevant field. The main objective of this paper is to develop an appropriate steady state numerical model of the rig, such that its results are comparable with experiments. A reference case, in which i) GV profile is symmetrical ii) clearance gap is excluded, and iii) measurements are taken in midspan location, are first computed taking into account the numerical sensitivity of results. These values are then validated with experimental results. Another objective of this paper is to use the validated numerical model to study the leakage flow when clearance gaps are present.

2. Test rig geometry

The test rig is a guide vane cascade of a low specific speed Francis turbine, which is able to produce the velocity distribution downstream, similar to that of the prototype turbine. The guide vane used in this rig is a prototype from Jhimruk Hydroelectric Centre (JHC) of Nepal with a net head of 201.5m. The power plant uses three units ($2.35 \text{ m}^3/\text{s}$ flow per unit) of low
speed number Francis turbines (0.32)\(^1\) with capacity 3x4.2 MW. These turbines have faced a serious problem of sediment erosion and over the last decade, a number of investigations have been performed to minimize the effect. The test rig was built with a purpose of understanding the flow field, including velocity and pressure distributions till the runner inlet. Secondly, the rig is also aimed to provide an insight of how erosion on the guide vane affects the flow. The test section of this rig, as shown in Figure 2, contains a single guide vane passage assembled inside a plexi-glass, such that the velocity field at different spans can be measured by Particle Image Velocimetry (PIV) techniques. The bottom cover of this test section contains an opening to let the laser sheet pass inside the passage. The rig is also equipped with pressure measurements along the mid span surface and face of the guide vane. Inlet of the rig contains a pipe of diameter 400 mm, which is connected to a pressure tank purposed to provide 10 bars maximum pressure to the rig. Outlet of the rig is connected to a flowmeter and back to the pressure tank.

3. Numerical model
The commercial CFD solver ANSYS-CFX-15.0 was used for numerical simulations in this study. Reynolds average method is used to solve the governing equations for an incompressible and isothermal flow. The first part of the work contains numerical sensitivity study based on convergence of results and experimental outputs. In this study, a steady analysis was carried out on the single guide vane cascade flow domain. All simulations used high resolution discretization in advection scheme and first order upwind scheme in turbulence equations. The solutions were considered to be converged when Root Mean Square (RMS) residuals for each equation solved was less than 10\(^{-5}\).

3.1. Boundary conditions and locations of measurement
A steady analysis was performed in the test rig with inlet, outlet and wall boundaries. Mass flow rate at the inlet of the test rig corresponds to two guide vane passages, i.e. 195.83 kg/s. A turbulence intensity of 5% in the inlet was used in this study. A static pressure of 9 atm was specified at the outlet to overcome cavitation at the GV outlet. Remaining boundaries including the blade are defined as no-slip smooth walls. Velocity measurements were carried out at three circumferential locations along the streamwise positions: 1) Stay Vane (SV) outlet, 2) Guide vane (GV) outlet and 3) Runner inlet. Table 1 shows the analytical values at these locations, which were used for the design of the turbine. The boundaries in the flow domain and the positions are clarified in Figure 2. The curves at each location is divided into 100 equidistant points where CFD results are investigated.

| Location       | Diameter (m) | \(C_u\) (m/s) | \(C_m\) (m/s) |
|----------------|--------------|---------------|---------------|
| SV Outlet      | 1.06         | 13.87         | 7.76          |
| GV Outlet      | 0.93         | 38.58         | 8.72          |
| Runner Inlet   | 0.89         | 40.51         | 9.26          |

3.2. Mesh independence study
In this study, a hexahedral structured mesh was used to discretize the domain by using ICEM. O-grid technique was used around the guide vane and the cylindrical regions. In the

\[
\omega = \frac{\omega}{\sqrt{2 g \cdot H}}
\]

\[
Q = \frac{Q}{\sqrt{2 g \cdot H}}
\]

\(^1\) Speed number \(\Omega = \frac{\omega \sqrt{Q}}{2 g \cdot H}\)
near-wall regions of the domain, mesh density was refined to resolve high velocity gradients. Inside the clearance gap, 25 elements per 1mm thickness was used with finer distribution near wall boundaries. The variation of velocity distribution at stay vane outlet, guide vane outlet and runner inlet were investigated for mesh sizes ranging from 0.3 million to 2 million node count, keeping the distribution constant. It was seen that the differences in the velocity components for all mesh sizes remained less than 1% in all locations. Therefore, the final mesh was chosen using the same mesh distribution. The area around the guide vane was refined to maintain the $y+$ value around 1. Table 2 shows properties of the final mesh selected for the further analysis in the study. The $y+$ values correspond to the operation at design condition. Figure 3 shows the mesh generated at different locations. It shows that the regions with high velocity gradients and potential turbulent flows are resolved with a fine mesh density.

### Table 2. Properties of the selected mesh

| No. of Nodes | Min. face angle [deg.] | $y+$ (min/avg/max) |
|--------------|------------------------|--------------------|
| 13120818     | 7.82°                  | 0.35/11.58/17.86   |

#### 3.3. Turbulence models

In this paper, turbulence models based on Reynolds Averaged Navier-Stokes (RANS) equations were employed due to its practicability and computational efficiency compared to Direct Numerical Simulation (DNS). This study compares results of 3 turbulence models: 1) SST 2) BSL and 3) Omega Reynolds Stress. Out of these models, SST and BSL are the 'Two-equation' type eddy viscosity turbulence models. The eddy viscosity turbulence model suggests that turbulence consists of small eddies that forms and dissipates continuously with Reynolds stresses proportional to mean velocity gradients. Two equation turbulence models consists of velocity and length scale, which are solved using separate transport equations. Omega Reynolds Stress model is a type of Reynolds Stress turbulence models which solve an equation for the transport of Reynolds stresses in the fluid instead of using the eddy viscosity hypothesis.

In this paper, results from the PIV experiment in a low flow condition (58.2 litres/s) have been presented. The CFD analyses for different turbulence models obtained were compared with the experimental results. To maintain equivalent boundary conditions for comparison, the simulations were carried out for low flow condition.
4. Experimental setup
The results obtained from the computational simulation were compared with the measurements done in the actual rig developed at Waterpower Laboratory in NTNU. The rig allows velocity measurement through PIV technique offered by Dantec System. The light plane was generated by two Double cavity Nd-YAG lasers providing 120 mJ by pulse. This plane was visualized as paired images by a HiSense 2M CCD PIV camera at 150µs. Fluorescent seeding particles with a density of 1.016 kg/m³ and mean diameter of 55µm were used. Other properties of camera, seed particles, post processor and the design of the test rig are described in [9] and [10]. PIV experiment was carried out on the existing 1 GV cascade rig. As mentioned in Section 3.3, the experiment was carried out in a low flow condition (58.2 litres/s) and corresponding CFD was performed using different turbulence models. The results are presented in Section 5.

5. Results and Discussions
Results from this study are presented in two parts. The first part contains the results of the low flow condition, where tangential velocities ($C_u$) at guide vane outlet and runner inlet obtained from PIV are compared with CFD for all turbulence models. The second part contains the results of the designed flow condition, which is focused on the study of leakage flow through clearance gaps and its effect on the flow.

5.1. Comparison between CFD and PIV at low flow condition
The PIV result compared with CFD is shown in Figure 4. This figure represents tangential velocity profiles at midspan, when the clearance gap was 2 mm. The X-axes of these graphs represent circumferential positions at GV outlet and runner inlet, as shown in Figure 2, starting from top edge (towards pressure side) to the bottom edge (towards suction side) of the guide vane. The velocities towards the top edge is higher than towards the bottom edge in the current rig because the circumferential location of the top edge lies in the region where the area of passage is smaller. In a real scenario, a periodic curve is expected throughout the guide vane ring. Both CFD and PIV presents this limitation in the current rig. At GV outlet, the CFD results for all turbulence models are close to the experimental values. Higher velocity values in the simulated results imply the effects of wall roughness, which were not accounted in CFD. The velocity deficit downstream of the guide vane’s trailing edge has been captured by both CFD and PIV. However, the deficit predicted by CFD is larger than the PIV result. Towards suction side, the velocity captured by PIV is much lower and unsteady compared to CFD. This implies...
that in real condition, the flow from pressure side is seen to disturb the suction side flow, even at the midspan regions. More explanations about the leakage flow is provided in Section 5.2. Figure 4 also shows that deviations in the velocity at GV outlet for different turbulence models are marginal.

At runner inlet, the deviation between the CFD and PIV result is much larger than GV outlet. Whereas CFD captures the velocity deficit due to the wake slightly in this position, PIV shows that the effect is negligible. The suction side regions at runner inlet have the similar variation as that of GV outlet. However, the pressure side shows that the variation
is much larger. This discrepancy can be explained from Figure 2, which shows the locations of measurement in the existing rig. The location of measurement towards suction side is very close to the guide vane, but towards pressure side, the flow covers much of the boundary walls before reaching the location. In CFD, the roughness of the wall has not been accounted, which limits the prediction of velocity drop near wall. During experiment, the effect of the rough wall is prominent. In the real scenario, except the neighboring GV walls, the walls do not exist in the gap between GV and runner. Within this gap, the flow should ideally follow free vortex theory, i.e. $C_{u,r} = \text{constant}$. Apparently, the application of this theory has been limited due to the presence of cover plates in the current rig. Figure 4 for runner inlet also shows that there are some discrepancies between the results of 3 turbulence models. The major differences are observed in the position and strength of the wake. Omega RS model shows that the wake from the GV trailing edge is dissipated almost completely as it reaches the runner. Similar results are observed from PIV. BSL model shows that the wake is shifted towards the suction side due to the pressure difference between the two sides without complete dissipation of the wake. SST model on the other hand shows that position of the wake in the circumferential position shifts marginally with maximum velocity deficit. It was found from literature that both SST and BSL models were introduced in a study by Menter[6]. BSL model was introduced as a first step, where $k-\omega$ model was activated in the near wall region and $k-\epsilon$ model was activated in the outer wake and free shear region. SST was then introduced as a second step or addition to the BSL model, where the definition of the eddy viscosity was modified to account for the transport of the principal turbulent shear stress. The study by Menter recommends SST for aerodynamic applications with flows involving adverse pressure gradients and separations. In addition to this study, a technical memorandum of NASA[7] concluded SST model to be the best model among two-equation turbulence models for such flows. For two equation turbulence models, this study also recommends implicit second-order numerical difference methods for diffusion terms of the momentum, energy and turbulence model equations. Based on these recommendations and results obtained after the study of numerical sensitivity, SST turbulence model was chosen for designed flow condition and study of the clearance flow.

5.2 Leakage flow

Considering symmetrical flow along the span (from hub to shroud) of the guide vane, clearance gap was added on one end. Similar arrangement has also been made for experimental studies. However, results from the experiment for leakage flow are not included in this paper. Moreover, this simulation was carried out for designed condition using SST turbulence model and second-order differential schemes. The aim of this section is to study the physics of the leakage flow and how the pressure distribution around the guide vane directly influences the flow. An extreme clearance gap of 4 mm is added to one end of the same numerical model. The gap is much less for the turbines operating at normal conditions but increases gradually as the surfaces are eroded by the sediment particles present in the flow.

Figure 6 shows pressure distribution around the guide vane at midspan and Figure 5 shows flow inside the clearance gap affecting the ideal flow. It also shows the change in the direction of the flow vector as it travels along the chord. The pressure distribution curve shows that the difference in the pressure between

![Figure 5. Leakage flow through the clearance gap and flow vectors along the chord line](image)
pressure and suction side is minimum towards the leading edge, whereas increases, as it flows more and more downstream.

This difference is expected to increase at part load conditions due to change in guide vane angle and eventual stagnation point. The pressure distribution has a direct influence on the behavior of the leakage flow too. These figures show that the flow is normal in the leading edge region of the clearance gap because of a small pressure difference. High pressure imposed from the pressure side near the axis region of the blade drives the flow towards the suction side. The intensity of this flow increases more as it moves towards trailing edge, which creates swirl flow at the suction side. When such flow travels into the runner, a pressure oscillation is initiated. Moreover, such flow is susceptible to efficiency drop and erosion of the turbine components. Figure 7 shows leakage flow at two span positions away from the clearance gap region. The disturbances due to leakage flow are predominantly seen near the gap, whereas it gradually diminishes when the observation plane is shifted away from it. This effect is explained more in Section 5.3.

5.3. Comparison of flow between various size of the gap
The effect of clearance gap was further studied by comparing the results between 1mm to 4mm size of the gap at an interval of 1mm. Results are compared based on the tangential velocity components at outlet of GV and inlet of runner for different span locations. Figure 8 shows how the flow is distorted when the clearance gaps are present. The observations were made for four planes along the span of the guide vane, which are presented in z-axis (bottom to top) in the figure. The distortion is predominantly seen in the region close to the gap. At 50% and 25% span, the flow is comparatively less distorted than 12.5% and 5% span locations. It can also be noticed that the flow in the suction side, which is towards left of the wake in figures, is affected more than in the pressure side. This difference is observed for all the planes. This observation infers that the flow from the pressure side enters the clearance gap at a velocity high enough to distort the ideal flow in the suction side, as discussed in Section 5.2. In the figure, this visualization is most distinct at 5% span of the guide vane outlet. While the effect worsens for higher clearance gaps towards the suction side, the pressure side remains marginally affected. A distinct velocity drop observed in the middle of curves indicates the wake effect from guide vane’s trailing edge. At GV outlet, the position of this drop remains same for all clearance...
gaps, although the magnitude of the drop varies slightly. At downstream locations, flow from the suction side is seen to perturb and mix with wake profiles shifting the drop in both sides depending upon the span location.

6. Summary and future works
This paper presented the development of a numerical model to predict the flow phenomena around guide vanes of low specific speed Francis turbines. The flow on one guide vane cascade rig was simulated computationally, which was validated with experiment carried out in Waterpower Laboratory of NTNU. The simulation also included leakage flows, which develops when clearance gap is added in the guide vane. In this study, clearance gaps of 1-4mm were added on one side of the guide vane. The results showed that the flow on the suction side are affected significantly due to the pressure-to-suction side flow through the gap. The span position near the gap was affected more than the midspan regions.

This study showed that the leakage flow through the clearance gap is related to the pressure difference between the pressure and suction side of the guide vane. During the part load conditions, i.e. when the guide vane is in the closing position, the pressure difference is expected to increase, which results in more leakages. While using symmetrical guide vane profiles is a conventional technique in Francis turbines, this study shows the possibility of changing the guide vane profile to reduce the pressure difference, and consequently, leakage flow through the clearance gap. The minimization of leakage flows in every operating conditions can eventually minimize erosion and pressure pulsation problems in the turbine.

References
[1] Brekke H 2010, A Review on Oscillatory Problems in Francis Turbines, New Trends in Technologies: Devices, Computer, Communication and Industrial Systems, Sciyo, 217-232
[2] Hasmatuchi V, Roth S, Botero F, Avellan F and Farhat M 2010, High Speed Flow Visualization in a Pump-Turbine under Off-Design Operating Conditions, 25th IAHR Symposium on Hydraulic Machinery and Systems

Figure 8. Effect of clearance gaps at 4 span positions
[3] Finstad P H E 2012, Secondary Flow Fields in Francis Turbines, PhD Thesis, Norwegian University of Science and Technology, Trondheim

[4] Su W T, Li X B, Li F C, Wei X Z, Han W F and Liu S H 2014, Experimental Investigation on the Charateristics of Hydrodynamic Stabilities in Francis Hydroturbine Models, Advances in Mechanical Engineering

[5] Antonsen O 2007, Unsteady Flow in Wicket Gate and Runner with focus on Static and Dynamic Load on Runner, PhD Thesis, Norwegian University of Science and Technology, Trondheim

[6] Menter F R 1994, Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications, AIAA Journal, Vol. 32, No. 8, 1598-1605

[7] Bardina J E, Huang P G, Coakley T J 1997, Turbulence Modeling Validation, Testing and Development, NASA Technical Memorandum 110446

[8] Eide S 2004, Numerical analysis of the head covers deflection and the leakage flow in the guide vanes of high head Francis turbines, PhD Thesis, Norwegian University of Science and Technology

[9] Thapa B S, Trivedi C, Dahlhaug O G 2015, Design and development of guide vane cascade for a low speed number Francis turbine, Journal of Hydrodynamics, Ser. B, Accepted on November 22

[10] Thapa B S, Dahlhaug O G and Thapa B 2016, "Velocity and pressure measurements in guide vane clearance gap of a low specific speed Francis turbine," in 28th IAHR Symposium on Hydraulic Machinery and Systems, Grenoble, France, 4-8 July