Numerical investigation of flow field and performance of the Francis turbine of Bhilangana-III hydropower plant

R Thapa,* S Sharma, K M Singh and B K Gandhi

Department of Mechanical and Industrial Engineering, Indian Institute of Technology Roorkee, Roorkee 247667, India
* Corresponding author (rthapa@me.iitr.ac.in)

Abstract. Computational fluid dynamics is an alternate tool to predict the performance as well as to investigate the flow field of a hydraulic turbine at different operating conditions. The objective of this paper is to analyze the flow field numerically and to predict the performance of Francis turbine of Bhilangana–III power plant. A numerical model of the prototype Francis turbine is undertaken to perform the simulation under the actual working condition of hydropower. The steady-state simulation has been performed at different operating conditions such as high load, part load, and best efficiency point using two turbulence model $k-\omega$ shear stress transport (SST) and standard $k-\varepsilon$ with zero clearance gap and with a gap due to erosion. Based on computational results, an attempt has been made to compare the performance of the turbine under erosive wear conditions. The computational results are very close to the expected performance characteristics of the prototype turbine under no-gap condition. A difference of 0.24% and 1.71% in efficiency is observed between the numerical and prototype hill chart curve at the best efficiency point (BEP) and full load conditions, respectively. The simulation has also been performed with a clearance gap of 1.85 mm to study the effect of the gap, due to erosion, on the efficiency. The leakage flow-through 1.85 mm clearance gap has resulted in a decrease in the efficiency of the turbine. The efficiency drops around 5.73 % at BEP condition and 5.83% at full load condition. These computational results may be useful for further study of the turbine of Bhilangana –III power plant.

Keywords: CFD, Francis turbine, performance, efficiency, clearance gap, leakage flow

1. Introduction

Power generation from hydropower plants reached to around 4,200 terawatt-hours (TWh). The worldwide total installed capacity was 1,292 gigawatts (GW) in 2018 [1]. Francis turbine works efficiently under a wide range of operating conditions. Hence, it is the most preferred hydraulic turbine in both small and large-scale hydropower plants. It contributes to 60% of the energy of global hydropower capacity [2]. The design of Francis turbine was based on the Best efficiency point (BEP) at given head and discharge condition rather than considering other significant problems in turbines. CFD tools provide greater flexibility to researchers for investigating and predicting the performance of Francis turbine at part load, full load, and BEP conditions. Using CFD aided design, the flow analysis
of rotor-stator interaction, risk of cavitation, sediment erosion problem, performance characteristics of the turbine, etc., can be analyzed over a wide range of operations during pre and post-construction [3]. The steady-state simulation of hydraulic turbines using CFD tools can provide hill chart and hydraulic losses in its components with a description of flow inside the turbine [4]. The operation of the turbine at off-design conditions such as load acceptance, load rejection, start-up, shutdown, etc. affect the dynamic stability of the turbine [5]. The real flow in a hydro turbine runner is unsteady, turbulent, and highly three dimensional with strong effects from the rotation and blade curvature, which make the flow extremely complicated [6]. The flow investigation and performance evaluation of the turbine at different operating conditions in the field is time-consuming as well as economically challenging. Hence computational method has been adopted for predicting the flow behavior inside the turbine with features of selecting a suitable mathematical flow model. Prasad et al. [7] have performed steady-state 3D flow analysis in an experimentally tested axial flow turbine at three operating conditions using k-ω turbulence model and observed that the computed results were closed to the experimental values. Trivedi et al. [2] performed an unsteady numerical simulation of Francis turbine taking five operating conditions using the k-ω shear stress transport (SST) and standard k-ε turbulence model. The computed results closely matched with experimental values at BEP point and differed widely at part load conditions. A relatively small difference (~1%) between the numerical and experimental results was obtained by using the standard k-ε turbulence model [2,8]. The k- ω SST turbulence model provides a more accurate flow simulation while simulating cavitation flow in turbomachines [9]. Nodvik et al. [4] compared the experimental hill chart of Francis 99 turbine with numerical simulation undertaking Standard k-ε model; the numerical efficiency was over predicted within a deviation of 2.87% for the whole hill chart. The leakage flow through the clearance in the rotor-stator region causes dynamic instability at the runner, reduces the efficiency of the turbine during continuous operation. Increasing the clearance gap linearly declines the performance of the turbine [10]. It was reported that vortex filament though the clearance gap originated at the tip of guide vane, reduces the stagnation angle at the inlet of the runner blade [11].

This paper investigates the performance of the turbine of Bhilangana hydropower plant (B-III) located at Uttarakhand state, India. It is a run of river (ROR) hydropower project having a capacity of 24 MW (8x3 MW) equipped with three horizontal axial Francis turbine units.

### Table 1: Specification of Francis turbine

| Specification of Francis Turbine | Details |
|---------------------------------|---------|
| Rated power (P)                 | 8.269MW |
| Rated head (H<sub>net</sub>)    | 213.25m |
| Nominal head (H)                | 207m    |
| Rated discharge (Q)             | 4.33m³/s|
| Rated speed (N)                 | 750RPM  |
| Runner diameter (D)             | 1.185m  |
| Unit Speed (n<sub>1p</sub>)     | 60.88   |
| Unit Discharge (Q<sub>1p</sub>) | 0.211   |

2. Numerical model

2.1 Geometric and discretization

Complete assembly of prototype Francis turbine was considered for numerical investigation. The components of the turbine are identical to the design dimension. The computational domain of the turbine includes two stationary domains, namely a distributor and a draft tube, and a rotating domain.
Total 16 stay vanes, 16 guide vanes, and 13 runner blades were assembled for simulation, as shown in figure 1. In ICEM CFD, the unstructured mesh was generated in one stationary and the rotating domains of the turbine, which consists of a tetrahedral element with a prism layer near to the surface wall to deal with boundary layer conditions. A structured mesh was created using a multi-block technique in the draft tube. The runner blade is complex in geometry; hence tetrahedral meshing was applied, as shown in figure 2. The runner and guide vane are the regions of study, so prism layers were generated near to the wall with an initial mesh size of 0.1 mm.

![Computational model of Francis turbine](image)

**Figure 1.** A computational model of Francis turbine.

| Domain        | M1         | M2         | M3         |
|---------------|------------|------------|------------|
| Distributor   | 5141025    | 7713039    | 14062046   |
| Runner        | 2822281    | 4108724    | 8548120    |
| Draft Tube    | 391721     | 2250114    | 6490829    |
| Total Elements| 8355027    | 14071877   | 29100995   |

**Table 2.** Number of elements for different domain

![Mesh generation](image)

(a) (b)
2.2 Governing equations

CFD solves the mathematical modeling equations of fluid flow using computational resources. Commercial CFD solver ANSYS 19.1 has been used for steady-state simulation with a high-resolution advection scheme. ANSYS CFX software solved the steady Navier stokes equation in the following form:

\[
\frac{\partial (\rho u_j)}{\partial x_j} = 0
\]  \hspace{1cm} (1)

\[
\frac{\partial (\rho u_i u_j)}{\partial x_i} = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + S
\]  \hspace{1cm} (2)

Where \( \rho \) is density, \( u_j \) is velocity vector, \( p \) is pressure, \( S \) is the sum of body forces, and \( \tau_{ij} \) is stress tensor. The continuity and momentum equations form a set of four equations in four unknowns, velocities in three directions \((u, v, w)\) and pressure \( (p) \). The Navier stoke equation is a highly nonlinear partial differential equation [12].

2.2.1 Turbulence model

The standard k-\( \varepsilon \) model uses the scalable wall functions, and it is ideal for predicting the flow behavior in the regions away from the boundary wall. It does not account for the flows with boundary separation, flow over a curved path, rotating fluids, sudden changes in the man strain rate, etc. [12]. The shear stress transport (SST) model is a combination of the standard k-\( \varepsilon \) model and k-\( \omega \) model from the inner part through the viscous sub-layer to the turbulent layer. The SST model switches to a k-\( \varepsilon \) behavior in the free-stream, which overcomes problems issued for standard k-\( \varepsilon \). The SST model resolves transport equations of turbulence shear stress and forecast about flow separation happening under an unfavourable pressure gradient. The switching of the SST model between standard k-\( \varepsilon \) and k-\( \omega \) model is ensured by using a blending function \( F1 \) and \( 1-F1 \) are multiplied to k-\( \omega \) and standard k-\( \varepsilon \) model respectively [12,13]. In these equations, both the velocity and the length scale are solved using separate transport equations. Hence, the equation becomes:

\[
U_j \frac{\partial k}{\partial x_j} = P_k - \beta' k \omega + \frac{\partial}{\partial x_j} \left( \nu + \sigma_k \nu_f \right) \frac{\partial k}{\partial x_j}
\]  \hspace{1cm} (3)

\( \beta' \) is a blending function that varies from 0 to 1. 

Figure 2. Mesh (a) Runner (b) Inflation layers near the runner wall and (c) draft tube component.
Specific Dissipation Rate

\[ U_j \frac{\partial \omega}{\partial x_j} = \alpha S^2 - \beta \omega^2 + \frac{\partial}{\partial x_j} \left[ (\nu + \sigma_{\omega} v_T) \frac{\partial \omega}{\partial x_j} \right] + 2(1-F1)\sigma_{\omega^2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \] (4)

2.3 Boundary condition

In a steady-state simulation, two relative domains are connected using the frozen rotor connections. Ansys CFX provides flexible boundaries set up options for simulation, which are as follows: (i) Mass flow rate as an inlet boundary condition and steady-state outlet pressure at outlet boundary condition (ii) Both inlet and outlet steady-state pressure boundary condition. The flow rate corresponding to the percentage opening of guide vane was known from the prototype hill chart provided by the manufacturer, so inlet and outlet pressure boundary conditions were applied to calculate the corresponding guide vane angle. The simulation has been performed at the rated speed of 750 RPM with load ranges from 130% rated load to 77% load to predict the close guide vane angle. The pressure values at both inlet and outlet is used as boundary conditions and as summarized in Table 3.

| Parameter                          | Description                                                                 |
|------------------------------------|-----------------------------------------------------------------------------|
| Component                          | Stationary Domain (Casing, 16 stay vane, 16 guide Vanes, Draft tube Rotating Domain: Runner (13 blades, Hub & Shroud) |
| Analysis type                      | Steady-state                                                                |
| Interface                          | GGI (Interface I: Between the distributor and runner Interface II: Between the runner and draft tube) |
| Fluid                              | Incompressible Newtonian fluid: water                                      |
| Boundary condition                 | Inlet: Total pressure corresponding to rated head Outlet: Opening type with static pressure |
| Turbulence intensity               | 5%                                                                          |
| Reference pressure                 | 0 [Pa]                                                                      |
| Wall                               | No-slip                                                                     |
| Turbulence model                   | Standard \( k-\varepsilon \) and \( k-\omega \) SST                       |
| Convergence control                | RMS parameters \( \leq 10\text{E}^{-5} \)                                   |
| Discretization and Advection scheme| High resolution                                                             |
| The solution controls turbulence numeric | High resolution                                                          |

2.4 Mesh sensitivity

Celix et al. [14] described the grid convergence index (GCI) method to assess the extrapolation values and discretization error in mesh convergence. A mesh independency test was carried out at BEP point by creating three different mesh densities, namely coarse, medium and fine, which consist of 8.35, 14.07, and 29.10 million elements, respectively, with mesh refinement factor of 1.3X. The size of mesh was taken in such a way that the \( Y^+ \) value is less than 200, which satisfies the requirement of turbulence model \( k-\omega \) SST used in simulations [12, 13]. Mesh quality is a crucial parameter for controlling the discretization errors and the numerical uncertainty in the solution, inadequate quality of mesh may cause the diverged solution. Therefore, the quality of mesh was maintained above 0.27, which was above the minimum acceptable limit of ANSYS CFX – Solver [12]. The M1 is fine mesh and M3 is coarse mesh.
and a corresponding average height of mesh size is in order of $h_1 < h_2 < h_3$. The computed flow parameter for the present numerical model was presented in Table 4. Uncertainties in the numerical calculation were estimated by monitoring flow rate, torque, and pressure points inside the turbine. Three monitoring pressure points have taken at a location of vaneless space region (VL01), draft tube (D01), and runner (R01), respectively. Uncertainties in torque values were found as 2.1% and 2.66% for fine and medium meshes, respectively. Numerical uncertainties for medium-mesh was lower than that for the fine mesh solutions for flow rate (0.2%) and draft tube pressure (0.8%). The medium-mesh has been used for further simulation at different operating conditions.

| Parameter | Torque [kN-m] | Flow rate [m³/s] | Efficiency [$\eta_h$] | VL01 [kpa] | D01 [kpa] | R01 [kpa] |
|-----------|---------------|-----------------|------------------------|-------------|-----------|-----------|
| $r_{21}$  | 1.36          | 1.36            | 1.36                   | 1.36        | 1.36      | 1.36      |
| $r_{32}$  | 1.32          | 1.32            | 1.32                   | 1.32        | 1.32      | 1.32      |
| $\phi_1$  | 105.93        | 4.341           | 91.58                  | 1197.48     | 798.52    | 235.052   |
| $\phi_2$  | 105.491       | 4.323           | 91.59                  | 1202.5      | 763.19    | 233.44    |
| $\phi_3$  | 105.004       | 4.315           | 91.33                  | 1198.6      | 753.96    | 231.51    |
| $k$       | 0.705         | 2.187           | 11.6955                | 0.77        | 3.81      | 0.982     |
| $\Phi_{ext}^{21}$ | 107.743 | 4.359           | 91.57                  | 1178.93     | 814.35    | 239.58    |
| $e_{a}^{21}$ | 0.00414 | 0.0041          | 0.000109               | 0.00419     | 0.044     | 0.0068    |
| $e_{ext}^{21}$ | 0.0209  | 0.0084          | 0.000112               | 0.01999     | 0.06282   | 0.0256    |
| $GC_{fine}^{21}$ | 0.0214 | 0.0054          | 3.85E-06               | 0.01936     | 0.02478   | 0.0241    |
| $GC_{med}^{21}$ | 0.02669 | 0.00276         | 0.000143               | 0.01679     | 0.00802   | 0.0328    |

3. Results and discussion

Steady-state simulations have been performed at design and off-design operating conditions. The simulated results were compared with the prototype hill chart provided by the manufacturer. The hydraulic efficiency of the turbine computed from the simulation was calculated as

$$\eta_h = \frac{T \times \omega}{(p_1 - p_2) \times Q}$$

Where $\eta_h$ = Hydraulic efficiency, $T$ = Torque, $\omega$ = rotational speed of runner, $p_1$ = Inlet Pressure, $p_2$ = Outlet pressure, $Q$ = Discharge in the turbine.

Figure 3 shows the comparison of discharge values for the prototype testing and simulated results. The numerical discharge obtained by the standard k-ε model was higher than the experimental discharge at all three operating conditions. The difference in discharge is obtained as 0.23 m³/s, 0.17 m³/s, and 0.15 m³/s at full load, BEP, and part load condition (77% of BEP), respectively. Similarly, the numerical discharge obtained by k-ω SST was lower than the experimental discharge at all conditions. The difference of 0.073 m³/s at full load, 0.01 m³/s at BEP, and 0.04 m³/s at part load condition is found for k-ω SST. The discharge computed by k-ω SST turbulence model was observed very much close to the
prototype test results. The observed variation on discharge among the two turbulence models simulating under the same head may be occurred due to different turbulence models in the Navier stroke equation.

![Figure 3](image1.png)

**Figure 3.** Comparison of experimental and computed discharge at different guide vane opening.

Figure 4 shows the comparison of torque obtained by numerical simulation with power developed by the prototype turbine at three different conditions. The torque computed from the simulation was higher than the prototype test results. The torque computed was very close at BEP and deviates more at part load conditions. A difference in experimental torque and numerical torque by k-ω SST were 0.39 kN-m, 0.10 kN-m, 0.556 kN-m and by the standard k-ε model were 1.45 kN-m, 1.18 kN-m, 2.78 kN-m at full load, BEP and part load condition, respectively. The rotor-stator interaction between runner blades and guide vanes develops complex pressure and velocity fields, which may have attributed the torque fluctuations, especially at low discharge operating point due to the presence of the larger vaneless space between the guide vanes and the runner blades [2].

![Figure 4](image2.png)

**Figure 4.** Comparison of numerical and experimental torque different guide vane opening.
Figure 5. Comparison of experimental and computed hydraulic efficiency of the turbine at different operating conditions

The experimental efficiency was higher than the computed efficiency at full load condition and lowered in other operating conditions as shown in figure 5. The experimental efficiency was higher by 1.71% (k-ω SST) and 1.69% (k-ε), at full load, and lower by 0.24% (k-ω SST model) and 0.2% (k-ε) at BEP condition. Similarly, at part load condition, experimental efficiency lower by 1.8% (k-ω SST) and 1.6% (k-ε), respectively. The numerical efficiency obtained by CFD was very close to the experimental efficiency at BEP and deviates at part load and high load. The hydraulic efficiency depends on torque and discharge in the turbine. The turbulence models performed well near BEP, where the flow was expected to be attached to the wall. When operating away from BEP, the turbulence models unable to accurately resolve the flow features well, such as separation, vortex breakdown, which lead to underestimated the losses and predicted inaccurate efficiency, particularly at the off-design condition. The numerical result provided only one highest efficiency point that was at BEP and trend to decrease toward high load and part load conditions; however, in the prototype turbine high efficiency existed at 100% guide vane opening condition. This may due to high discharge in the prototype turbine. The flow phenomenon predicted by simulation is based on the selection of the turbulence model. The turbulence model resembles a similar kind of flow at design and off-design conditions as suggested by the past experience [2,10] as the difference in flow phenomenon, the energy distribution by the flow on the turbine is also a reason for having a difference in CFD and experimental result. The presence of a clearance gap in the guide vane causes flow disturbance in the vaneless region, increases hydraulic losses, and reduce the efficiency of the turbine [10]. Next section deals with the effect of the clearance gap on efficiency.

3.1 Effect of leakage flow on efficiency

A small clearance gap (CG) is provided at both ends of guide vane and facing plate to adjust the guide vane at different opening angles based on various operating conditions. During the operation of the turbine, the dry clearance gap increases due to sediment erosion initiated by leakage flow. A clearance gap of 1.85 mm on each side has been undertaken to investigate the effect of the clearance gap on turbine efficiency. Guide vane regulates the flow and convert a large portion of pressure energy into kinetic energy, thus, increasing meridional velocity and decreasing pressure toward guide vane passage. Due to this energy conversion, pressure difference existed. The flow is accelerated toward the runner, and the flow velocity is higher at the trailing edge compared to the leading edge. A relative reduction in pressure
across the guide vane results in wake formation at the end of the trailing tip as shown in Figure 6. The provided clearance gap causes the flow variation in streamline flow from guide vane to a runner.

**Figure 6.** (a) pressure counter in guide vane at mid-span, (b) wake flow at the end of the trailing edge.

Figure 7 shows the leakage cross-flow through the clearance gap. The pressure difference across the guide vane drive cross leakage flow, the flow intensity is more at trailing edge than leading-edge, resulting in high crossflow at trailing edge of the suction side of guide vane. The cross-flow and leakage flow form a secondary flow, which affects the turbine performance. The streamline on the plane shows high turbulence flow through the clearance gap and vortices at the suction side of leading-edge and trailing edge of guide vane. The vortices formed at leading-edge fade away in downstream due to changing of fluid velocity, as shown in Figure 8. The cross-flow and leakage flow mix with the main flow to form a rotating component at suction side by intense vortices formed at trailing edge, i.e., vertex flow where vertex filament originated which move towards the runner.

**Figure 7.** (a) Measurement plane at guide vane, (b) Radial flow in guide vane through the clearance gap

**Figure 8.** Vortex flow on the plane for an upstream region of guide vane at 1.85mm clearance gap.

The clearance gap losses the efficiency of the turbine. Leakage flow through clearance gap disturbs the main flow entering toward the runner resulting in an undesirable flow such as secondary flow, vortex flow, wake flow, and cross-flow, which has considerable mechanical and hydraulic effects in the turbine. Such flow reduces the velocity around the guide vane causes the pressure drop and trend in lowering the
energy conversion. The leakage flow, wake flow through clearance gap never enter into the runner as it undergoes dissipation in vaneless space before reaching the runner; thus, the energy carried by leakage flow remains unutilized, bearing a significant part of the loss in Francis turbine [15]. The vortex flow driven by leakage flow through the clearance gap changes the flow direction at the inlet. It causes a reduction of stagnation angle (α), which reduced the tangential component of absolute velocity (Cu) at the inlet of the runner [10]. The hydraulic efficiency of the runner depends on the tangential velocity component according to Euler’s equation, which eventually reduces the efficiency.

The efficiency with no clearance gap and with a clearance gap of 1.85 mm was compared. The numerical result obtained by k-ω SST turbulence model was taken for comparison. The leakage flow through the 1.85 mm clearance gap resulted in a decrease in the efficiency of the turbine. The efficiency drops around 5.73% at BEP condition, 5.83% at full load condition, and 5.27% at part load condition, as shown in Figure 9.

![Figure 9. Comparison of efficiency at no clearance and with a clearance gap of 1.85 mm](image)

Three operating conditions of turbine summarized in Table 5.

| Guide-vane opening | Prototype turbine flowrate (m³/s) | Computed flow Rate (k-ω SST) (m³/s) | Percentage error in flow rate (%) | Load (%) |
|--------------------|----------------------------------|--------------------------------------|----------------------------------|----------|
| 100%               | 5.57                             | 5.64                                 | 1.25                             | 130%     |
| 77%                | 4.337                            | 4.323                                | 0.322                            | 100%     |
| 60%                | 3.36                             | 3.34                                 | 0.59                             | 77%      |

4. Concluding Remarks

Hydraulic performance analysis of Francis turbine was performed by a numerical approach under the k-ε turbulence model and k-ω SST turbulence model at design and off-design conditions. Based on the comparison of CFD result with the prototype test results (manufacturer data) of Francis turbine, the following conclusions are drawn:

(i) The efficiency obtained by the standard k-ε turbulence model provided a small difference with the experimental efficiency. At the best efficiency point, the lowest difference between
numerical and experimental efficiency was found (0.2%) and maximum difference at the off-design condition like 1.71% at full load and 1.8% at part load was observed.

(ii) The maximum error of 1.25% is observed in flow at full load condition for $k$-$\omega$ SST model.

(iii) The presence of a clearance gap in the guide vane causes secondary flow, vortex flow, wake flow in the turbine which increases the hydraulic losses, pressure fluctuation, and affects the overall performance of the turbine.

(iv) The presence of clearance gap in guide vane reduced the efficiency around 5% at both design and off-design conditions.

References

[1] Hydropower Status Report 2019 by the International Hydropower Association. https://www.hydropower.org/status2019#:~:text=The%202019%20Hydropower%20Status%0.
[2] Trivedi C, Cervantes M J, Gandhi B K and Dahlhaug O G 2013 Experimental and numerical studies for a high head Francis turbine at several operating points. *Journal of Fluids Engineering*, 135(11).
[3] Laín S, García M, Quintero B and Orrego S 2010 CFD Numerical simulations of Francis turbines. *Revista Facultad de Ingeniería Universidad de Antioquia*, 51, 24-33.
[4] Nordvik A, Iliev I, Trivedi C and Dahlhaug O G 2019 Numerical prediction of hill charts of Francis turbines. *Journal of Physics: Conference Series* Vol. 1266, No. 1, IOP Publishing.
[5] Trivedi C, Gogstad P J and Dahlhaug O G 2018 Investigation of the unsteady pressure pulsations in the prototype Francis turbines–Part 1: steady state operating conditions. Mechanical Systems and Signal Processing, 108, 188-202.
[6] Anup K C, Thapa B and Lee Y H 2014 Transient numerical analysis of rotor-stator interaction in a Francis turbine. *Renewable Energy*, 65, 227-235.
[7] Prasad V, Gahlot V K and Krishnamachar P 2009 CFD approach for design optimization and validation for axial flow hydraulic turbine, *Indian Journal of Engineering and Material Sciences*, vol. 16, 229-236.
[8] Yaping Z, Weili L, Hui R and Xingqi L 2015 Performance study for Francis-99 by using different turbulence models. *Journal of Fluids Engineering* Vol. 579, No 1, IOP Publishing.
[9] Yulin Wu, Jintao Liu, Yuekun Sun, Shuhong Liu and Zhigang Zuo 2013 Numerical analysis of flow in a Francis turbine on an equal critical cavitation coefficient line. *Journal of Mechanical Science and Technology*, 27, 1635-1641.
[10] Chitrakar S, Dahlhaug O G and Neopane H P 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, 662-667.
[11] Chitrakar S, Neopane H P and Dahlhaug G 2017 Recent findings related to sediment erosion in Francis Turbines. *AFOR*, 149-149.
[12] ANSYS, Release 11.0 2006 Solver Modelling.
[13] ANSYS, C F X 2009 ANSYS CFX-Solver Theory Guide.
[14] 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, *ASME, J Fluids Eng.*, 135.
[15] Koirala R, Zhu B and Neopane H P 2016 Effect of guide vane clearance gap on Francis turbine performance. *Energies*, 9(4), 27.