Numerical study of the Winter-Kennedy method for relative transient flow rate measurement

To cite this article: B Baidar et al 2019 IOP Conf. Ser.: Earth Environ. Sci. 405 012022

View the article online for updates and enhancements.
Numerical study of the Winter-Kennedy method for relative transient flow rate measurement

B Baidar¹, J Nicolle², B K Gandhi³, M J Cervantes¹
¹ Department of Engineering Sciences and Mathematics, Luleå University of Technology, Luleå 971 87, Sweden
² Institut de recherche d’Hydro-Québec, Varennes, QC J3X 1S1, Canada
³ Department of Mechanical and Industrial Engineering, Indian Institute of Technology Roorkee, India

E-mail: binaya.baidar@ltu.se

Abstract. The Winter-Kennedy (WK) method is used to estimate relative flow rate using the differential pressure between two taps located at a radial section of a spiral casing (SC). It is widely used in index testing, for double regulated turbines optimization and sometimes for continuous discharge measurement in low head plants. This paper explores the possibility of using the WK method for relative transient flow rate measurements.

A numerical model of a Kaplan model turbine from the penstock to the distributor has been developed. Unsteady RANS simulations with k-ω SST turbulence model are performed. Previously conducted experiments on the model turbine are used to validate the numerical results. In the simulations, the guide vanes (GVs) are closed from 26.5°, the best efficient point (BEP), to about 5° opening angle. Two azimuthal locations of the SC and four different WK configurations at each location are considered. The variation of the WK coefficients with time are investigated and compared to the ones at several stationary GV angles. The results showed a difference between the WK coefficients obtained at transient and stationary operations. However, there may be a possibility of using the WK method during transients by locating the pressure taps in appropriate locations for an acceptable variation of the WK coefficient from its BEP value.

1. Introduction
There is a rapid growth of intermittent renewable energy sources like wind and solar energy in present years. The integration of the electricity generated from these sources into the grid results in grid fluctuations. The hydraulic turbines are commonly used to balance the grid and hence undergo frequent hydraulic transient events. Several researchers have been conducting studies on the hydro-mechanical effects of transients, like wear and tear of the machine components [1, 2]. The flow rate (discharge) is an important parameter to know in order to perform any modelling. Apart from these studies, it is also of interest for hydropower companies to quantify the discharge to calculate the efficiency loss during such operations.

It is difficult to quantify the discharge in low head plants. One of the popular methods to measure the flow rate in a low head plant is the Winter-Kennedy method. In the method, a pair of pressure taps are placed at a different radius in a section of the spiral casing (SC). The discharge Q follows the relation:
where $K_{WK}$ is the WK coefficient and $n$ is an exponent whose value varies from 0.48 to 0.52 [3]. \( \Delta P \) is the differential pressure between the outer and inner pressure taps placed on the SC. The method is popular due to its cost-attractiveness and no downtime of the machine if the pressure taps have already been installed. The method may also be used for continuous discharge measurements [4].

The present work explores the possibility of using the WK method for transient flow rate measurements in a low head machine. A numerical model of a Kaplan turbine model from the penstock to the distributor is developed. The CFD simulations were carried out using unsteady RANS simulations. The WK coefficients during the transient and steady operating conditions are studied and reported.

2. Test case and numerical setup

The turbine model of a unit of the Porjus hydropower plant, Porjus U9, located in Sweden is considered. The turbine’s nominal head is 55 m and the maximum flow rate through the turbine is approximately 20 m$^3$/s. A 1:3.1 scaled model of the U9 unit is considered for this study. The turbine is composed of a penstock, a full spiral casing, a distributor consisting of 18 stay vanes (SVs) and 20 guide vanes (GVs), a Kaplan runner with six blades and an elbow draft tube.

2.1. Numerical setup

The computational domain in this study consisted of the penstock, spiral casing, and distributor; see Figure 1(a). The mesh for all the domains was generated using the software ICEM CFD. The total number of mesh elements was 9.63 million: 1.44 million in the penstock domain, 1.82 in the SC and 6.37 in the distributor. All mesh elements were hexahedrons with a minimum angle of 18 degrees, see Figure 1(b). For the distributor, the mesh was created for a passage consisting of a GV and an SV, then the passage was copied and rotated to create the distributor. The periodic faces were considered ensuring one to one mapping between the GV interfaces. The general grid interface (GGI) was considered between the domains. A similar set of mesh was found to be mesh independent in the study presented by Amiri et al. [5].

![Figure 1. a) Computational domain, b) mesh at the spiral casing and guide vane](image)

Numerical simulations were performed using ANSYS CFX. The Unsteady-Reynolds-averaged Navier-Stokes (URANS) equations were utilized for the simulations. The ‘High Resolution’ spatial discretization scheme was used, in which the discretization is achieved by varying the blend factor from 0.0 to 1.0 based on the local solution field. The second order backward Euler scheme was used to discretize time. All the simulations were performed using the Menter’s two-equation shear stress transport (SST) model [6] using the automatic wall treatment implemented in the software.
A total pressure of 31 kPa was imposed at the inlet to match the flow rate at BEP. The flow direction was normal to the inlet without boundary layer, i.e. plug flow. The turbulence intensity of 5% was considered at the inlet. An opening boundary condition with a relative pressure (0 Pa) was considered at the outlet. The same boundary conditions were imposed for all set of simulations performed in this study, including the simulations for all the stationary GV opening cases. The convergence criterion on the root mean square (RMS) residual was set to $1 \times 10^{-5}$ and various parameters including the WK coefficients for all the WK configurations were monitored to ensure convergence.

Two time-steps were investigated at the BEP condition, i.e., the guide vane opening angle (GVA) of 26.5°. The time steps considered were $2.26 \times 10^{-2}$ and $5.66 \times 10^{-3}$ s. The time steps correspond to 0.2° and 0.05° of the GVs closure speed of $8.83 \text{°}/\text{s}$. The considered time steps showed a similar result with a negligible difference in the discharge and WK coefficients prediction. Therefore, the time step corresponding to 0.2° of the GVs closure speed was chosen for the study.

2.2. WK configurations

Two azimuthal locations of the SC at $\theta = 60°$ and $124°$ were considered for the study, see Figure 2(a). At each cross-section four WK configurations: WK$_{1u}$, WK$_{1d}$, WK$_{2u}$, and WK$_{2d}$ were considered. The locations of the respective pressure points and the related WK configurations are shown in Figure 2(b).

![Figure 2](image-url)  
**Figure 2.** Locations of the WK pressure points and measurement locations for validation studies; a) Top view of the SC showing azimuthal locations, $\theta = 60°$ and $124°$, for the WK configurations, $S_I$ and $S_{II}$ represent the velocity measurement locations conducted by Mulu and Cervantes [7], b) WK configurations at a section with their pressure point locations.

The WK coefficient $K_{WK}$ is calculated from equation (1), with the exponent $n = 0.5$. The differential pressure $\Delta P$ is calculated by the pressure difference between the outer pressure ($P_{ou}$ or $P_{od}$) and an inner pressure ($P_{1u}$, $P_{2u}$, $P_{1d}$ or $P_{od}$); represented by the lines in Figure 2(b).

3. Results and Discussion

3.1. Validation studies

The tangential and radial velocities from a previously conducted experimental study by Mulu and Cervantes [7] and present numerical study are shown in Figure 3(a) and 3(b), respectively. The measurement locations, $S_I$ and $S_{II}$, were indicated in Figure 2(a). The experiment was conducted using a two-component LDA system with an 85 mm fiber optic probe from Dantec. The overall trend of the tangential and radial velocities shows a good agreement with the experimental results. At section $S_I$, the tangential velocity is larger towards the bottom of the SC and decreases towards the top wall. But at section $S_{II}$, the tangential velocity is smaller towards the bottom of the SC and increases towards the top wall. The magnitude of the tangential velocity is also in good agreement with the experimental results.
However, the magnitude of the radial velocity towards the SC bottom shows a larger discrepancy with the experiment; indicating an over-prediction of the secondary flows. The upper tank was not modeled in this study, which may have caused the discrepancy. The study conducted by Amiri et al. [5] showed the secondary flow structure can be influenced by the interaction from the penstock, upstream tank and the spiral casing.

3.2. GV closure set up and discharge
The GVs were closed from their BEP positions at 26.5° to about 5° GVA. The GV closure time was set to 3s, i.e., the closure speed of 8.83°/s. In order to maintain the quality of the mesh, meshes at three different GVAs (BEP, 18° and 8°) were created. The simulations were performed by considering the overlap between two consecutive simulations to ensure the continuity in the solutions – which were ensured with the monitored variables. The discharge as a function of the time and GVA is shown in Figure 4. The discharge shows an almost linear relation with the GVA. The discharge at five different guide vane angles at steady operations also follows the transient discharge, with a maximum deviation of ~7%.

3.3. WK coefficients during the transient
Figures 5(a) and 5(b) show the variations of the WK coefficients with time at \( \theta = 60^\circ \) and 124°, respectively. The coefficients are normalized by their values at BEP condition. The lines (continuous...
and dashed black and red lines) show the WK coefficients during the transient; the dots and circles show the coefficients at stationary GV positions for the respective WK configurations. The variation of the WK coefficients with time for the considered WK configurations at $\theta = 60^\circ$ and $124^\circ$ is different. This signifies the dependence of the coefficients on the WK configurations and their azimuthal locations. The figures show WK$_{1u}$ and WK$_{1d}$ values decline at $\theta = 60^\circ$ while they increase at $\theta = 124^\circ$; and vice versa for WK$_{2u}$ and WK$_{2d}$. Therefore, it seems that there could be certain $\theta$ locations in the SC where the WK behavior is more stable during the transient.

The WK coefficients during the transient vary with the coefficients at stationary operations, although some points seem to match for WK$_{2u}$ and WK$_{2d}$ at the beginning of the closure. WK$_{2u}$ configuration at $\theta = 60^\circ$ shows the least sensitivity for which the maximum deviation between the transient and stationary condition was about 6% until 5s, i.e., GV closure of about 18$^\circ$ from its BEP angle. The deviation increases with the decrement of the GV angle. It is reasonable to have different values of the coefficients during transient and stationary operations, as the flow takes a while to stabilize. The WK coefficients are usually averaged for some time (few minutes) to get average results on site as well. The results show that the stationary WK coefficient may be difficult to use for the transients.

Figure 5. Variation of the WK coefficients with time and guide vane angle (GVA). The markers (dots and circles) show the WK coefficient at stationary conditions, i.e., at fixed GV angles for the respective WK configurations: ○ WK$_{1u}$, ● WK$_{1d}$, ○ WK$_{2u}$, ● WK$_{2d}$.

Figure 6 shows the variation of the pressure difference with time for the considered WK configurations. The pressure difference for all the configurations tends towards zero irrespective of how large the pressure difference is at the BEP condition. So, the reduction rate is larger for the WK configurations with larger pressure difference, i.e., for WK$_{2u}$ and WK$_{2d}$. These configurations are also relatively stable at the beginning of the transient (see Figure 5).

Figure 6. Variation of the pressure difference ($\Delta P$) with time and guide vane angle (GVA).
4. Conclusion

The WK configurations with larger pressure difference are more stable than with a smaller pressure difference at the beginning of the transient. During the later stage, the coefficients change significantly and depend on the azimuthal location of the spiral casing and the WK configuration. The study also showed the transient WK coefficients varies from the coefficients at stationary conditions; therefore, the coefficients obtained at stationary conditions may not be used for the transients. From the present study, the WK2u configuration at $\theta = 60^\circ$ is the most stable location for a transient WK measurement, with the maximum deviation of about 6% until the GVA of about 9° (~18° closure from the BEP angle). Further studies are planned to locate more WK configurations at several azimuthal locations and study their behaviors. The experimental studies are also planned to explore the possibility of using the WK method in transients and for numerical validations.

Acknowledgment

The research presented was carried out as a part of ‘Swedish Hydropower Centre- SVC”. SVC has been established by the Swedish Energy Agency, Elford and Svenska Kraftnät together with Luleå University of Technology, The Royal Institute of Technology (KTH), Chalmers University of Technology, and Uppsala University (www.svc.nu).

References