Study of Flow Characteristics inside Francis Turbine Draft Tube with Adjustable Guide Vanes

Jesline Joy\textsuperscript{1*}, Mehrdad Raisee\textsuperscript{2} and Michel J. Cervantes\textsuperscript{1}

\textsuperscript{1}Department of Engineering Sciences and Mathematics, Luleå University of Technology, Sweden
\textsuperscript{2}Hydraulic Machinery Research Institute, School of Mechanical Engineering, College of Engineering, University of Tehran, Iran

\*Corresponding author: jesline.joy@ltu.se

Abstract. Numerical investigation was performed on a semi-model (one stay vane, two guide vanes, one runner passage inclusive of one main and one splitter blade, and the draft tube) of a high-head Francis turbine model with adjustable guide vanes in the draft tube. The motive of the present study is to investigate the possibility to mitigate the rotating vortex rope (RVR) formed at part load operating condition. Each guide vane in the draft tube consists of two hydrofoils. The upper hydrofoil is adjustable according to the flow angles leaving the runner. The lower hydrofoil is stationary and corresponds to the flow angle at best efficiency point (BEP). The factors considered while designing these guide vanes were a) number of guide vanes, b) chord length, c) span and d) position in the draft tube. The preliminary results indicate that the RVR pressure amplitude was suppressed by 97\% compared to the reference model with no guide vanes at part load (PL) operating condition. An 8.7\% increment in the draft tube pressure recovery was obtained which indicates that implementation of the guide vanes in the draft tube could positively impact the turbine efficiency beside the mitigation of the pressure pulsations at PL operation.

1. Introduction

Hydraulic turbines are considered as one of the most favoured energy-efficient technique for power production. Such techniques are widely preferred due to their less complicated setup and lower maintenance costs [1-2]. Hydraulic turbine can operate with a large efficiency at BEP operating condition [1-2]. However, the efficiency is largely compromised at lower operating conditions due to development of RVR in the draft tube, which has not been resolved yet [3]. RVR is a ‘progressive’ type of helical vortex formed due to self-induced unsteadiness caused by swirling flows within the draft tube. The pressure fluctuations of RVR were found to be most dominating at one-third of the runner frequency approximately [3-6]. The presence of RVR is usually characterize by two modes: the plunging (synchronous) and rotating (asynchronous) modes, respectively, which is explained further in section 3.1. The understanding of RVR formation and development could be quite useful to mitigate the RVR or suppress its adverse effects [7].

Though, several attempts were made in the past to mitigate RVR in the draft tube [7-14], the issue of pressure pulsations in the draft tube has not been resolved yet. In the present study, the influence of a guide vane system in the draft tube of Francis model turbine has been investigated numerically. As the present study is at its preliminary stages, the main objective is only to assess the possibility of RVR mitigation or reduction of pressure pulsations with a guide vane system in the draft tube. The guide vane system is expected to alter the swirling flow at PL operating condition, to reduce the RVR pressure pulsations. The present paper will also present an extensive comparison of flow behaviour in terms of pressure pulsations, tangential velocity component and pressure recovery of Francis model turbine with and without a guide vane system the draft tube.
2. Numerical Approach

2.1. Geometry description

The Francis model turbine considered in the present study is taken from NVKS Francis-99 second workshop [15]. The model turbine consists of a spiral casing, 14 stay vanes, 28 guide vanes, one runner with 15 main and 15 splitter blades and a draft tube diffuser. The runner diameter is 349 mm. The model turbine has a cascade-type arrangement of stay vanes, guide vanes and runner blades, which divides the domains into passages [2]. The flow in these passages is usually considered periodic and the influence from the flow in neighbouring passages are usually considered negligible. Therefore, to save computational time and data storage, a standard passage flow based numerical approach was considered for the present study. This means that the numerical domain in the present study consists of a passage of the distributor domain (one stay vane and two guide vanes), a runner passage (one main blade and one splitter blade) and the draft tube domain. This passage domain is called Francis-99 ‘semi-model’. The three-dimensional computer-aided design (CAD) model of Francis-99 semi-model turbine is presented in figure 1. For pressure and velocity measurements, the details of the monitor points are available in NVKS Francis-99 second workshop [15]. Additionally, 18 new monitor points (J1-J18) were created to monitor the pressure pulsations around the mid-section of the newly designed guide vanes. These monitor points are evenly distributed circumferentially in the draft tube. To monitor the pressure pulsations along the draft tube domain, 6 new monitor points (MC1-MC6) were created. The details of the monitor points are given in Appendix-A in Tables A1 and A2, respectively.

![Figure 1 Computational domain of Francis-99 semi-model, with adjustable guide vanes in the draft tube](image)

2.2. Guide vanes in the draft tube

Guide vanes are mechanical devices that can be useful in re-directing the flow, so that the tangential component is suppressed to reduce the excess swirl. To design the guide vanes, there were certain parametric design considerations as follows:

- Number of guide vanes to be placed inside the draft tube,
- Location of the guide vane inside the draft tube,
- Inlet and outlet angle of the guide vane,
- Chord-length of the guide vane,
- Span of the guide vane.

The study performed by Nishi et al. [13] and Zhou et al. [14] indicate that 2-4 external objects (like fins or baffles, respectively) inside the draft tube can suppress RVR. For the present study, a set of three
guide vanes (3GV) was considered. The 3GV system was evenly distributed circumferentially across the draft tube. This means that each guide vane is 120° apart from other circumferentially. The reason to use three guide vanes set is to minimize fluid-surface interaction, i.e., minimize losses. The guide vanes were positioned at a reasonable distance between the runner exit and the draft tube elbow. This way strong rotor-stator interaction (RSI) effect could be avoided in the draft tube at PL. The inlet and exit angle of the guide vane was calculated based on flow angles at PL and BEP operating condition, respectively at the desired guide vane location.

The chord-length of the guide vane system was considered as 86% of the runner radius. The span of the guide vane was 30% of the runner radius. As the current study is at its preliminary stage, the profile shape of the guide vanes was not considered. Hence, the thickness of the guide vane was set to 2 mm. The guide vanes system was designed in ANSYS BladeGen [16]. As mentioned earlier, a single guide vane consists of two hydrofoil blades. The upper hydrofoil is aimed to be adjustable according to the operating conditions and the lower hydrofoil is stationary with the outlet angle corresponding to the flow at BEP. The guide vane system in the draft tube is presented in figure 1.

2.3. Mesh, boundary conditions and solver

The mesh of the reference model domains is available from NVKS Francis-99 second workshop [15]. The details of the mesh have been presented by Trivedi et. al. [1-2]. The mesh for the distributor and runner domains were considered same for all numerical models. However, new mesh for the draft tube with guide vane system was created. A three-dimensional hexa-dominant structured meshing approach was considered for the draft tube with guide vane system. The tool used to generate the mesh is ANSYS Mesh 17.2 [16]. Three different meshes were generated for the draft tube domain with guide vanes, see Table 1.

| Quality type                  | Mesh 1 | Mesh 2 | Mesh 3 |
|-------------------------------|--------|--------|--------|
| Elements (million)            | 2.1    | 2.6    | 3.2    |
| Max. aspect ratio             | 52.05  | 52     | 46     |
| Volume (m³)                   | 1.10   | 1.11   | 1.1    |
| Max. edge-length ratio        | 57     | 57     | 58     |
| y+                            | 54     | 40     | 36     |
| Min. orthogonality            | 75     | 70     | 68     |

The numerical analysis was performed in ANSYS CFX 17.2 [16]. Mass flow inlet and pressure outlet boundary conditions were considered for the present study. The inlet mass flow rate for PL operating condition is 139.62 kg/s, which is approximately 70% of the flow rate at BEP [1-5]. The outlet boundary condition was considered as ‘opening’ type and the gauge pressure was specified as 0 kPa. The reference absolute pressure was specified as 113.17 kPa. The rotational frequency of the runner was specified to 5.55 Hz. The present study is based on multiple domains connected using ‘domain interface’. For interface between the distributor and runner domain, the pitch change angle of 24° and 25.714° was specified, respectively. For interface between the runner and draft tube domain, the pitch change angle of 25.714° and 360° was specified, respectively. The k-ε turbulence model with the standard wall functions was considered for the present numerical study. The time-step used in the study is 2° of the runner rotation. The high-resolution advection scheme and the second-order backward Euler transient scheme were respectively used for the discretization of the nonlinear convective term and temporal term in all transport equations. Initially, a steady state computation was performed for the Francis-99 semi-model with no guide vanes was performed. The steady state results were then used as the initial value for the time-dependant analysis, with double precision for all numerical models. With reference to [7], the convergence criteria were set as RMS $10^{-6}$ and the number of coefficient loop was specified as 10.
3. Results and Discussion

3.1. Effect of guide vanes in the draft tube on pressure pulsations

As there are 3GVs in the guide vane system considered in the present study, the plunging and rotating modes for three pressure points, corresponding to 3GVs, must be considered. The plunging mode indicates the oscillation of a signal in axial direction and the rotating mode refers to the rotational behaviour of the signal. The pressure signal at each location of evenly distributed guide vanes circumferentially can be written as:

At guide vane 1: \[ P_1 = P_{\text{plung}} \cos(\omega t) + P_{\text{rot}} \cos(\omega t) \] (1)

At guide vane 2: \[ P_2 = P_{\text{plung}} \cos(\omega t) + P_{\text{rot}} \cos\left(\omega t + \frac{2\pi}{3}\right) \] (2)

At guide vane 3: \[ P_3 = P_{\text{plung}} \cos(\omega t) + P_{\text{rot}} \cos\left(\omega t + \frac{4\pi}{3}\right) \] (3)

By solving eqs. (1), (2) and (3), the new plunging and rotating mode for 3GV system is:

Plunging mode: \[ P_{\text{plung}} = \frac{P_1 + P_2 + P_3}{3} \] (4)

Rotating mode: \[ P_{\text{rot}} = \frac{(2P_1) - P_2 - P_3}{3} \] (5)

Figure 2 compares the fast-Fourier transform (FFT) of the plunging and rotating modes between the Francis model turbine with (mesh 1, mesh 2 and mesh 3) and without (reference model) the guide vane system. The FFT of the rotating mode of the reference model pressure amplitude shows the dominating RVR frequency at \( f/f_0 = 0.3 \). There is approximately 97% decrement in pressure amplitude of the RVR frequency in the presence of guide vane system in the draft tube, which is seen for all three meshes. This means that the guide vanes can successfully re-direct the flow in the draft tube which in turn results in
decreasing the rotating component of the pressure signal. However, the plunging mode’s amplitude decrement was relatively lower when compared to the rotating mode.

Figure 3 FFT analysis of plunging and rotating modes of draft tube with and without guide vanes for pressure points MC1 to MC6.
The plunging and rotating mode for two pressure signals (for monitor points MC1-MC6) are calculated as:

Plunging mode:

$$P_{\text{plung}} = \frac{P_1 + P_2}{2}$$

(6)

Rotating mode:

$$P_{\text{rot}} = \frac{P_1 - P_2}{2}$$

(7)

Similar trends were observed for monitor points MC1 to MC6, as shown in figure 3. The rotating mode of the reference Francis-99 turbine model with no guide vanes, from MC1 to MC6 indicates that the pressure pulsations of the RVR extends beyond the elbow of the draft tube. However, the magnitude of RVR pressure amplitude reduces downstream the draft tube. The rotating mode of the RVR completely disappears in the presence of guide vanes in the draft tube for all three meshes. There is a significant reduction in the plunging mode at \(ff_0 = 0.3\), with the guide vanes placed inside the draft tube. Considering points MC1 to MC4, a very low frequency (between \(ff_0 = 0.05\) to \(0.1\)) amplitude was observed.

The pressure recovery in the draft tube can be calculated as follows:

$$C_p = \frac{P_{\text{outlet}} - P_{\text{inlet}}}{\frac{1}{2} \rho (v_{\text{inlet}})^2}$$

(8)

In eq. (8), \(P_{\text{outlet}}\) is the area averaged outlet pressure of the draft tube (=0 kPa), \(P_{\text{inlet}}\) stands for the area averaged draft tube inlet pressure, \(\rho\) represents the density of water (=998 kg/m³) and \(v_{\text{inlet}}\) is the area averaged draft tube inlet velocity. The pressure recovery data for the reference draft tube model and draft tube with guide vanes for different meshes are presented in Table 2. The pressure recovery factor of the reference turbine model was found to be 0.458. No significant variation was observed for the pressure recovery factor of different mesh types for turbine model with guide vanes in the draft tube. There was 7.6\% increase in the pressure recovery factor for mesh 1, 8.1\% increment in mesh 2 and 8.7\% increment in case of mesh 3 compared with the reference model, respectively. This implies that the presence of guide vanes increased the pressure recovery within the draft tube, which indicates superior performance characteristics of the turbine runner.

| Model         | \(C_p\) |
|---------------|---------|
| Reference model | 0.458   |
| Mesh 1        | 0.493   |
| Mesh 2        | 0.495   |
| Mesh 3        | 0.498   |

3.2. Effect of guide vanes on tangential velocity

The tangential velocity along lines 1 and 2 of the 2-D PIV plane locations are presented in figure 4. Both line 1 and line 2 are perpendicular to the runner axis of rotation. It was found that there is slight reduction in the maximum tangential component of the flow at \(r/r_0 = \pm 0.5\) at line 1. The tangential velocity of the turbine model with guide vanes at \(r/r_0 = \pm 0.5\) was 13.5\% lower that the tangential velocity of the reference turbine model at line 1. Moreover, the flow appears to be symmetric along line 1. However, along line 2 location, the tangential velocity profile demonstrates some asymmetry. The asymmetry was
expected due to the presence of an odd number of guide vanes in the draft tube. As a result, the decrement in the tangential velocity component at line 2 ranges from 13.3% to 24.6% with the guide vanes in the draft tube.

Figure 4 Tangential velocity at line 1 and line 2.

Figure 5 Velocity streamline for Francis-99 semi-model without and with guide vane system.

The streamlines of the flow in the draft tube are presented in figure 5. Figure 5a) shows the swirling behaviour of the flow in the absence of the guide vanes in the draft tube during PL operating condition. The re-directed flow from PL operating condition to BEP is seen in figure 6b). As all meshes with the guide vane system showed similar flow characteristics, only mesh 3 which is the finest mesh results are presented. The streamlines show that the guide vanes, with minimum thickness, can channel the flow without causing severe flow separation. The tangential velocity (or velocity circumferential) contours were plotted for different planes along the draft tube (as presented in figure 6). In both cases, the tangential velocity contours of plane 1 are similar. At the centre of the plane 1, there is no swirling effect due to the presence of ‘stagnation’ zone and the swirling component increases as one moves away from the centre of the draft tube. In draft tube with three guide vane system, there is a significant reduction in the tangential velocity component of the flow in plane 2 and plane 3. This is due to the flow re-directing mechanism in the presence of guide vanes, the tangential velocity component in the flow is converted to the axial flow.
4. Conclusion and future work
Numerical study was performed on Francis model turbine from NVKS Francis-99 second workshop using ‘passage flow’ technique. The study was performed with and without the guide vane system in the draft tube. The guide vanes were included in the draft tube to re-direct the flow from lower operating conditions to BEP, to mitigate RVR. The factors considered in designing of the guide vanes were a) number of guide vanes to be placed inside the draft tube, b) location of the guide vanes, c) inlet and outlet angle of the guide vanes, d) chord-length and e) span of the guide vanes. The results indicate that the guide vanes in the draft tube can significantly suppress the RVR by 97% with almost 8.7% increment in the pressure recovery factor of draft tube. This indicates that the turbine with guide vane system in the draft tube has the potential of superior performance at lower operating conditions. The streamlines showed no significant flow separation around the guide vanes and the tangential velocity component in the flow within the draft tube showed consistent decrement along the draft tube. Further work should emphasize on investigating the guide vane system structural and fluid-structure interaction (FSI) modelling of the design for practical applications. Also, detailed study on the influence of a flexible guide vane system on turbine’s overall efficiency at various operating conditions is the future scope of the research study.

Acknowledgment
The authors would like to acknowledge the support and funding received for this study from the European Union’s Horizon 2020 research and innovation programme under grant agreement No 764011. The authors would also like to acknowledge the support provided by NVKS Francis-99 second workshop.

References
[1] Trivedi, C., Cervantes, M. J. and Dahlhang, O. G., 2016, “Experimental and numerical studies of a high-head Francis turbine: A review of the Francis-99 test case”, Energies, 9, 74; doi:10.3390/en9020074.
[2] Trivedi, C., Cervantes, M. J. and Dahlhaug, O. G., 2015, “Numerical techniques applied to hydraulic turbines: A perspective review” ASME Journal: Applied mechanics reviews, 68(1).

[3] Goyal R., Trivedi C., Gandhi B. K., Cervantes M. J., and Dahlhaug O. G., 2016, “Transient pressure measurements at part load operating condition of a high head model Francis turbine”, Sadhana (Springer), 41 (11), pp. 1311-1320.

[4] Goyal R., Bergan C., Cervantes M. J., Gandhi B. K., and Dahlhaug O. G., 2016, “Experimental investigation on a high head model Francis turbine during load rejection”, IOP Conf. Series: Earth and Environmental Sci., 49: 082004.

[5] Goyal R., Cervantes M. J., and Gandhi B. K, 2017, “Vortex rope formation in a high head model Francis turbine”, ASME Journal of Fluids Engineering, 139 (4), pp. 041102:1-14.

[6] Goyal R., Trivedi C., Gandhi B. K., and Cervantes M. J., 2017, “Numerical simulation and validation of a high head model Francis turbine at part load Operating Condition”, Institute of Engineers: India (Springer).

[7] Resiga R.S., Vu T. C., Muntean S., Ciocan G. D., and Nennemann B., “Jet control of the draft tube vortex rope in Francis turbines at partial discharge”, 29th IAHR Symposium, Yokohama, October 2008.

[8] Blommaert G. and Boyer A., “Active control of Francis turbine operating stability”, 3rd ASME/JSME Joint Fluids Engineering Conference, July 1999, San Francisco, California, USA.

[9] Fraser R., Deschénes C., Gokhman A. and Huang J., 2008, “Development of an exit stay apparatus for Francis turbine”, Hydrovision 2008, HCI publication, no. -117.

[10] Constantin Tănasă, S.M., Alin Ilie Bosioc, Romeo Susan-Resiga, Tiberiu Ciocan, Influence of the air admission on the unsteady pressure field in a decelerated swirling flow. UPB Scientific Bulletin, Series D: Mechanical Engineering, 2016. 78(3): p. 161-170.

[11] Rudolf P., Litera J., Bolanos J.A.I. and Stefan D., 2018, “Manipulation of the swirling flow instability in hydraulic turbine diffuser by different methods of water injection”, Experimental fluid mechanics 2017, EPJ Web of Conferences 180, 02090.

[12] Junichi Kurokawa, S.I.S., Jun Matsui and Takaya Kitahora. A new passive device to suppress several instabilities in turbomachines by use of J-grooves. 1998. Osaka, Japan.

[13] Nishi M., Wang X.M., Yoshida K., Takahashi T., and Tsukamoto T., 1996, “An experimental study on fins, their role control of the draft tube surging”, Hydraulic machinery and cavitation, pp 905-914.48.

[14] Zhou X., Wu H., and Shi C., “Numerical and experimental investigation of baffles on flow instabilities in a Francis turbine draft tube under partial load condition”, 2019, Advances in Mechanical Engineering, Vol. 11(1), pp. 1-15.

[15] NVKS Francis-99 second workshop (2016), Norwegian University of Science and Technology, Trondheim, Norway.

[16] Ansys Academic Research, Release 17.2, Help System, Coupled Field Analysis Guide, ANSYS, Inc.
APPENDIX A

Table A1 Monitor points around newly designed guide vanes blade.

| Monitor points | Coordinate points |
|----------------|-------------------|
|                | x (m)  | y (m)  | z (m)  |
| J1             | 0       | 0.179864 | 0.4558 |
| J2             | 0.06152 | 0.16907 | 0.4558 |
| J3             | 0.11561 | 0.13778 | 0.4558 |
| J4             | 0.15577 | 0.08993 | 0.4558 |
| J5             | 0.17713 | 0.03123 | 0.4558 |
| J6             | 0.17713 | -0.03123 | 0.4558 |
| J7             | 0.15577 | -0.08993 | 0.4558 |
| J8             | 0.11561 | -0.13778 | 0.4558 |
| J9             | 0.06152 | -0.16902 | 0.4558 |
| J10            | 0       | -0.17986 | 0.4558 |
| J11            | -0.06152 | -0.16902 | 0.4558 |
| J12            | -0.11561 | -0.13778 | 0.4558 |
| J13            | -0.15577 | -0.08993 | 0.4558 |
| J14            | -0.17713 | -0.03123 | 0.4558 |
| J15            | -0.17713 | 0.03123 | 0.4558 |
| J16            | -0.15577 | 0.08993 | 0.4558 |
| J17            | -0.11562 | 0.13778 | 0.4558 |
| J18            | -0.06152 | 0.16902 | 0.4558 |

Table A2 Monitor points created along draft tube.

| Monitor points | Coordinate points |
|----------------|-------------------|
|                | x (m)  | y (m)  | z (m)  |
| MC1            | 0       | 0.21   | -1     |
| MC2            | 0       | -0.21  | -1     |
| MC3            | 0.75    | 0.27   | -1.5   |
| MC4            | 0.75    | -0.27  | -1.5   |
| MC5            | 1.7     | 0.29   | -1.35  |
| MC6            | 1.7     | -0.29  | -1.35  |