Vortex rope mitigation with azimuthal perturbations: A numerical study

To cite this article: H Holmström et al 2021 IOP Conf. Ser.: Earth Environ. Sci. 774 012144

You may also like
- Extending Blade-Element Model to Contra-Rotating Configuration
  Ohad Gur
- Structural strength of iso-polyhedral beryllium alloy rotating mirror for ultra-high-speed camera
  Chunbo Li, Yinchen Wen, Shoujun Chen et al.
- The proper motion of sunspots umbra in the rising phase of Cycle 24
  K Permata and D Herdiwijaya
Vortex rope mitigation with azimuthal perturbations: A numerical study

H Holmström¹, J Sundström¹, M J Cervantes¹

¹ Luleå University of Technology, Luleå, Sweden
henrik.holmstrom@ltu.se

Abstract. A novel method to mitigate the rotating vortex rope is investigated numerically on a propeller turbine using ANSYS CFX. Pulsating momentum is injected in a horizontal plane in the diffuser cone from four evenly spaced jets. Three mitigation strategies are tested: M1 in which the momentum is injected perpendicular to the axial flow direction, M2, which exhibit a 12 degree angle against the tangential velocity in the diffuser cone, and finally M3, which exhibit the same horizontal angle as M2 but at a 15 % higher flow rate. It is shown that mitigation attempts M1, M2 and M3 decrease the amplitude of the rotating mode by 51%, 96% and 97%, respectively. The amplitude of the plunging mode, on the other hand, increase for all mitigation attempts. However, the amplitude of the plunging mode of the unperturbed RVR is an order of magnitude smaller than the rotating mode, and thus, the overall amplitude of the pressure fluctuations in the diffuser decreases significantly. The more efficient mitigation using attempt M2 and M3 are explained using velocity contour in the diffuser cone, which show that the RVR is significantly reduced downstream of the injection plane in between injections, which is not the case for attempt M1.

1. Introduction

Increased demand for renewable energy resources requires an increased flexibility of the hydroelectric. However, operating away from the designed operating point decreases the overall efficiency and potentially damage the turbine [1]. Amiri et al. [2] investigated experimentally the hydraulic loads on the runner of a Kaplan turbine on a propeller curve. The rotating vortex rope (RVR) in the draft tube produced pressure waves throughout the entire conduit, as well as bending moment on the runner blades at part load operations. Therefore, implementing solutions to mitigate the adverse effect of vortex breakdown is becoming increasingly relevant as the hydraulic turbines mantles the role of power regulator on the power distribution grid.

Several methods aiming at mitigating the vortex formation in the draft tube have previously been investigated with the most common being air injection. Air have been injected into the draft tube from various positions on the runner with some common places being the runner crown and the bottom of the runner cone [3] [4]. Unterluggauer et al. [5] investigated the effect of injecting air upstream of the runner with results showing not only a reduction of the vortex rope but also an increase to the general fatigue life of the runner.

Water injections has also been investigated as a method for RVR mitigation. Susan-Resiga et al. [6] investigated ways to mitigate the vortex rope in the draft tube by water jet injection from the runner cone. A swirl generator setup to mimic a realistic RVR in a hydraulic turbine was used. Bosioc et al. [7] investigated experimentally the jet discharge required to mitigate the vortex rope in the draft tube using the same swirl generator as [6]. A jet discharge of 11.5 % of the total flow was found necessary to completely mitigate the RVR. Tănăsă et al. [8] introduced a flow feedback method to the setup of [6] in which the jet discharge was recirculated using twin spiral cases in the draft tube directing the flow through return pipes to the tubular shaft back to the injection inlet at the runner cone. Javadi and Nilsson [9] performed numerical simulations on the experimental setup of [6] and investigated the effect of continuous injection from various slots on the runner cone. It was concluded that the jet angle and position along with the momentum flux was critical factors for effective vortex rope mitigation. Focus
of aforementioned papers have been to mitigate the vortex rope by injecting momentum along the vertical axis into the quasi-stagnant region. Although it has proven to be an effective method for RVR mitigation, it still requires noticeable jet discharge to do so. However, inducing momentum horizontally into a local area of the quasi-stagnant region may be an alternative as the local mitigation of RVR may hinder further development downstream of the mitigation point. This is, to the best of the authors’ knowledge, a new method of RVR mitigation. This paper aims to further contribute to the knowledge regarding RVR mitigation by numerically investigating the effects of pulsating jet injection azimuthally to the axial direction near the draft tube inlet.

2. Numerical model

2.1 Down-scaled turbine and operating properties.

The numerical model aims to replicate the flow in a down-scale propeller turbine located at Luleå University of Technology. The down-scaled turbine model includes six guide vanes, a four-bladed runner with a diameter of 0.1 m, a conical diffuser and an elbow bend, where the elbow bend serves to enhance the plunging effects of the RVR. At part load, the down-scaled turbine operates with a rotational speed of 1232 rpm, a guide vane angle of 15 degrees and a flow rate of 4 l/s.

2.2 Numerical configuration

To reduce the computational cost, the numerical model includes an excerpt of the down-scaled turbine including one runner passage, the conical diffuser and the elbow bend, see figure 1. This truncation of the domain is believed not to affect the results appreciably since Pasche et al. [10] showed that the flow phenomena in the draft tube could be captured well with a numerical model including only the complete runner and draft tube. Although not presented here, additional simulations performed on the complete runner geometry and a single runner passage showed that the RVR frequency was not affected by reducing the runner geometry; hence, it was deemed sufficient to perform simulations using a single runner blade only.

A time step corresponding to 91 degrees of runner rotation is used. This time step is fairly large compared to conventional hydraulic turbine simulations in the range of 1-5 degrees of runner rotation. This is not expected to affect the frequency of the RVR since, Iovănel et al. [11] showed that the RVR frequency of a single regulated Kaplan turbine model was largely independent of the size of the time step. The amplitude of the pressure pulsations was, however, shown to depend on the time step size. Specifically, large time steps tended to underpredict the pressure fluctuations. The main objective of the present paper is to investigate how the relative amplitude of RVR induced pressure fluctuations are affected by azimuthal perturbations. As such, any potential offset of the pressure amplitudes prior to the mitigation are not of prime importance as long as it can be shown that the pressure amplitude is reduced following the jet injections; hence, a relatively large time step was chosen to speed up the numerical simulations.
Figure 1. Numerical setup consisting of runner, conical diffuser and an elbow bend. Monitoring point 1 and 2 are located on the same horizontal level downstream of the injection inlets.

The commercial software ANSYS CFX v.19.2 was used to perform the simulations. A blend of both first order UPWIND and a second order scheme was used to solve the advection term. The temporal derivatives are discretized using second order backward Euler method. Reynold stress tensor was closed using the two-equation eddy-viscosity shear stress transport (SST) turbulence model. Inlet boundary condition uses angled mass flux set to match the expected incoming flow direction from the guide vanes operating at part load. Outlet boundary type is set to opening to enable flow in both directions. Interface between the rotating runner passage and stationary diffuser has a transient rotor stator frame change applied with a pitch. No slip condition and smooth wall are applied to all walls in the setup as well as the runner hub and shroud. Diffuser and elbow bend are connected through a general interface connection. A 3D hexahedral mesh is created using ICEM CFD for diffuser and elbow bend while a 3D hexahedral runner mesh is generated using ANSYS TURBOGRID.

Three different mesh sizes were investigated to ensure grid independence. Table 1 presents the mesh size for each case and value of the RVR frequency. As all three meshes yield similar value of the RVR frequency, the coarsest grid was selected.

Table 1. List of three different grid sizes used to investigate grid independence.

| Domain            | Element size N1 | Element size N2 | Element size N3 |
|-------------------|-----------------|-----------------|-----------------|
| Runner passage    | 306 000         | 565 000         | 1 060 000       |
| Conical diffuser  | 861 000         | 2 530 000       | 7 650 000       |
| Elbow bend        | 20 000          | 110 000         | 431 000         |
| Total number of elements | 1 187 000   | 3 205 000       | 9 141 000       |
| Normalized RVR frequency [-] | 0.2086     | 0.1932          | 0.1932          |
2.3 Water jet injection method

Four holes are evenly distributed circumferentially around a horizontal plane 0.045 m below the bottom of the runner cone as shown in figure 1. Pulsating sinusoidal water jets are injected with a 180-degree phase shift as presented in figure 2, i.e., opposite injections are in phase. All injection inlets have a normalized jet diameter equal to 0.06 of the runner diameter. The idea is to induce enough momentum to disrupt the RVR formation. As the injection is not continuous, the frequency of the injection should be set to avoid RVR redeveloping between injections. The injected flow can be characterized as the ratio between the injected and the operating flow.

\[ Q_r = \frac{Q_{jet}}{Q_{tot}} \]  

(1)

Where \( Q_{jet} \) is quantified by the jet average velocity and inlet area.

![Injection pattern](image)

**Figure 2.** Injection pattern. Two 180-degree phase shifted injection occur simultaneously as two 180-degree phase shifted ejections.

2.4 Data reduction

Two monitoring points located opposite each other on the diffuser wall 0.169 m below the bottom of the runner cone were used to record the pressure, see figure 1. The pressure signals obtained are used to decompose the RVR into a plunging (synchronous) and rotating (asynchronous) mode according to formulation introduced by Bosioc [7].

\[ P_{plung} = \frac{p_1 + p_2}{2} \]  

(2)

\[ P_{rot} = \frac{p_1 - p_2}{2} \]  

(3)

In which \( p_1 \) and \( p_2 \) denotes the pressure signals obtained from respective monitoring point. The amplitudes of the plunging and rotating components are used to characterize the RVR, and to quantify the effectiveness of the mitigation. For each mitigation attempt, as well as for the unperturbed RVR, a total of 20 RVR periods were simulated.
2.5 Mitigation attempts

Three mitigation attempts M1, M2 and M3 are investigated. Each simulation start from the same initial condition of a steady state RVR. The parameters characterizing the mitigation attempts such as jet velocity, jet diameter and injection frequency are normalized using the average flow velocity through the runner, the runner diameter and the runner frequency, respectively. The jet diameter and injection frequency are kept constant for all mitigation attempts. For attempt M1, the flow is injected perpendicular to the axial flow direction at a frequency set to match the frequency of the unperturbed RVR. M2 has a similar parameter setting as M1 but introduces a 12-degree horizontal angle against the tangential direction of the flow. Attempt M3 is similar to M2, except that the jet velocity is larger. Table 2 summarizes the mitigation attempts.

Table 2. List of parameters for respective mitigation attempt.

| Parameters                              | M1     | M2     | M3     |
|-----------------------------------------|--------|--------|--------|
| Normalized Injection Frequency [-]      | 0.2086 | 0.2086 | 0.2086 |
| Counter-rotating horizontal angle       | 0      | 12     | 12     |
| Flow rate ratio, Qr [%]                 | 4.85   | 4.96   | 5.73   |
| Normalized Injection velocity [-]       | 6.7    | 6.9    | 8.0    |
| Normalized jet diameter [-]             | 0.06   | 0.06   | 0.06   |
| Injection phase shift                   | 180-degrees | 180-degrees | 180-degrees |

3. Result

Pressure monitored over time at monitor point 1 and 2 before and after activating respective jet perturbation is presented in figure 3. Each mitigation attempt is activated after 1000 timesteps. All three mitigation attempts reduce the pressure fluctuation captured by the monitor points by 53%, 79%, 83% for M1, M2 and M3, respectively. Mitigation attempt M1 in figure 3 (a) reaches steady state behaviour approximately 100 iterations after injecting the perturbation, while it takes less than approximately 80 iterations for both M2, figure 3 (b), and M3, figure 3 (c). Both M2 and M3 aligns the pressure fluctuations as they reach steady state behaviour.
Figure 3. Pressure fluctuation at monitor point 1 and 2 over time for M1(a), M2(b) and M3(c). The jet perturbations are injected from timestep 1000.

Frequency spectra of the RVR plunging and rotating components for each mitigation attempt and the corresponding unperturbed RVR are presented in figure 4 and 5, respectively. For the unperturbed RVR, the rotating mode dominates the pressure fluctuations, being an order of magnitude larger than the plunging component. Mitigation attempts M1, M2 and M3 increases the amplitude of the plunging component by 90%, 290% and 230%, respectively. The overall noise level in the plunging component was reduced for M2 and M3 while M1 experienced an overall increase compared with the unperturbed RVR. The rotating component was reduced by 56%, 97% and 98% for M1, M2 and M3, respectively. Thus, the overall pressure amplitude due to the RVR decreases significantly using mitigation attempt M2 and M3.

Figure 4. Frequency spectra of RVR plunging mode in the diffuser with unperturbed RVR, M1, M2 and M3. The frequency is normalized with the runner frequency.
**Figure 5.** Frequency spectra of RVR rotating mode in the diffuser with unperturbed RVR, M1, M2 and M3. The frequency is normalized with the runner frequency.

Velocity contours parallel and normal to the plane of injection at a maximum flow injection for each mitigation cases along with the unperturbed one are presented in figure 6. The injected jets are seen to be influenced by the flow in the diffuser. Specifically, figure 6 (a) shows that perpendicular injection fails to reach the low velocity region in the centre of the diffuser due to the combined influence of the axial and tangential velocity of the flow exiting the runner. However, the jet injections decrease the width of the low velocity region, reducing the rotating mode amplitude. For attempt M2 with a horizontal angle against the flow, the jets reach further into the low velocity region, creating a smaller width of the low velocity region downstream of the jet injection seen in the vertical contour; see figure 6 (b). For attempt M3, figure 6 (c), the low velocity region is separated into two parts in the horizontal plane. The vertical plane indicate that the injected perturbation momentarily hinders the low velocity region from developing downstream of the injection inlets, thus decreasing the low velocity region. Figure 6 (d) is included as a reference case of unperturbed RVR.
4. Discussion & Conclusions

A numerical investigation of azimuthal injection to mitigate the vortex rope occurring at part load conditions in hydraulic turbines has been presented. The injection inlets are located close to the diffuser inlet and injects momentum with a pulsating sinusoidal signal with a phase shift of 180 degree between each injection inlet.

The results show that the method reduces the pressure amplitudes resulting from the RVR downstream of the injection position, using approximately 5-6% of the turbine flow rate. However, as the vortex rope does not disappear upstream the plane of injection, the injection plane should be placed on the same horizontal level as the bottom of the runner cone.

It is shown that the momentum should be injected with a counter-rotating angle to the tangential velocity of the flow in the diffuser, to increase the penetration length of the jets in the plane of injection, thus inhibiting the redevelopment of the RVR between injections. Two angled mitigation attempts were investigated and these yielded the largest reduction of the amplitude of the RVR rotating mode; however, they also yielded a noticeable increase of the plunging mode of the RVR. Nonetheless, an overall reduction of the RVR induced pressure fluctuations downstream of the jet injection plane is achieved.

Further investigations of how each injection parameter influences the RVR mitigation as well as investigating how the jet perturbation affects the pressure pulsations thorough the turbine are required to find the optimal parameter space for RVR mitigation.

Figure 6. Velocity contours presenting the jet protrusion for mitigation M1 (a), M2 (b), M3 (c), and without jet perturbation activated (d).
Acknowledgement
This project has received funding from the European Union's Horizon 2020 research and innovation programme under grant agreement No 814958.

References

[1] Amiri, K, Mulu, B., Cervantes, M.J., Raisee, M., 2016, “Effects of load variation on a Kaplan turbine runner” International Journal of Fluid Machinery and Systems, Vol. 9, No.2, pp. 182-193.

[2] Amiri, K., Cervantes, M.J., Mulu, B., 2015,” Experimental investigation of the hydraulic loads on the runner of a Kaplan turbine model and the corresponding prototype” Journal of Hydraulic Research, Vol. 53, No. 4, pp. 452-465.

[3] Chirkov, D.V., Shcherbakov, P.K., Cherny, S.G., Skorospelov, V.A., Turuk, P.A., 2017,” Numerical investigation of the air injection effect on the cavitating flow in Francis hydro turbine” Thermophysics and Aeromechanics, Vol. 24, No. 5, pp.691-703.

[4] Huang, R., Yu, A., Luo, X., Ji, B., & Xu, H. (2014, August). Numerical simulation of pressure vibrations in a Francis turbine draft tube with air admission. In Fluids Engineering Division Summer Meeting (Vol. 46223, p. V01BT10A027). American Society of Mechanical Engineers.

[5] Unterluggauer, J., Maly, A., & Doujak, E. (2019). Investigation on the impact of air admission in a prototype francis turbine at low-load operation. Energies, 12(15), 2893.

[6] Susan-Resiga, R., Vu, T. C., Muntean, S., Ciocan, G. D., & Nennemann, B. (2006). Jet control of the draft tube vortex rope in Francis turbines at partial discharge. In Proceedings of the 23rd IAHR Symposium (Vol. 192, pp. 1-14).

[7] Bosioc, A. I., Susan-Resiga, R., Muntean, S., & Tanasa, C. (2012). Unsteady pressure analysis of a swirling flow with vortex rope and axial water injection in a discharge cone. Journal of fluids engineering, 134(8).

[8] Tănăsă, C., Susan-Resiga, R., Muntean, S., & Bosioc, A. I. (2013). Flow-feedback method for mitigating the vortex rope in decelerated swirling flows. Journal of fluids engineering, 135(6).

[9] Javadi, A., & Nilsson, H. (2017). Active flow control of the vortex rope and pressure pulsations in a swirl generator. Engineering Applications of Computational Fluid Mechanics, 11(1), 30-41.

[10] Pasche, S., Avellan, F., & Gallaire, F. (2017). Part load vortex rope as a global unstable mode. Journal of Fluids Engineering, 139(5).

[11] Iovânel, R. G., Bucur, D. M., Dunca, G., & Cervantes, M. J. (2019, March). Numerical analysis of a Kaplan turbine model during transient operation. In IOP Conference Series: Earth and Environmental Science (Vol. 240, No. 2, p. 022046). IOP Publishing.