Influence of the runner cone design on the pressure fluctuations in the draft tube of a low head Francis turbine

Subodh Khullar¹, *, Sandeep Kumar¹, Krishna M Singh¹, Michel J Cervantes² and Bhupendra K Gandhi¹
¹Department of Mechanical & Industrial Engineering, IIT Roorkee, India
²Division of Fluid and Experimental Mechanics, Department of Engineering Sciences and Mathematics, LTU, Sweden
*skhullar@me.iitr.ac.in

Abstract. This paper presents a preliminary numerical investigation on the design of the runner cone for a small turbine test rig under development at IIT Roorkee, India. The test rig is designed to study the formation and mitigation of rotating vortex rope (RVR) in the draft tube of a low head Francis turbine. The original runner cone design needs to be modified to include the provision for axial jet injection. Three different configurations, namely, a truncated cone and two original length cones with different nozzle positions, have been compared with the original cone to choose a suitable one for further investigations. The entire three-dimensional flow passage of the turbine unit has been simulated using the shear stress transport turbulence model after carrying out a mesh independence test on the original design. The proposed designs are compared at three different operating points. The amplitude and frequency of the pressure pulsations in the draft tube have been analyzed using the fast Fourier transform (FFT) method.

1. Introduction
Hydropower plays a significant role in providing stability to the power grid. The global demand for energy is being met by continual as well as intermittent sources of energy. Thermal, nuclear, and hydropower plants fall in the category of continual sources of energy, while solar energy, wind energy, tidal energy come under intermittent sources of energy. To meet the real-time demand, the grid network needs to be flexible and capable of handling the inputs from intermittent sources of energy. Hydropower plants, because of their capability of speedy changeovers, are being widely used to provide this flexibility and stability to the grid. Thus, modern hydraulic machines are increasingly subjected to off-design operations, startup and shutdown sequences, and quick set point changes [1].

However, the flow field inside the turbine is severely affected when operated at off-design conditions. Francis turbines, which make up around 65% of the turbines installed, are worst affected as there is larger residual swirl at the blade exit because of their fixed pitch blades. This residual swirl, when ingested by a draft tube, results in an abrupt decrease in its performance [2]. The overall efficiency of the turbine unit drops, and undesirable flow characteristics such as the formation of rotating vortex rope (RVR), pressure fluctuations, power swings are observed [3]. Over the years, various methodologies have been proposed to either mitigate the vortex rope or to reduce its adverse effects on turbine operation. These methodologies include design modifications [4-6] as well as air injection in the draft tube [7-9]. It was found that the deficit in axial velocity and specific energy at the center of the draft tube is responsible for the development of vortex rope [10]. It led to the idea of using a water jet injection at the axis of draft tube to reduce or mitigate the pressure fluctuations associated with the
vortex breakdown and increase the momentum in the central region [11]. The jet injection is reported more effective than geometrical modifications, which are usually based on hit and trial [12].

A small turbine test rig is under development at IIT Roorkee with the aim to study the flow field during RVR formation and various methods to mitigate it. The turbine unit is a scaled-down model of a low head Francis turbine designed by BHEL, India. The cone profile of the runner needs to be modified to incorporate the provisions for axial injection for air as well as water jet. This work presents the preliminary numerical simulations carried out to compare three different runner cone configurations with the original cone design. The objective of the study is to identify the most suitable design with respect to the original runner for further studies. The details of the geometry, along with the numerical model, are discussed in the next section. The cone profiles, along with measurement locations, are also presented. The last part of the paper presents the results followed by conclusions.

2. Numerical Model
The numerical model consists of a complete turbine unit, namely casing, guide vanes, runner, and draft tube, as shown in Figure 1. The computational domain is divided into three subdomains. The first subdomain is stationary and consists of the casing along with 24 stay vanes and guide vanes. The second subdomain is the rotating domain, which is composed of a runner with 14 blades. The third subdomain is the draft tube, which is also stationary. The commercial code ANSYS ICEM is used to generate the hexahedral grid of all the turbine components. The meshed components are shown in Figure 2. The solution is obtained using the commercial code ANSYS CFX, and the details of the numerical setup are given in Table 1. Pressure values are monitored at different points along the draft tube cone (Figure 3) and in the vaneless space.

![Figure 1. Numerical model of the Francis turbine](image1)

![Figure 2. Hexahedral mesh of the turbine spiral and distributor](image2)

Figure 4 shows the four cone profiles investigated for the new design. The first one (Figure 4(a)) is the original cone to which the other profiles were compared. Figure 4(b) shows the truncated cone (Case 1) in which the profile of the cone is identical to the original, but the length has been truncated to introduce the jet. In Figure 4(c) and 4(d), the length of the cone has been kept the same as the original, but the profile has been changed to accommodate the nozzle. However, the position of the nozzle is different in the two designs. In Case 2, the nozzle is at the bottom of the cone, while in Case 3, the nozzle is little upstream of the bottom to provide a stabilizing length for the jet.
Three grids of different sizes ($M1$-$M3$) were prepared to carry out the mesh independency test. The grid independency test was carried out on the original cone at 30° guide vane opening (GVO). The test was performed using the Grid Convergence Index (GCI) method [13] to evaluate numerical uncertainties. M3 was the coarsest mesh with 5.40 million elements, while M1 was the finest with 35.29 million elements. Table 2 lists the parameters computed to perform the grid independency test. The local pressure monitor points are VL01 in the vaneless space, and DT11, DT21, DT31 in the draft tube cone. Torque is the globally measured value on the turbine blades along the rotation axis.

The maximum estimated numerical uncertainties for the pressure points are 0.143% and 0.157% for fine ($GCI_{fine}$) and medium grids ($GCI_{med}$), respectively, at location DT31. The uncertainties shown by the medium density grid for torque measurement were lower (0.418%) than that predicted by the fine grid (0.974%). Therefore, the converged solutions with the medium grid are further used to evaluate the results.
Figure 4. Cone profile designs

Table 2. GCI parameters and uncertainties in the numerical solution

| Parameter       | VL01 (kPa) | DT11 (kPa) | DT21 (kPa) | DT31 (kPa) | Torque (Nm) |
|-----------------|------------|------------|------------|------------|-------------|
| $r_{21}$        | 1.326      | 1.326      | 1.326      | 1.326      | 1.326       |
| $r_{32}$        | 1.410      | 1.410      | 1.410      | 1.410      | 1.410       |
| $\varphi_1$    | 102.814    | 96.910     | 97.444     | 98.108     | 5.893       |
| $\varphi_2$    | 102.850    | 96.892     | 97.450     | 98.097     | 5.848       |
| $\varphi_3$    | 102.975    | 96.882     | 97.445     | 98.083     | 5.873       |
| $p$             | 3.276      | 3.210      | 0.749      | 0.329      | 2.393       |
| $\varphi_{ext}^{21}$ | 102.790 | 96.922     | 97.419     | 98.220     | 5.939       |
| $e_a^{21}$      | 0.036      | -0.018     | 0.006      | -0.011     | -0.044      |
| $e_{ext}^{21}$  | 0.001      | 0.000      | 0.000      | 0.001      | 0.015       |
| $GCI_{fine}^{21}$ (%) | 0.029 | 0.015      | 0.032      | 0.143      | 0.974       |
| $GCI_{med}^{32}$ (%) | 0.073 | 0.006      | 0.021      | 0.157      | 0.418       |
3. Results and Discussion

3.1. Original Runner Cone

The simulations were carried out at three operating points corresponding to the guide vane openings (GVOs) of 30°, 22°, and 16°. At GVO of 30°, the turbine operation is found to be the most efficient (ηh = 90.7 %) among all the operating points analyzed. The GVOs of 22° and 16° correspond to 77.5 % and 54 % of the flow rate at 30° GVO. Figure 5 shows the axial velocity in the measurement plane of the draft tube, as shown in Figure 3. The flow velocity is in the negative Y direction, and hence the positive velocities in Figure 5 represent the reverse flow regimes.

At 30° GVO, there is no stagnant or reverse flow region present in the draft tube. At 22° GVO, a spiral like RVR structure is visible around the draft tube axis, which further expands into a large central stagnant region at 16° GVO. To further investigate the flow field, FFT of the pressure data at local monitor points in the draft tube and vaneless space has been carried out, see Figure 6. The maximum amplitudes for each operating point are marked in Figure 6. The frequency is made non-dimensional with respect to the runner rotational frequency (fn). The pressure amplitudes (Cp) are normalized and padded around zero by subtracting the corresponding mean pressure (p) and then divided by the reference pressure corresponding to the available head (H) as:

\[ C_p = \frac{(p - \bar{p})}{\rho g H} \]

The pressure pulsations amplitudes at 22° GVO are largest among all the operating points, and the frequency of these amplitudes (~0.28 fn) lies in the range of RVR frequency reported in the literature. The amplitudes at 30° GVO are insignificant for all the draft tube monitor points. The amplitudes in the draft tube for 16° GVO are around ten times less than that at 22° GVO. Because at deep part load operation, a large stagnant region is present at the center of the draft tube instead of the vortex rope. The pulsations at this operating point are due to the inter blade vortices observed at deep part load operations. For the monitor points in the vaneless space, pulsations are also observed at a non-dimensional frequency of 14, which corresponds to the blade passing frequency of the runner.

![Figure 5](image-url)

*Figure 5. Axial velocity contours in the measurement plane for original runner cone at three GVO’s (a) 30° (b) 22° (c) 16°*
3.2. Modified Runner Cones

The simulations were then carried out at the above-mentioned GVOs for all the proposed designs under the same boundary conditions. Since the test rig is being designed to study the formation and mitigation of RVR, the results obtained at 22° GVO will be discussed in this section. Moreover, the results obtained at 30° and 16° GVOs showed negligible changes in the pressure fluctuation amplitudes with a change in runner cone design. Figure 7 shows the iso-pressure surface of 99.4 kPa in the draft tube and the runner of the turbine. Figure 8 shows axial flow velocities in the central plane of the draft tube for different designs of the runner cone.

The cone in Case 1 is truncated; consequently, the RVR has shifted upwards coiling up in the draft tube cone. This reduces the pressure recovery in the draft tube cone. Thus, this configuration results in an efficiency drop of around 1.3%, as shown in Table 3. The modifications in cases 2 and 3, as mentioned before, have the initial cone length, but the profile has been changed to accommodate the nozzle. This resulted in a similar effect to runner cone extension [6], which shifts the RVR downstream and reduces the pressure pulsations. The predicted drop in efficiency for both Case 2 and Case 3 is around 0.5%.

To further investigate the effect of runner cone modification, FFT of the pressure data at monitor points in the draft tube and vaneless space was conducted, and the results are presented in Figure 9. The pressure fluctuation amplitude is maximum for all cases at DT21, which is near the middle of the draft tube cone. Moreover, the amplitude of the pulsations is lowest for Case 3. Based on these preliminary simulations, it can be concluded that the runner cone in Case 3 performs better than the other two cones.
Figure 7. Iso-pressure surface (at 99.4 kPa) in the draft tube for different runner cones (a) Original Cone (b) Case 1 (c) Case 2 (d) Case 3

Figure 8. Axial velocity contours in the measurement plane for different runner cones at 22° GVO (a) Case 1 (b) Case 2 (c) Case 3
Table 3: Efficiency at various operating points for different runner cone design

| Cone Angle | GVO Original | Case 1 | Case 2 | Case 3 |
|------------|-------------|--------|--------|--------|
| 16°        | 74.40%      | 74.38% | 74.41% | 74.50% |
| 22°        | 86.06%      | 84.78% | 85.62% | 85.59% |
| 30°        | 90.71%      | 90.64% | 90.73% | 90.72% |

Figure 9. FFT analysis of the pressure time signals in draft tube (DT11, DT21, and DT31) and vaneless space (VL01) for different runner cones at 22° GVO

4. Concluding remarks
Simulations have been performed on a low head small Francis turbine to choose a suitable runner cone for the provision of axial jet injection. Three different configurations, namely, a truncated cone and two original length cones with different nozzle positions, have been compared with the original cone. The results obtained indicate that the pressure pulsations are least in Case 3 (nozzle with a stabilizing length) with almost similar efficiency as that of the original runner. This runner cone configuration combines the effect of runner cone extension with axial jet injection. Moreover, the nozzle is located a little upstream of the hub tip to provide a stabilizing length for the jet.

In future work, this runner cone design will be used experimentally for further investigations to study the RVR formation at different operating points and mitigation of the pressure pulsations associated with RVR using axial jet injection.
Acknowledgment
We would like to show our gratitude to the Central Power Research Institute (CPRI), India, for providing financial assistance to this project under grant no. CPR-1133-MID. The authors are also thankful to Bharat Heavy Electrical Limited (BHEL), India, for sharing the details of turbine design.

References
[1] Trivedi C, Gandhi B K and Michel C J 2013. Effect of transients on Francis turbine runner life: A review, *J. of Hydraulic Research* **51**(2), 121-132.
[2] Susan-Resiga R, Dan Ciocan G, Anton I and Avellan F 2006. Analysis of the swirling flow downstream a Francis turbine runner, *J. of Fluids Eng.* **128**(1), 177-189.
[3] Palde U 1972. Influence of draft tube shape on surging characteristics of reaction turbines, Tech. rep. REC-ERC-72-24 Hydraulics Branch Division of General Research Engineering and Research Center US Department of the Interior, Bureau of Reclamation.
[4] Thicke R H 1981. Practical solutions for draft tube instability, *Water Power & Dam Construction* **33**(2), 31-37.
[5] Qian Z D, Li W, Huai W X and Wu Y L 2012. The effect of runner cone design on pressure oscillation characteristics in a Francis hydraulic turbine, *Proc. of the Institution of Mechanical Eng. Part A: J. of Power and Energy* **226**(1), 137-150.
[6] Gogstad, P J, 2017. Experimental investigation and mitigation of pressure pulsations in Francis turbines, Ph. D. Thesis Trondheim: Norwegian University of Science and Technology.
[7] Nicolle J and Morissette J F 2016. Simulation of air admission in a propeller hydroturbine during transient events, *IOP Conf. Series: Earth and Environmental Science* **49**(7), 072008.
[8] Susan-Resiga R, Câmpian V, Dumbravă C and Muntean A C S 2014. In-situ unsteady pressure measurements on the draft tube cone of the Francis turbine with air injection over an extended operating range, *UPB Sci. Bull.* **76**(3), 173-80.
[9] Bosioc A I, Tanasa C, Muntean S and Susan-Resiga R F 2010. Unsteady pressure measurements and numerical investigation of the jet control method in a conical diffuser with swirling flow, *IOP Conf. Series: Earth and Environmental Science* **12**(1), 012017.
[10] Nishi M 1982. Flow regimes in an elbow-type draft tube, In *Proc. 11th IAHR Symposium on Hydraulic Machinery and System, Amsterdam*, 1-13.
[11] Susan-Resiga R, Muntean S, Bernad S, Hasmaţuchi V, Anton I and Avellan F 2006. Inviscid vortex breakdown in decelerated swirling flows and flow control methods, *2nd Workshop on Vortex Dominated Flows, Romania*.
[12] Foroutan H, Yavuzkurt S 2014. Flow in the simplified draft tube of a Francis turbine operating at partial load—part II: control of the vortex rope, *J. of Applied Mechanics* **81**(6), 061011.
[13] Celik I B, Ghia U, Roache P J, Freitas C J, Coleman H, Raad P E 2008. Procedure for estimation and reporting of uncertainty due to discretization in CFD applications, *J. Fluids Eng.* **130**(7), 078001.