Study of Pressure Pulsations and Mitigation of RVR in Francis-99 HydroTurbine

HydroFlex

Background

- The formation and the nature of the Rotating Vortex Rope **(RVR)** in hydro-turbines, mainly at part load conditions, have been the focus of many research studies [1-2].
- The long ***'snake-like'*** vortex seen at the center on the draft tube has spurred many researchers to understand the flow phenomena in detail.
- Studies claim that development of 'nearly zero' velocities (axial and radial) along the draft tube centerline are responsible for the formation of RVR.
- In addition to this, a region of 'high shear' develops due to high swirling outflow and the recirculating flow around the low-velocity region.
- Previous studies were found to be beneficial for preliminary study of the RVR mitigation in the draft tube.

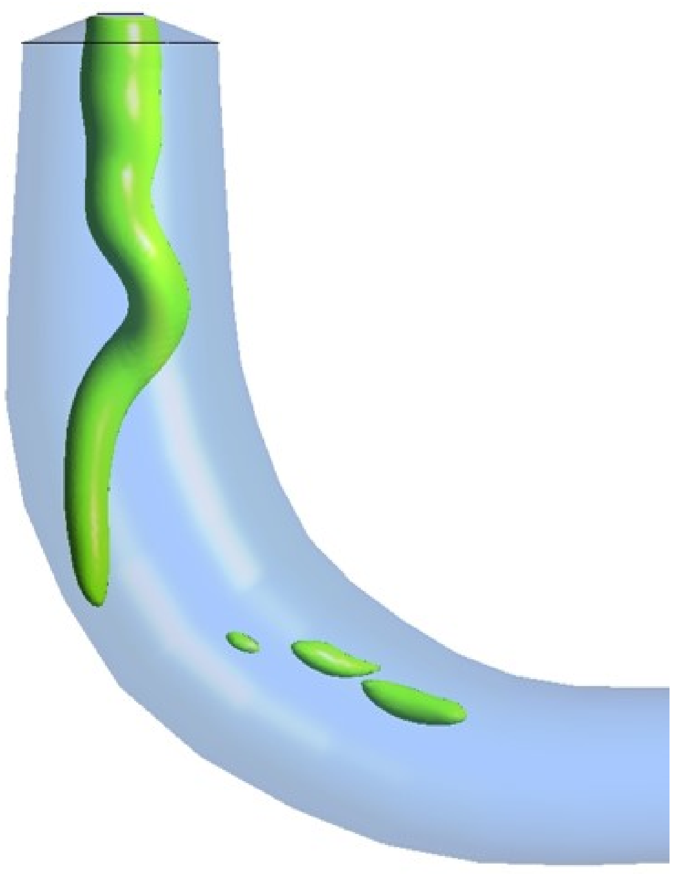

Figure 1: Rotating Vortex Rope (RVR) inside Draft tube

Research objectives

- Develop of a reduced model of the Francis-99 turbine at different operating conditions
- Mitigate the RVR by introducing guide vanes (GVs) in the draft tube based on three factors: a) number of GVs, b) blade height and c) blade span.
- Study the effects of GV in the draft tube.
- Implement and test the design on the Francis-99 model

Design and Numerical Approach

- A reduced computational model of the reference draft tube flow was developed by employing the upstream conditions at the draft tube inlet using ANSYS CFX 17.2.

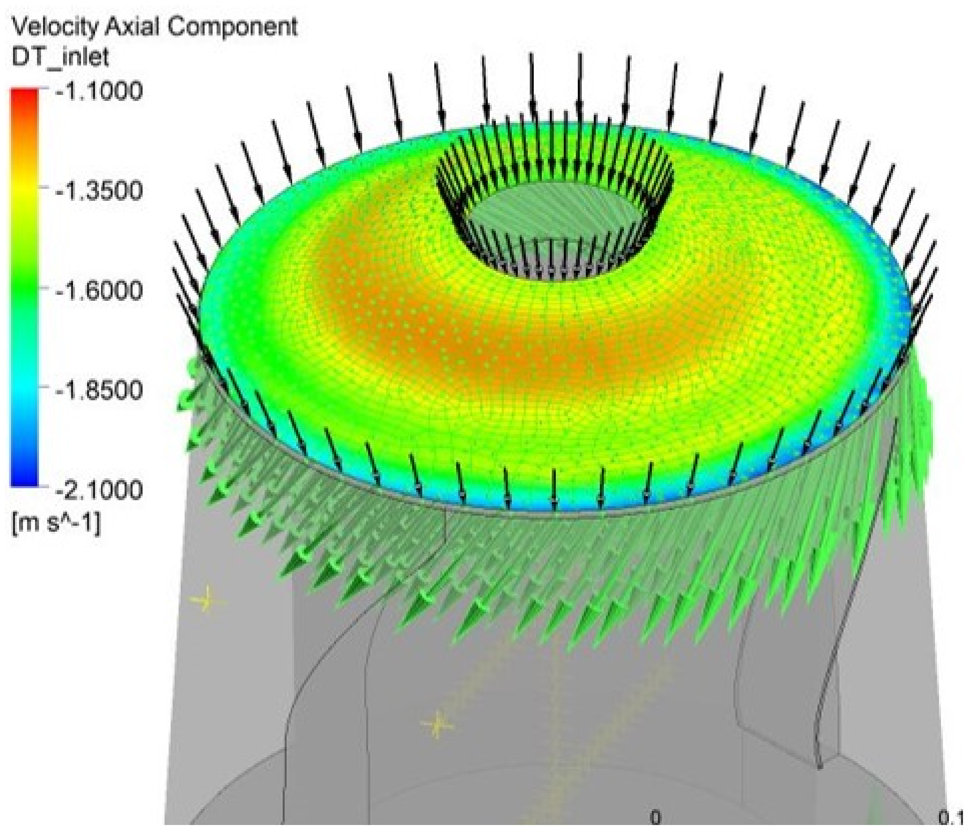

Figure 2: Reduced Draft tube model of Francis-99 turbine.

- The initial estimation of the GV inlet and outlet angle was determined using the axial and tangential velocity component at part load (PL) and best efficiency point (BEP), respectively.
- The inlet angle of the GV is derived from the flow at PL condition and the exit angle will lead the flow according to the BEP flow conditions.

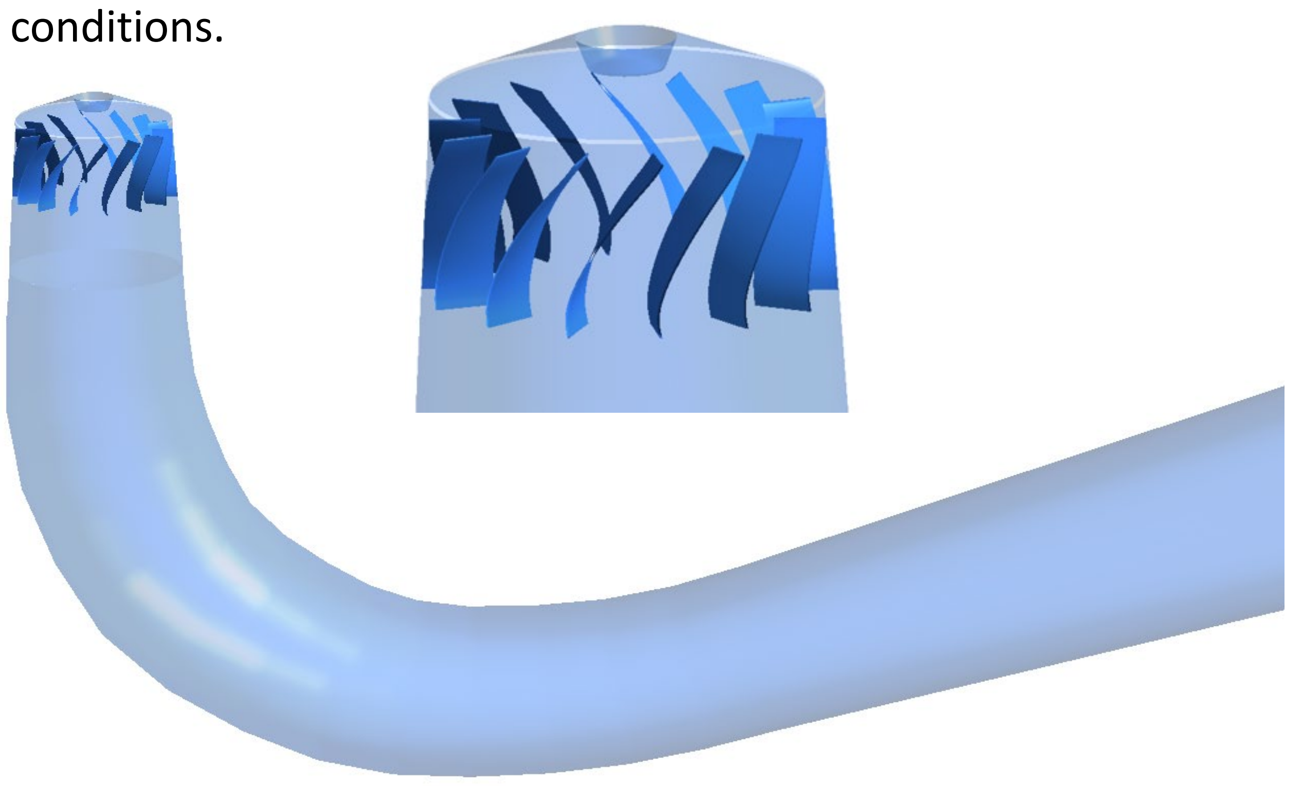

Figure 3: Preliminary design of the draft tube with GV.

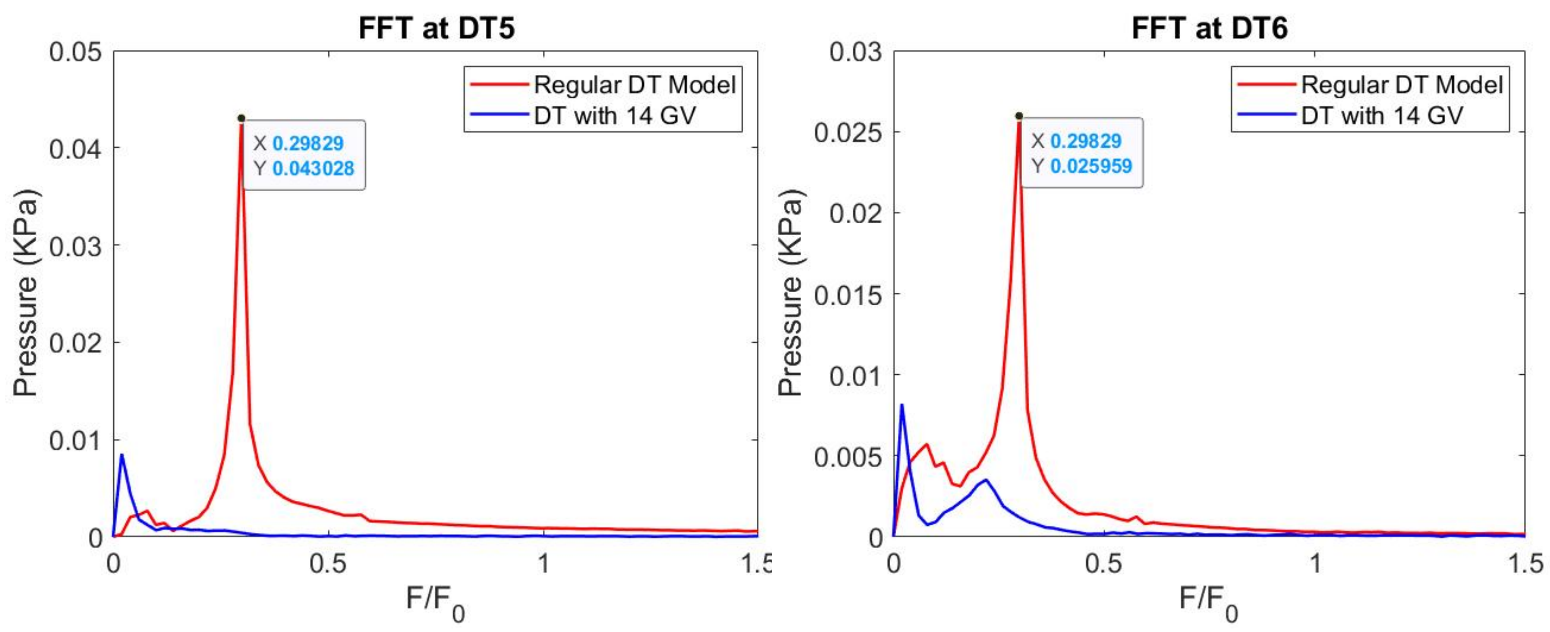

Figure 4: Comparative FFT results between the reference DT model and the raft tube with GVs.

- The number of guide vanes was kept as same as number of stay vanes and were placed right below the runner, for initial analysis.
- Hexa-dominant hybrid multizone meshing technique was employed using ANSYS Mesh for the analysis, with average mesh quality of 0.81 and K-ε turbulence model was used to run the simulations.
- The post processing of the results includes the analysis of pressure pulsations and FFT, pressure recovery (*Cp*) and the velocity component in the Draft tube.

Results and Future work

- The pressure-time and FFT analysis of the very first design indicate that by introducing GV in the draft tube, it is possible to weaken the strength of RVR.
- It was found that in case of 14GV, there was no change in Cp. Hence there was no change in the efficiency of turbine at part load.
- The influence of number of GV on the RVR is the current on-going study.
- The optimization work in the near future is aimed on resizing the GV with respect to the blade height and span, as discussed in research objectives

References

1. R. Goyal, M.J. Cervantes and B.K. Gandhi, 'Vortex rope formation in a high head model Francis turbine' (2017), *Journal of Fluids Engineering*, vol. 139/041102-1.
2. R. Goyal, C. Trivedi, B.K. Gandhi and M.J. Cervantes, 'Numerical simulation and validation of a high head model Francis turbine at part load operating condition' (2017), *Journal of Institute of Engineering, India*, DOI: 10.1007/s40032-0380-z.

Author: Jesline Joy, Michel Cervantes
Institution: Luleå Tekniska Universitet, Sweden
Date: 20/06/2019

This project has received funding from the European Union's Horizon 2020 research and innovation programme under grant agreement No 764011.