IOP Conference Series: Earth and Environmental Science

Francis turbine high load instabilities – Model test and CFD simulation

To cite this article: D Rodriguez *et al* 2019 *IOP Conf. Ser.: Earth Environ. Sci.* **240** 022053

IOP Conf. Series: Earth and Environmental Science **240** (2019) 022053 doi:10.1088/1755-1315/240/2/022053

Francis turbine high load instabilities – Model test and CFD simulation

D Rodriguez[1], A Rivetti[1], M Angulo[1], C Lucino[1], S Liscia[1]

[1]Laboratory of Hydromechanics, National University of La Plata, La Plata, Argentina

E-mail: daniel.amancio.rod@gmail.com

Abstract. When operating under high load conditions, Francis turbines tend to develop a typical central vortex located under the runner cone. Usually, this central vortex has an axisymmetric main part with helicoidal tail. Under cavitating conditions, this central vapor cavity may become unstable, generating synchronic pressure pulsations (known as self-excited oscillations) which propagate into the entire machine. The volume of the vapor cavity is a relevant feature as it influences the frequency of these pressure pulsations. Numerical flow simulations together with model test measurements and visualizations allow the characterization of the high load vortex pattern developed under different operating and Sigma plant conditions. In this work, model tests and transient two-phase CFD simulations were carried out for a medium-head Francis model scale operating under high load conditions. The vortex instability zone measured and numerical simulated on model scale is presented.

1. Introduction

The typical behavior of Francis turbines operating at high load presents an axisymmetric vortex flow which develops in the draft tube cone and extends to the elbow. Depending on the turbine specific number and runner design, the vortex develops different axisymmetric shapes. Operating at those off design conditions, Francis runners are prone to generate pressure pulsations and vibrations. In particular, the high load vortex has a self-excited nature and could produce hydraulic pressure oscillations due to resonances with the surrounding hydraulic systems as was reported at prototypes [1,2,3].

In general, model tests are performed with a homologous geometry which extends from the inlet spiral case section to the outlet draft tube section, both connected to the test rig hydraulic system. This modeled geometry differs from the prototype hydraulic system, which includes the penstock and exit channel among others. From this point of view, model scale measurements of self-excited vortex pressure fluctuations have uncertainty to be scaled up to prototype.

This work presents physical and numerical simulations in order to study the central cavity dynamic behavior for different operating points (n_{ED},Q_{ED}) corresponding to a Francis model (nq=51) with a scale factor e=7.8. The hydraulic passage and runner are homologous to the prototype which corresponds to an existing project that recently was upgraded, increasing the power output. On site prototype measurements related to high load operation are going to be performed.

Figure 1. Francis model mounted on the UNLP Test Rig –Laboratory of Hydromechanics.

The synchronic oscillating vortex measured on Model is complemented with a CFD analysis where the water vapor cavity behavior is numerical simulated through two-phase transient calculations.
Finally, a further analysis is presented related to the vapor cavity volume and its oscillation frequency [4].

2. Methodology

2.1. Physical modeling

The Francis model test was carried out on the UNLP Test Rig of the Hydraulic Laboratory following the IEC 60193 code [5]. The test head was H = 18mwc (close to Froude Number) and the runner rotational speed were around 600 rpm. The Thoma number (σ) was controlled by a low- pressure tank assembled to the draft tube outlet which is connected to a vacuum pump. Sigma values were kept constant around σ = 0.09.

The test was performed at constant speed factor n_{ED} (corresponding to maximum and minimum prototype heads) varying the discharge factor Q_{ED}. The locations of the test points relative to the speed and discharge factors corresponding to the peak efficiency point are detailed in Table 1. Also a brief summary of vortex behavior is included.

Table 1. Tested points.

Point OP#	n_{ED}/n_{EDpeak}	Q_{ED}/Q_{EDpeak}	Vortex behaviour	GVO [mm]	Vortex development
A	0.95	1.26	oscillating	36	Fully developed
B	0.95	1.22	Intermittent pulse	33	Fully developed
C	0.95	1.15	Intermittent pulse	30	weak development
D	0.95	1.13	Stable cavity	29	incipient
E	1.08	1.12	-	30	No vortex
F	1.08	1.21	Stable cavity	34	incipient

On each tested point was measured the pressure pulsations with two dynamic pressure transducers located in the draft tube cone 90° one from each other (Kistler 710A, P1 and P2). The signals were acquired during 30 seconds at 4 kHz. In order to process the acquired data, the signal was conditioned with a 100 Hz low-pass filter.

The vortex visualization was performed through the acrylic cone mounted on the upper part of the draft tube.

The maximum tested guide vane opening (GVO=36mm) correspond to a point located outside the prototype operating zone.

Although the model has the possibility to inject air through the draft tube cone fins, the present tests are performed without air injection (the prototype has no air admission through main shaft).

2.2. Numerical simulation

The numerical simulations were performed using the ANSYS CFX software which uses a RANS scheme to solve for the pressure and velocity, modeling turbulence thought an extra set of equations. The simulations were performed using the SST model, both in combination with scalable wall functions.

The multiphase configuration was defined by the native CFX cavitation model, where the inter-phase mass transfer between the liquid and vapor phases uses a simplified version of the Rayleigh-Plesset model with saturation pressure of 3500 Pa. The default CFX coefficients and constants were used. For the vapor definition parameters, it was used a calorically perfect ideal gas with density variation.

Using periodic boundary conditions, the configuration with model scale includes only one flow channel for the tandem cascade and the runner. The complete draft tube was modeled with an extension at the outlet to avoid disturbances caused by the outlet condition.

The meshes were made using ICEM CFD. The tandem cascade was meshed with Tetra/Prism elements and the runner, draft tube and extension with hexa-elements. The complete mesh includes $6.0x10^5$ nodes, of which $3.0x10^5$ nodes are used in the draft tube. The mesh and the complete configuration are shown in figure 2.

For the dynamic response analysis, the transient simulations were performed with a timestep of 0.002 s.

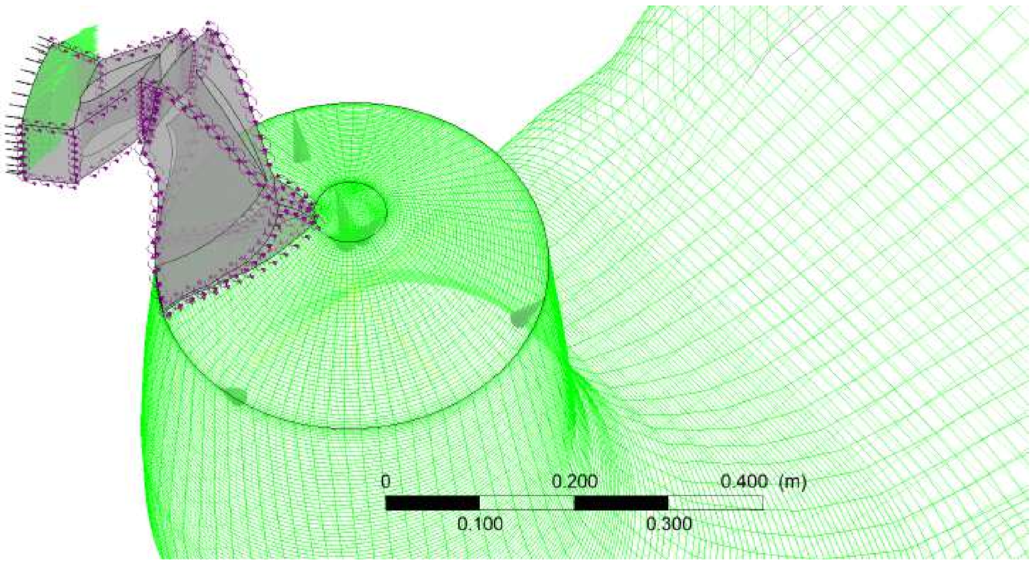

Figure 2. Domain discretization for the guide vane, runner and draft tube. Model scale.

IOP Conf. Series: Earth and Environmental Science **240** (2019) 022053 doi:10.1088/1755-1315/240/2/022053

3. Model testing and CFD results

3.1. Model Test

As Follows, a series of test points from OP#A to OP#D for constant n_{ED} corresponding to the maximum prototype head is presented. This run is performed from GVO=36 mm, decreasing the discharge factor Q_{ED} by closing the guide vane in steps and maintaining Thoma number constant.

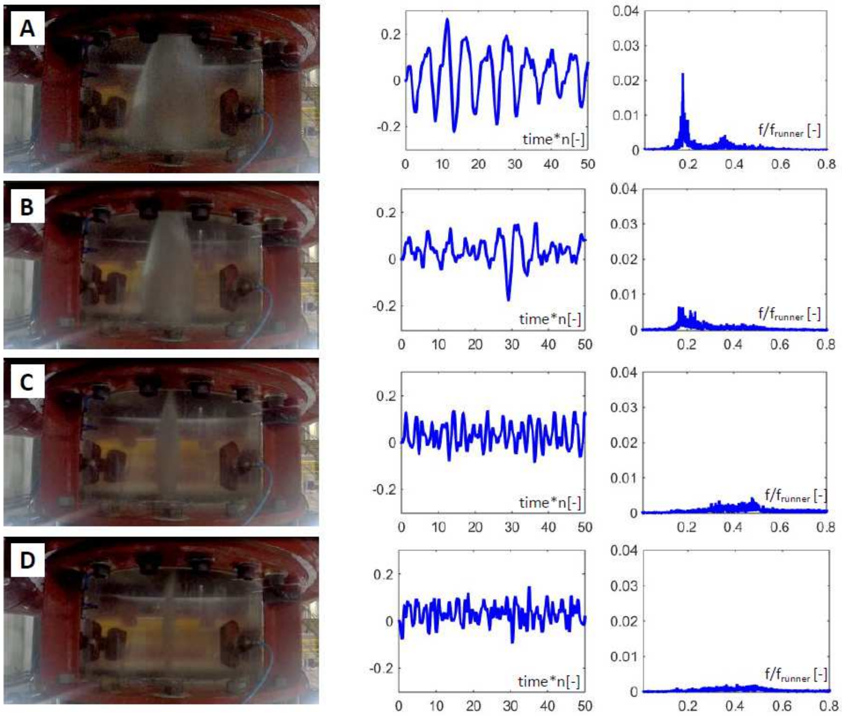

Figure 3. High load vortex visualization, time domain pressure signal and FFT frequency spectrum corresponding to n_{ED}/n_{EDpeak}=0.95 for tested points OP# A,B,C and D.

Analysing figure 3, it could be observed the high load vortex tested sequence from fully developed (OP#A) to incipient (OP#D) condition. In these cases, the flow pattern with counter rotating direction induces a low pressure zone which generates an axisymmetric water vapour cavity.

Point OP#A shows a very well-defined oscillating cavity with frequency f/f_{runner}=0.18. This oscillating behaviour is synchronic as seen in figure 4 where P1 and P2 pressure transducers time domain signals have a zero phase between them. Also, this point could be characterized as synchronic periodic pulsation due to its big water vapour cavity volume oscillation which is visualized at naked eye.

From the prototype point of view, points OP#B and OP#C have special interest. The existing turbine passed through a generator uprating and is expected to operate close to these points.

Point OP#B remains with water vapour cavity oscillation but not periodicaly, which means the synchronic pressure pulsations become intermittent. This unstable oscillation has a frequency f/f_{runner} around 0.2. Same as point OP#A, the intermittent synchronic oscillation could be observed directly.

Different behaviour is visualized on point OP#C. The intermittent cavity pulsation is almost disappearing and has a frequency f/f_{runner} spread around 0.4.

Point OP#D presents a stable small water vapour cavity which delimit the incipient curve (vortex free zone).

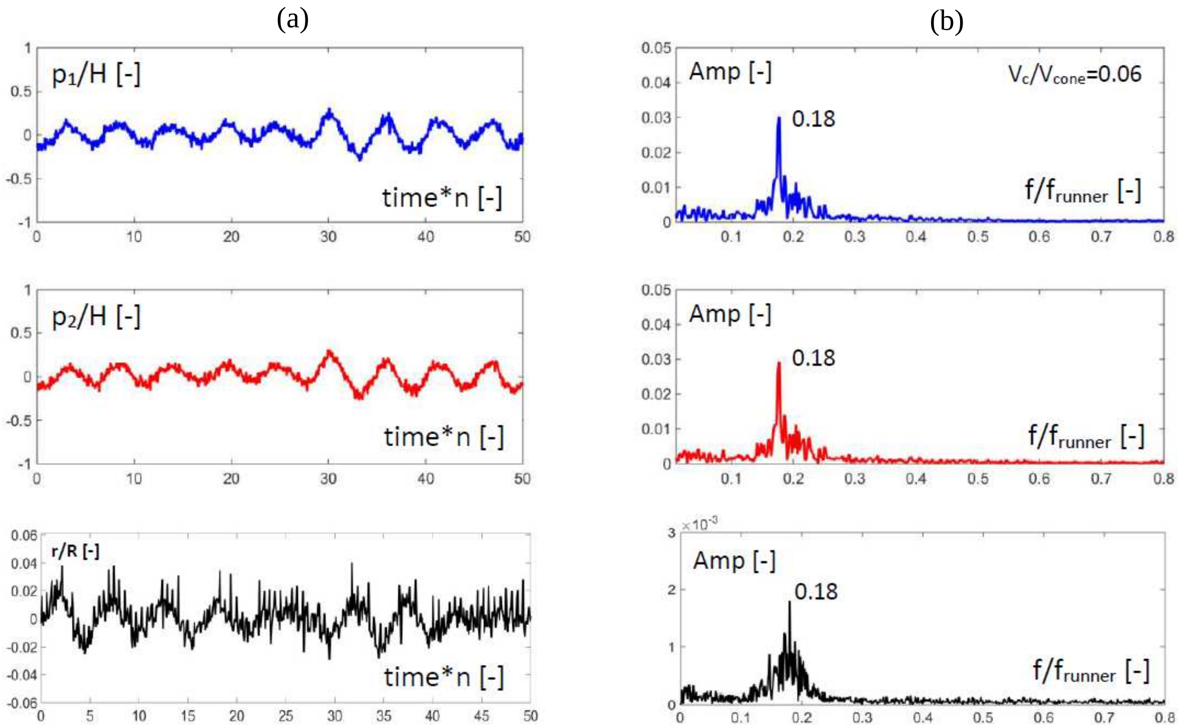

Figure 4. Pressure pulsation signals acquired from transducers P1 and P2 located in the draft tube cone and cavity volume oscillation obtained through high speed image processing. (a) Time domain signal with phase zero between them. (b) FFT frequency spectrum.

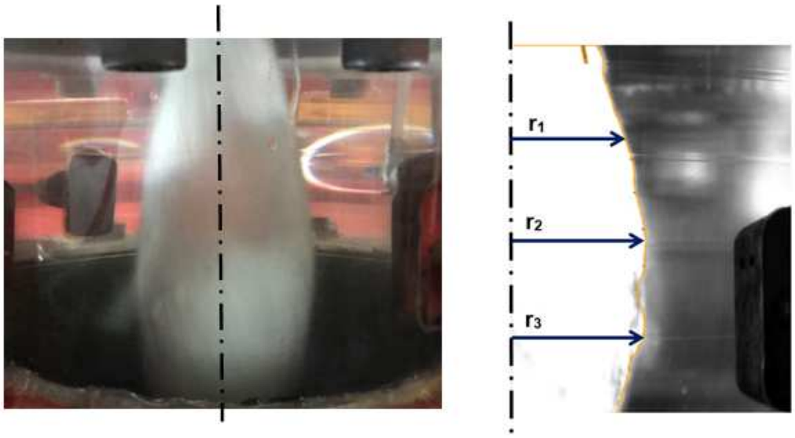

Figure 5. Image processing. Video frames taken from high speed camera (125 fps, 100 s). Vortex rope diameter as a function of time is obtained through image processing for every movie frame.

3.2. Numerical results

In order to get an additional understanding of the water vapor cavity oscillations frequency, a CFD transient 2-phase simulation was run between points OP#B and OP#C. For that purpose, the steady state simulation was used as the initial condition for the transient CFD calculations. A point at GVO=32 mm was simulated (see figure 6), obtaining the free water vapor cavity dumped oscillation due to a pressure pulse application at the extended draft tube outlet section.

IOP Conf. Series: Earth and Environmental Science **240** (2019) 022053 doi:10.1088/1755-1315/240/2/022053

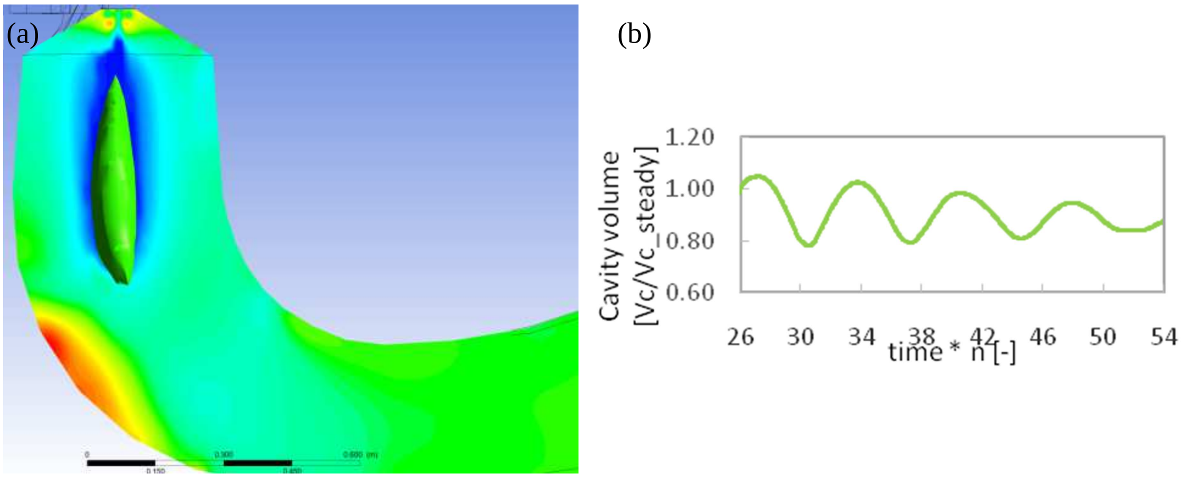

Figure 6. (a) Pressure contour on longitudinal draft tube plane and water vapour cavity (volume fraction =0.5). (b) damped cavity volume oscillations due to pressure pulse application as a function of runner rotations. f/f_{runner}=0.14, V_c/V_{cone}=0.034.

Analysis of the numerical simulated cavity volume response due to the pressure pulse application, allows us to know the natural oscillation frequency of the draft tube. Although the oscillations showed in figure 5 are not self-sustained, model test results present intermittent instabilities with synchronic pressure pulsations on OP#B. Thus, the simulated fluid domain is not capturing the self-induced instabilities; however, the oscillation frequency could be obtained. In this regard, the CFD results could help us to know the natural frequency of the draft tube for certain operating points (similar to OP#B and C) where a specific oscillating frequency is not clearly defined.

4. Discussion

The high load vortex represented by its axisymmetric cavity volume development was tested in different operating points, from incipient to fully developed condition.

The measured synchronic pressure oscillations define the natural draft tube frequency. In model operation point OP#A, the results show a peak oscillation frequency at f/f_{runner}=0.18 which could be transposed to prototype but actually is not an operating point.

Model tests corresponding to points OP#B and OP#C are close to prototype operating conditions. For these cases, the possible draft tube natural frequencies were measured and numerical simulated by capturing the self-excited axisymmetrical high load vortex oscillations. As seen in figure 3, the FFT frequency spectrums diagrams do not show a clearly defined oscillation frequency with its zero phase between transducers P1 and P2. Anyway, could be defined for OP#B and OP#C its characteristic frequencies coefficients f/f_{runner} sufficiently close (chosen between the cloud of frequencies peaks) by the pressure oscillation phase analysis.

Analysing point OP#B, it could be compared the physical model with the CFD results. This numerical simulation underestimates the frequency $f/f_{runner} = 0.14 < f/f_{runner} = 0.2$. One of the reasons for this difference is the draft tube mesh sizing definition and its corresponding water vapor cavity volume calculation. Besides, a not precise simulated absolute pressure in the draft tube cone could be another reason.

The tested and numerical simulated results could be represented by a power law regression curve. On figure 6, the frequencies and cavity volumes are normalized by the runner rotation frequency (f_{runner}) and the draft tube cone volume (V_{cone}) along the vortex development respectively.

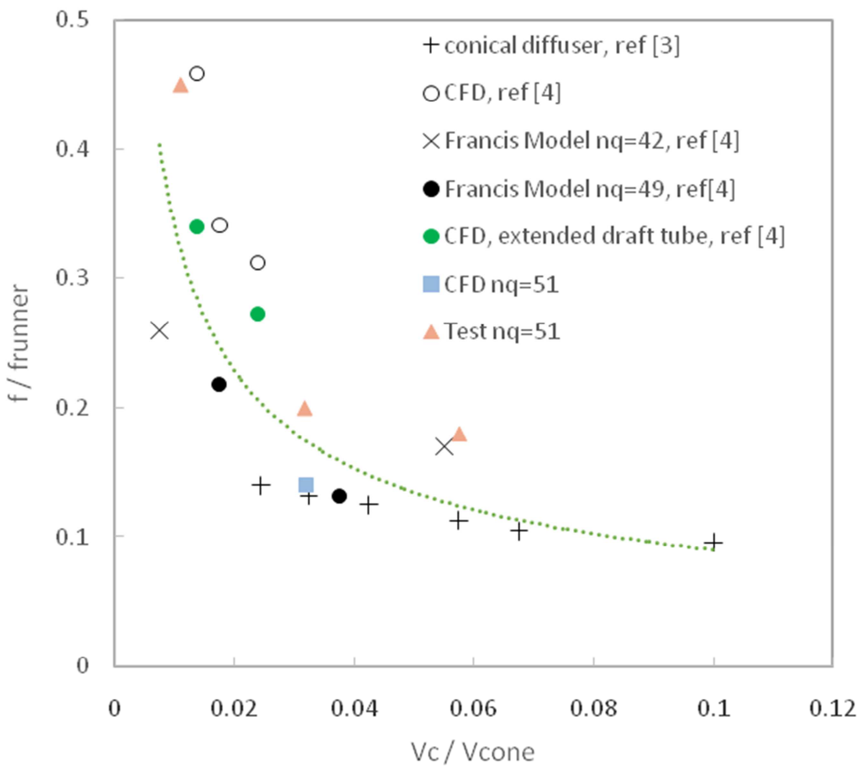

Figure 7. Central vortex oscillation frequencies vs vapor cavity volume.

In accordance with figure 7, where the oscillation frequency coefficient is a function of the cavity volume, it could help us to characterize the natural draft tube frequency according to the high load vortex development. In order to transpose those oscillations frequencies to Prototype, the frequency coefficient f/f_{runner} has to be evaluated on model taking into account the Thoma number, the Froude number and the air injection issues.

5. Conclusion

This work has studied the behavior of a Francis turbine nq=51 operating under high load conditions where an axisymmetric vortex core develops.

Model test measurements and visualizations captured the natural frequencies of the draft tube by analyzing the water vapor cavity oscillations and the pressure signal phase between two transducers located in the draft tube cone.

The 2-phase transient simulation has allowed the water vapor cavity volume frequency calculation in order to evaluate the dynamic behavior of the high load self-excited hydraulic oscillation. The CFD simulation has been correlated against physical model results describing the cavity oscillation frequency in relation with the vortex volume.

Model measured frequencies coefficient f/f_{runner} will be transposed to prototype and compared with on-site measurements.

References

[1] Arzola F, Azuaje C and Zambrano P et al. 2006 Undesired power oscillations at high load in large Francis turbines experimental study and solution *Proc. of 23rd IAHR Symp. on Hydraulic Machinery Systems (Yokohama)* 1-9
[2] Koutník J, Nicolet C, Schohl G and Avellan F 2006 Overload surge event in a pumped storage power plant Proc. of 23rd IAHR Symp. on Hydraulic Machinery Systems (Yokohama)
[3] Müller A 2014 Physical mechanisms governing self-excited pressure oscillations in Francis

turbines PhD thesis EPFL Lausanne
[4] Rodriguez D et al. 2016 High load vortex oscillations developed in Francis turbines. Proc. of 28rd IAHR Symp. on Hydraulic Machinery Systems (Grenoble)
[5] International lectrotechnical committee 1999 Hydraulic turbines, storage pumps and pump-turbines – model acceptance tests international standard IEC 60193, 2nd Edition

Acknowledgments
The authors would like to thank to Leonardo Díaz and Gonzalo Mereb for their collaboration in the execution of the experimental test and to Paula Consoli for her help with the implementation of an image processing technique for vortex rope dimension calculation.