Draft tube pressure pulsation predictions in Francis turbines with transient Computational Fluid Dynamics methodology

M Melot, B Nennemann and N Désy
Andritz Hydro Ltd., 6100 Trans Canada Hwy., H9R 1B9, QC, Canada
matthieu.melot@andritz.com

Abstract. An automatic Computational Fluid Dynamics (CFD) procedure that aims at predicting Draft Tube Pressure Pulsations (DTPP) at part load is presented. After a brief review of the physics involved, a description of the transient numerical setup is given. Next, the paper describes a post processing technique, namely the separation of pressure signals into synchronous, asynchronous and random pulsations. Combining the CFD calculation with the post-processing technique allows the quantification of the potential excitation of the mechanical system during the design phase. Consequently it provides the hydraulic designer with a tool to specifically target DTPP and thus helps in the development of more robust designs for part load operation of turbines.

1. Introduction

In today’s electricity grids, utilities want to operate their machines with the most flexibility possible. More operation time at part load is therefore expected. However when Francis turbines are operated at part load, they exhibit Draft Tube Pressure Pulsations (DTPP). Those DTPP are typically measured by means of four pressure sensors located on the draft cone wall at a given elevation. Those pulsations are known to have a major impact on the dynamic behaviour of a hydraulic Francis turbine that can lead, in the worst cases, to excessive mechanical vibrations or power swings. The pulsations are created because of the higher swirling flow that enters into the draft at operating points far from the best efficiency point, either at part load or at full load. For a manufacturer it is important to be able to understand and predict this phenomenon in order to be able to distinguish between harmful and harmless pressure pulsations under machine fatigue life considerations.

2. Physics of the part load Draft Tube Pressure Pulsation

2.1. Physical description

The part load DTPP is essentially due to a special flow instability called vortex breakdown. Researchers have studied this vortex breakdown phenomenon which is characteristic to swirling flows, for more than fifty years in laboratories [1]. Vortex breakdown can be observed in situations where swirling flows occur such as on delta plane wings, in industrial burners or on hydraulic Francis turbines. An axisymmetric swirling flow with a straight line vortex core will undergo a flow reorganisation in two distinct regions of different axial velocity. As shown in figure 1, the vortex core can take several shapes. Two of them are the corkscrew-like filament and the double helix that are typical of part load in the case of Francis turbines, with the former much more frequent than the latter.
There is also another shape where the vortex core swells into an axisymmetric stagnation bubble. This stagnation bubble is typical of the so-called full load rope which is not studied in this paper.

Figure 1. Two typical shapes of draft tube part load vortex obtained with CFD for model scale. The left figure (a) presents a corkscrew-like filament (in white) called a rope, surrounding a large zone of low-speed axial velocity (dead water core) coloured in blue. The right figure (b) shows a double helix arrangement which can happen at lower load.

Experience has shown that the main factor for triggering the vortex breakdown is the non-dimensional swirl number:

\[
m = \frac{\int_0^R C_m C_r r^2 dr}{R \int_0^R C_m^2 r dr}
\]  

(1)

This swirl number is defined as the ratio of axial flux of moment-of-momentum to the axial flux of axial momentum times the inlet radius. In the equation (1) \( C_m \) and \( C_r \) are the axial (or meridional) and tangential velocity at the inlet of the draft tube. They are presented in figure 2, where \( C \) is the absolute velocity, \( U \) is the blade velocity and \( W \) is the relative velocity. The only contributor to the axial moment-of-momentum is the tangential velocity. The swirl number is a non-dimensional number that expresses the ratio of momentum due to the tangential velocity over the momentum due to the axial velocity.

Figure 2. Runner outlet velocity triangle for three operating points, at part load (a), best efficiency point (b) and full load (c).

Because in Francis turbines the blade angle is fixed, the relative angle between \( U \) and \( W \) at the runner outlet is practically constant for every operating point. There is necessarily some tangential velocity \( C_r \) for the majority of operating points, and thus some swirl is generated. As shown in figure 2(b), close to the best efficiency point, there is very little tangential velocity because the flow rate (which is proportional to \( C_m \)) is adapted to the specific blade angle of the runner. For flow rate values below that optimal flow rate, the tangential velocity is in the same direction as the runner rotation,
resulting in a co-rotating corkscrew like vortex. On the contrary at full load the vortex is counter-rotating because the tangential velocity and the runner rotation are in opposite directions.

As suggested by [2] the flow rate is a quantity more easily accessible than the swirl number, especially in a model turbine laboratory. Because of the change of regime of vortex from the optimal flow rate, in the context of the draft tube pressure pulsations, it is sensible to characterise the operating points with the ratio $Q/Q_{op}$. It is indeed a quantity that is directly related to the swirl number so no loss of generality is incurred.

2.2. Part load vortex and cavitation

The part load vortex breakdown that develops in the draft tube is called a rope, especially when it has the shape of the corkscrew-like filament. The vortex rotation frequency is relatively stable and is usually in the range of 0.2 to 0.4 times the runner rotation speed. On a test stand, this filament is visible through a transparent draft tube cone when its core is cavitating. If the cavitation number is raised, such that the model is operated under non-cavitating condition, the rope will visually disappear. However, the draft tube pressure pulsations associated with the vortex breakdown, are still present at a level very similar to those under cavitating. Nevertheless it must be noted that under cavitating condition a resonance of the water column can occur. Cavitation introduces a local compressibility of the water column as is thoroughly explained in [2], and forms a one dimension oscillator where cavitation is directly related to the stiffness of the system and thus controls its natural frequency. The rope plays the role of an excitation force for the hydraulic system. Under unfavourable conditions the ropes’ frequency matches the natural frequency of the hydraulic system resulting in unacceptable vibrations or power swings. These occur because of significant pressure waves travelling back and forth in the hydraulic column which in turn give rise to associated flow rate variations.

3. CFD setup

3.1. Previous studies

It is known for some time now that part load draft tube vortices can be properly simulated with the help of Unsteady Reynolds-Averaged Navier-Stokes CFD (URANS CFD). For instance Stein et al. [3] have reported very good predictions of the pressure fluctuation amplitudes, within 2% accuracy, at the draft tube cone together with their frequency for single phase, i.e. non cavitating, flows. To achieve these results they used very fine meshes, up to 35 Million nodes in their simulations and applied a Reynolds Stress turbulence model. They also demonstrated that for two phase flow simulations, with the cavitating models available at the time, even finer meshes were required to capture adequately the small scale cavitation bubbles. Even on up-to-date computer clusters applying such refinements for single flow simulations, remains a prohibitive effort during the design phase. Moreover, practical experience with Reynolds stress turbulence models has shown their very high numerical sensitivity. This results in difficulties to achieve proper convergence, and consequently induces a lot of engineering time to be spent on tuning the turbulence model parameters for every new case to simulate. Other researchers have reported relatively accurate predictions of the pressure pulsations with much more modest computational resources [4][5]. The frequency predictions reported by these researchers however are only accurate to about 10-15%. For practical purposes this is usually sufficient.

A more practical and automatic CFD setup, where the design engineer can use his time in the results interpretation rather than in the setup tuning, remained to be made. The goal of such an automatic setup is to predict, in a reasonable time frame during the design phase, the level of pressure fluctuation amplitudes. An appropriate level of precision is needed to be able to assess the risk of fluctuation problems.
3.2. Setup and meshing
As can be seen in figure 3, the computational domain is divided in three parts: a single guide vane channel, the full runner and the draft tube. All meshes are generated with in-house meshing tools and are composed of hexahedrons except for the guide vanes where a hybrid mesh made of prisms and hexahedrons in the guide-vane skin region is used. The total mesh size is about 3M nodes.

![Figure 3. View of the CFD domains and meshes used](image)

A total pressure type boundary condition is imposed at the inlet and a zero static pressure is defined at the outlet. This represents with reasonable accuracy the net head and allows the flow rate to be a result of the computation. Moreover, a stage interface is defined between the guide vane and the runner and a transient rotor stator interface lies between the runner domain and the draft tube. After a steady state initialisation computation, the transient computation is launched. The first runner rotations as presented in figure 4 are discarded at post-processing to make sure that the part load vortex has enough time to develop and stabilise. The time step must be selected such that the vortex period is resolved with an adequate number of samples. The SAS turbulence model has proven to be appropriate for this type of unsteady phenomenon.

![Figure 4. CFD time signal of the four pressure monitors located on the draft tube cone wall.](image)

3.3. Automation of the procedure
This setup is to be used during the design phase. Because each operating point requires approximately 24 hours computational clock time, there is a risk to block the shared cluster resources especially when several operating points are to be evaluated in parallel. That is why a tool has been developed that launches and oversees the computations of a set of operating points in an efficient way. This tool
cooperates with the cluster queuing system to allow an automatic and temporary pause to make room for some other quicker and more urgent computations that must be done in-between. This behaviour helps to make an optimal use of the cluster during its “idle” periods and thus mitigates the detrimental effect of long computational times required for a set of DTPP operating points. Consequently, this computational effort becomes viable for a standard design procedure in which the DTPP have to be predicted.

4. Post processing and validation
Once the computation is done, it is possible to apply the same kind of advanced post processing to the CFD results of the pressure sensors at the draft tube cone as for the laboratory measurements. This is done to assess the quality of the simulation. Once validated, the CFD results can be used to get a deeper understanding of the flow behaviour within the draft tube.

4.1. Pressure fluctuations at the draft tube cone wall
The IEC code for model acceptance tests [6] advises to use four pressure sensors equidistantly located on a circumference of the draft tube cone wall to be able to measure the pressure fluctuations at high frequency (typically 2000 Hz). Those pressure sensors can also be defined as monitoring points in the CFD model. We can then compare the fluctuations and the dominant frequency associated with those fluctuations.

There are different ways to quantify the pressure fluctuations. One of the most common in the industry, as proposed by the IEC code, is the difference between the 1.5\textsuperscript{th} percentile and the 98.5\textsuperscript{th} percentile pressure values. This is equivalent to say that the pressure fluctuation is a random variable and the measured samples define the probability distribution of that random variable. One example is plotted in figure 5. For a given pressure value the probability distribution function gives the probability of the pressure fluctuation being less than or equal to that pressure value. According to IEC, a known probability of, in this case, 97\% is assumed and supposed to be representative of the characteristic, i.e. deterministic, fluctuations.

The figure 5 shows a satisfactory general agreement between the laboratory measurements and the CFD computations. The wavy aspect of the CFD results is certainly due to the shortness of the signal in comparison with the laboratory measurements: the longer the signals, the better the estimation of the true distribution functions.

For a low-medium head Francis turbine, the comparison of measured and predicted 97\% pressure fluctuations is presented in figure 6. For values of Q/Q\textsubscript{opt} between 0.8 and 1, we can observe a good agreement between the measurements and the CFD results. There is a maximum fluctuation obtained in the measurements around Q/Q\textsubscript{opt} = 0.7. This value is almost obtained with CFD but for a lower value of Q/Q\textsubscript{opt} = 0.55. While not predicting the maximum at the correct operating point, from an engineering point of view, the deciding factor is the maximum amplitude. In this respect, the CFD captures a very satisfactory value.

The presence of the experimental peak of pulsations for values of Q/Q\textsubscript{opt} = 0.7 could be due to the so-called Upper Partial-Load Vortex as studied in [7]. It usually appears on the test stand, but does not transpose to the prototype probably due to a lack of Froude similarity. The intersections of the light grey lines of 1.5\% and 98.5\% probability and the probability distribution functions in figure 5 reveal that the CFD misses, for that very same operating point of Q/Q\textsubscript{opt} = 0.7, the 98.5\% probability but has a good prediction of the 1.5\% probability.
4.2. Synchronous, Asynchronous and Random separation

When looking at the four pressure sensors around the draft tube cone, it is clear that there is one that systematically has higher fluctuations than the others. The mechanism involved has been explained for the first time by Nishi et al [8]: the pressure fluctuations, for operating points where a rope is present, can be thought of as the sum of two signals called synchronous and asynchronous. This nomenclature is not related to the synchronism with the runner rotation frequency but with the phase of the pulsation between the different pressure sensors.

By definition the synchronous pulsations are pulsations that happen simultaneously in the four draft tube cone pressure sensors. This corresponds to a plane pressure wave travelling past the sensors and that continues its way in the conduit. The asynchronous part is due to the precession of the rope that passes in front of each pressure sensors (and thus produces a similar fluctuation) with a certain lag between two consecutive sensors. Because the synchronous part can travel through the hydraulic circuit (runner, spiral casing, penstock etc.), it is the most critical part for mechanically related issues like resonance or power swings. Being able to quantify accurately this part of the signal is of prime practical interest in the design phase.

Recently Dörfler and Ruchonnet [9] proposed a refinement to the analysis of the draft tube pressure signals. We briefly recall here the ideas of the proposed method. We assume that the pressure sensor signals are actually composed of a synchronous, asynchronous and random part. We assume also that one random part is perfectly uncorrelated with the three other random parts. If we define a mean signal that is the average of the four pressure sensors for each time step, this mean signal will be the sum of the synchronous part and some random signal. The asynchronous part will cancel out due to the equidistant position of the sensors. Then, for each of the sensor signals, we can subtract the computed mean signal to get four “corrected” signals composed of the asynchronous part (correlated between the fours signals) and the random part. By computing the coherence, in the frequency domain, between each one of those corrected signals, it is possible to distinguish and to compute the power spectrum of the asynchronous signal and the power spectrum of the random part, and finally to compute the synchronous power spectrum without its random part.
Once the power spectrum density estimations are available, one can determine the signal energy, in RMS units for each of the three components (synchronous, asynchronous and random). Additionally the total energy, corresponding to the square root of the sum of the squared components can be computed. This technique has been applied both for the laboratory measurements and the CFD predictions. Results are compared in figure 7.

First of all we can observe that the peak of the total energy at $Q/Q_{\text{opt}} = 0.7$ is predicted by CFD almost at the same place albeit with a higher amplitude in comparison with the measurements. Because the total energy is a sum of squared terms, the total energy is mainly driven by the asynchronous component in the range of $Q/Q_{\text{opt}}$ between 0.65 and 1. In that range the CFD results, while having a similar shape, overestimate this asynchronous component, which explains the discrepancy in the total energy. The most interesting finding is the level of the synchronous part which is the most important one for resonance issues. In the range of $Q/Q_{\text{opt}} > 0.65$, this component is well predicted in comparison with the measurements.

In the CFD results the predicted peak at $Q/Q_{\text{opt}} = 0.55$ that was already present in figure 6 is found again in figure 7. This is probably due to an operating point that is challenging for the CFD model: the rope disintegrates and reappears in regular intervals. Such a physical behaviour can pose numerical challenges because it corresponds typically to a solution that, even physically, is hard to converge into a periodic state. With good probability this situation explains the presence of the peak: a non-stabilized regime. Moreover this appearance followed by a disappearance of the rope is by nature a low frequency phenomenon and thus would require a much longer simulated time, which is not practical for the design phase. The same reasoning explains the difference of behaviour between experimental measurements and the CFD for the other deterministic parts of the signals, namely the synchronous and asynchronous part that both have their peak at the same operating point.

**Figure 7.** Comparison of the pressure fluctuations separation in Asynchronous, Synchronous and Random part at draft tube cone wall between laboratory measurements (on the left) and CFD analysis (on the right). Same scale on “y”-axis.

Finally, the random part is clearly under predicted by the CFD analysis. The most probable reason is linked with the turbulence model because this random part is essentially due to stochastic hydraulic turbulence. Turbulence models are one of the weakest points in the CFD modelling, and their parameters are hard to adjust adequately.

One can note the following: Even if the fluctuations of the deterministic parts (synchronous and asynchronous) are over predicted in CFD, especially in the range where $Q/Q_{\text{opt}}$ is less than 0.7, they are on the conservative side for the prediction of the mechanical source of excitation. The frequency of the synchronous signal is plotted in figure 8 and, as can be seen, is adequately predicted on a large
range of operation. Despite the deviations from the measurements, the fluctuation predictions provide a good input to correctly assess the risk of hydraulic resonance.

5. Future work: possible improvements

5.1. Improvements of the CFD setup
As always, mesh refinement is an obvious candidate for CFD modelling improvements. However, in the interest of reasonable computation times, too much refinement is not desirable and will only be applied if necessary for a reasonable precision.

We have seen in the previous section that the turbulence model can lead to an under-prediction of the stochastic nature of the flow, especially at very low load. Adjustments of some turbulence model parameters as well as different turbulence models may allow better predictions under such conditions. One outcome of the Synchronous-Asynchronous-Random pulsation separation is that it gives a quick and easy a quantitative measure of how much of the turbulence level, is predicted relative to ordinary experimental measurements.

Another desirable improvement for the future is the cavitation modelling. There are at least two effects that are neglected presently. First of all, no resonance is possible with the present setup because the flow is purely incompressible. To some degree this effect can be mitigated by the analysis proposed in the next subsection. However, the cavitation itself can introduce a flow pattern modification with an undetermined overall impact. The down side of simulating two-phase cavitating flow is a significant increase in computational effort with an increased risk of numerical issues (divergence, low rate of convergence, etc.)

5.2. Additional post processing
As shown in subsection 4.2, the major source of mechanical excitation, namely the synchronous part of the draft tube part load pulsation, can be predicted using CFD, both in amplitude and in frequency, with a good level of accuracy down to \(Q/Q_{opt} = 0.7\). Below \(Q/Q_{opt} = 0.7\) we see an over-prediction. This computed synchronous excitation can be used as an input to a simple one dimensional oscillator that would model the hydraulic circuit behaviour. This would permit to identify the risk of resonance and potentially estimate resonance pressure amplitudes.

The primary parameter of influence on the hydraulic circuit dynamics, as explained in [2], is the cavitation volume that exists in the draft tube. If the cavitation is not modelled in the CFD, it can be estimated by computing the volume of the domain where the pressure is below vapour pressure in the

Figure 8. Comparison of the non-dimensional frequency of the synchronous part between the laboratory measurements and the CFD analysis.
single phase calculation. An amplification factor can be computed and applied to the fluctuations computed with CFD. If this method gives reasonable results, it will be much faster than the full cavitation modelling.

6. Conclusion
The goal of this study is to demonstrate that the use of a systematic approach to compute the draft tube pressure pulsations at the design phase is now amenable. The results are not expected to be perfect because of the simplifications (e.g. mesh density, exclusion of cavitation etc.) taken to make the computations reasonably fast. Even with those simplifications, the predictions are valid provided there is no resonance of the water column in the hydraulic circuit.

The advanced post-processing technique of Synchronous-Asynchronous-Random separation applied to the pressure pulsations in the draft tube cone provides the basic input for a possible additional analysis of the hydraulic system dynamics. This is possible even if the cavitation is not resolved but only estimated.

References
[1] Lucca-Negro O and O’Doherty T 2001 Vortex breakdown: a review Prog Energ Combust 27 431–481
[2] Dörfler P, Sick M and Coutu A 2013 Flow-Induced Pulsation and Vibration in Hydroelectric Machinery (Berlin: Springer)
[3] Stein P, Sick M, Dörfler P, White P and Braune P 2006 Numerical simulation of the cavitating draft tube vortex in a Francis turbine IAHR Section Hydraulic Machinery, Equipment and Cavitation, 23rd Symp., Yokohama, Japan
[4] Vu TC, Nennemann B, Ciocan, Iliescu M S, Braun O, Avellan F 2004 Experimental study and unsteady simulation of the Flindt draft tube rotating vortex rope Proceedings Hydro 2004, Porto, Portugal
[5] Ciocan G D, Iliescu M S, Vu T C, Nennemann B, Avellan F 2007 Experimental study and numerical simulation of the Flindt draft tube rotating vortex Journal of Fluids Engineering 129 146
[6] IEC standard 60193, ed. 2.0 1999, Hydraulic turbines, storage pumps and pump-turbines – Model acceptance tests
[7] Nicolet C, Zobeiri A, Maruzewski P and Avellan F 2010 On the upper part load vortex rope in Francis turbine: Experimental investigation. IAHR Symp. On Hydraulic Machinery and Cavitation, Timisoara
[8] Nishi M, Matsunaga S, Kubota T, Senoo and Surgin Y 1982 Characteristics of conical and elbow-type draft tubes, 12th IAHR Symp, On Hydraulic Machinery, Equipment and Cavitation, Stirling
[9] Dörfler P, Ruchonnet N 2012 A statistical method for draft tube pressure pulsation analysis. IAHR Symp. On Hydraulic Machinery and Cavitation, Beijing