Fluid-structure simulations of the stochastic behaviour of a medium head Francis turbine during startup

J F Morissette\textsuperscript{1} and J Nicolle\textsuperscript{1}
\textsuperscript{1}Institut de recherche d’Hydro-Québec (IREQ), 1800 boul. Lionel-Boulet, Varennes, Qc, J3X 1S1

Morissette.jean-francois@ireq.ca

Abstract. The use of dynamic CFD and FEA simulations of hydraulic turbines for steady operation is now widespread for fatigue analysis. However, machines will undergo a major role change from the traditional base load operation in the coming years. This can have a significant impact on life expectancy. In this regard, a lot of research efforts have been recently devoted to improve understanding of hydraulic turbines transient operations such as startup and runaway. Our recent experiences have shown that almost all turbines present unique behaviour from a stochastic point of view during transient operation, and we consider there is still a lot to learn about how to correlate load patterns to life expectancy. In past work, startup simulations and no-load stochastic predictions were presented separately. To our knowledge, no one has attempted to combine them to predict stochastic stresses by simulations in the case of turbine startup. The challenge is thus to capture much more physics while respecting fluid-structure interaction modelling requirements. This paper presents specifically transient CFD and FEA simulations of a medium head Francis turbine startup aimed at getting stochastic dynamic loads on the runner. The CFD simulations involve, among other things, variable rotating speed, mesh deformation, labyrinth seals, resistive torque, roughness, SAS turbulence modelling. 1-way fluid-structure simulations with time-dependent pressure loads are used to determine the stochastic stresses. The stochastic loads are challenging, and only a part of them were captured. Encouraging results are obtained at the leading edge, but the trailing edge deformations lack most of the content. Simulations and experiments might indicate that a stronger coupling is required to get both the fluid load and the mechanical answer right.

1. Introduction
Historically in developed markets, hydropower was used to provide base-load electricity production with an affordable and mature technology. Nowadays, hydraulic turbines are increasingly required to operate in off-design conditions due to their inherent flexibility. On account of the integration of intermittent renewable energy on the grid (mostly solar and wind) or the liberalization of the energy market, hydro is now seen as valuable asset while pursuing the transition toward a low-carbon world, both for its storage capacity and its fast-response time. However this installed hydro base, while still greatly valuable, consist mostly of aging assets that cannot be replaced or refurbished rapidly without heavy constraints to the grid operation. Utilities then face important challenges to optimize the unit maintenance, reduce the down-time and report investments while, in the meantime, using those assets very differently than their original mission.
All those factors taken together point toward the need for increased knowledge of the impact of operation on the life expectancy and reliability of hydraulic turbines. It is also accepted that this requirement for increased flexibility should be integrated at the design stage for new runner.

It is commonly accepted that startup, runaway and speed no load are among the most damaging conditions a turbine can experience. Ironically, those are also the regimes that are the less understood, and for which predictive simulations are generally not accurate.

In recent years, a lot of efforts has been devoted to better understand and characterize transient operations such as startup and runaway (for example [1][5][7]. Both numerical and experimental approaches have been used to provide detailed measurements and simulations. Those provide some insights to determine which are the critical operating regimes and maneuvers and what are the physical mechanisms responsible for the fatigue damage leading to a turbine failure. From a utility’s point of view, the whole idea is to incorporate the operating cost into decision making algorithms to select which units to operate. This would obviously lead to better economic decisions. To take a step further, this could eventually lead to the concept of a digital twin [3], in which each unit can be considered unique with regards to its design and operation history and should be treated as such.

As a first step towards this goal, computational fluid dynamics (CFD) and finite-element analysis (FEA) practices used for steady state regime analysis had to be adapted to address transient and highly dynamic regimes. However a lot of the approaches developed so far still decouple or simplify the fluid-structure interaction (FSI) analysis to an extent which is not yet fully understood due to a lack of validation data. With increasing computational capability and research effort, we expect the coming years will bring many answers that will lead to better life-assessment studies. This paper is a step toward this goal.

In this paper, we present a numerical case study where a transient stochastic behaviour was experimentally observed during startup. The first part of the paper will present an overview of the case. The second part will show some of the advanced CFD setup that was used. Mechanical simulations are shown at last with a decoupled approach. Since this paper must be seen as a snapshot of a work in progress, concluding remarks will address different aspects of remaining software and computational challenges.

2. Case study
In order to characterize the different operating regimes and transient, an experimental campaign was lead on a Hydro-Quebec medium-head Francis unit that was known to have structural issues. Sensors were installed on the runner to measure pressure and strains on different locations (Figure 1 and Figure 7), while other data were acquired outside the runner like opening, power, rotating speed, vibrations. Many startup scenarios were tested to optimize the runner fatigue endurance [1]. Those data were also collected to further validate numerical models.

Figure 1. Location of the strain gages on the runner, with the experimental setup shown on the right.
Here the most aggressive startup scenario has been retained because an important stochastic behaviour was observed in the pressure and strain measurements, mostly during the acceleration of the runner. The Figure 2 shows maximum principal strain on a strain gage located at the trailing edge of a blade, near the band, as well as the guide vane opening and the rotating speed of the runner for this startup.

![Figure 2. Startup measurements: guide vane opening, rotating speed (left) and max principal strain over time (right).](image)

### 3. CFD results

In order to reproduce the transient stochastic behaviour, the first step is obviously to predict the load. To do so, Ansys CFX was used to model the flow during the startup.

#### 3.1. Setup description

Two different simulation setups (shown on Figure 3) were used to model the whole startup from rest to speed no load, corresponding to a 25 seconds transient event. The smaller one consist of three stay vanes and guide vanes (out of 20), two runner blades (out of 13) and the full draft tube with an extension. It was used to highlight which parameters are more important to capture the global behaviour adequately. While there is a slight mismatch in the angular pitch covered by the fixed and rotating component, it was determined to be an acceptable tradeoff for preliminary simulations. The larger and more complete setup was deemed necessary to capture the expected stochastic behaviour of the flow during the critical phase of the startup. It includes the whole spiral casing with the distributor, a full runner as well as the draft tube. This later setup led to a quite large mesh (around 29.5 million elements, including 12 million in the runner alone) compared to the lighter 3/2 case with just over 4.3 million elements. Both setups are compared with appropriate experimental measurements to ensure that the flow, and thus the loads, are modelled correctly.

![Figure 3. CFD setups: simplified 3/2 (on the left) and complete (on the right).](image)
CFD simulations for hydraulic turbine startup require many not so-standard features from our toolbox. First, the runner rotational speed being variable, it is governed by a first-order derivative from the acceleration, combined with the angular momentum equation of the turbine, and including resistive torques from the generator wind losses, bearings and crown labyrinth seal. Some user-Fortran scripts update the rotational speed accordingly into CFX at each timestep. Second, the motion of the guide vane must be taken into account. As it is known a priori, the motion is prescribed and thus the guide vane mesh deforms accordingly. To model it, Numeca Autogrid was scripted to provide a mesh for a fixed number of openings between the minimal and maximal values recorded during the startup. To keep a good mesh quality over the whole opening spectrum, smaller openings (in this case under 0.75°) are not meshed but handled with porous losses instead. An empirical equation providing a linear increase in the massflow from 0 to 0.75° opening thus covers the first 1.89s of this startup.

Studies were conducted to assess how to include correctly labyrinth seals as it was thought to be not-negligible during a transient like this, more so at speed no load where the leakage flow and the resistive torque can become relatively important. As they are not really any data nor well establish empirical correlations for such flows, simulations were done with both labyrinth seals completely meshed with Numeca Autogrid. In the end since it has quasi-zero resistive torque and flowrate yet requiring an expansive mesh, we concluded that the crown seal wasn’t worth meshing. As mentioned before, an empirical equation was retained to model the small torque. The band seal was kept in the final configuration with a periodic sector (except at its entrance and exit) in order to minimize its computational cost. The band side gap is connected with the runner mesh trough mixing-plane (stage) interfaces, which seem reasonable considering that the flow is mostly asymmetric over there.

To get the proper hydraulic losses, roughness was specified on every wall according to standard values from the different material involved. First cell size in the mesh is chosen to respect the wall function requirements before adding roughness to evaluate y+ values. Considering the variability in flow speed in both space and time during such a transient, it is virtually impossible to be in the full roughness regime (so the model is less accurate), but it is believed that the losses are still better predicted this way than with smooth wall.

In order to predict the stochastic behaviour, a Scale Adaptive Simulation (SAS-SST) approach is used to model turbulence [6]. A “high resolution” spatial discretization scheme in the RANS portion of the flow and a centered scheme when LES is activated are used. 2nd order backward Euler scheme is used to get adequate temporal resolution, while turbulence in itself is kept to a first order scheme. Boundary conditions used are in all cases the same, i.e. a total pressure at the entrance corresponding approximately to the net head, and an “opening” condition with entrainment at the exit, with a zero static pressure specification and zero gradient on the turbulence. Initial condition is a steady state simulation with a prescribed massflow corresponding to the leakage flow through the guide vanes.

Most of the simulations had a fixed timestep of 0.0025s during the whole startup, which corresponds to a maximum of 2.5 degrees of runner rotation (and even under 1deg during the stochastic portion). While the simplified setup led to relatively reasonable computational effort (few hundreds cpus and the whole transient calculated under two days), the full setup led to a costly simulation with over 2-weeks calculation on 640 cpus.

3.2. Global simulation results

CFD simulations were first investigated by looking at the runner rotating speed over time. As the speed is an integral quantity, it is a good indicator to determine how well the average torque is predicted during the startup. Unfortunately, despite many attempts to obtain the correct speed no load velocity, all our simulations overpredict this speed by a margin of 10 to 20% depending on the modelling choices. The inclusion of roughness, labyrinth seals and additional resistive torque from the generator and the bearings resulted in the best setup, visible on Figure 4, with an overprediction of 10%. While an error in the path to speed no load can point toward a bad inertia (coming mostly from the generator, a data that can be hard to recover for existing unit), the curves follow the good tendency, with an increasing offset. This point towards a mismatch with the torque prediction at the speed no-
load regime. Figure 4 also shows a remarkable correspondence between the simplified and complete setup, meaning that the simplified setup is accurate enough for global prediction of startup. Unfortunately, it does not give us access to pressure loads over each individual blades further down.

![Rotating speed over time: CFD versus experiments.](image1)

**Figure 4.** Rotating speed over time: CFD versus experiments.

Looking more closely at the global torque on the shaft (Figure 5), we can see how the CFD compares with the experiments. Overall, the whole tendency is well captured, but the numerical torque is overpredicted in the first 10s and slightly underestimated during the rest of the startup. More importantly, we can see an important dynamic and stochastic content on the experimental strain measurements during the whole startup which should translate to some extent to the individual blade of the runner. This could be linked with the rotor dynamics vibrations which are of course not simulated. While there is much less noise and amplitude, the numerical setup is partially successful in reproducing the stochastic fluctuations on the global behaviour, allowing us to take a step further in the analysis.

![Normalized numerical torque (complete setup) versus experiments over time.](image2)

**Figure 5.** Normalized numerical torque (complete setup) versus experiments over time.

![Normalized numerical torque on each individual blade over time, against mean torque.](image3)

**Figure 6.** Normalized numerical torque on each individual blade over time, against mean torque.

3.3. Stochastic flow
Numerical simulations give access to much richer and detailed results not available in the experiments, thus allowing us to go further that basic validation. To assess the stochastic behaviour and look at a less integral quantity, Figure 6 shows the hydraulic torque per individual blade. It can be seen that there is a lot of variability in the flow. From a fatigue perspective, it is important to note that not a single blade sees exactly the same load over a startup, and each startup is different in itself. Therefore
a certain level of uncertainties or statistical analysis should be taken into account when assessing the mechanical behaviour of the runner.

Pressure measurements also allow comparing the numerical simulations with the experimental results. Figure 7 shows some pressure taps on both the intrados (on the upstream part of the blade) and extrados (further downstream), on two distinct blade channels when available. Results are encouraging on the intrados (Figure 7, left) as the trend is well captured and there is a decent level of fluctuations. However on the extrados (Figure 7, right) values obtained by simulations are almost two orders of magnitude smaller than the experiments. This denotes something is missing on the flow representation near the trailing edge, which could be related to cavitation or fluid-structure interaction, not captured with infinitely rigid blades.

![Figure 7. Pressure on intrados (left) and extrados (right). Experiments (•••) and numerical (---) on two different blades.](image)

As shown on Figure 2, the startup consists of guide vanes opening at a constant rate until reaching a prescribed value (in this case 30% opening), followed by a plateau, and then a closure in order to bring the runner to the synchronous speed. Almost all of the stochastic excitation of the turbine seems to happen in the first part, which last for about 15s. During this phase, the runner accelerates as the massflow increases along the hydraulic torque. It is believed that the non-optimal combination of opening and rotating speed led to formation of important vortical structures at the leading edge. The stochastic behaviour observed could originate from the shedding and collapse of those vortices. As can be seen in Figure 8, many vortices are present at the entrance of the inter-blade channels, denoting a highly disorganized flow with a lot of dynamic fluctuations. They start to regroup into larger and more coherent structures around 10s and the losses diminish rapidly.

![Figure 8. Vortex structures (iso-contours of Q-criterion) showing the evolution of the transient flow, as well as pressure on the blades at three instants during the acceleration phase of the runner.](image)
4. Mechanical results

Following the CFD simulations, the next step is to apply those pressure loads on the runner to evaluate the strains. While some caveats were highlighted with the fluid loads, it is worth it pursuing the work on the mechanical side since it allows closing out the loop between the numerical and experimental campaign. Since some difficulties were expected with the process itself, in this first attempt at modelling a stochastic startup, the fluid-structure approach is kept simple. Only the one-way interaction hypothesis is investigated, assuming that the deformation of the runner doesn’t influence the fluid flow. Although it could shift the response slightly, added-mass effects weren’t taken into account yet to simplify the setup also. Considering the nature of the flow presented previously, a full-runner analysis is done, with a complete pressure load on all the wet surfaces, to take into account the spatial and temporal variability.

To reduce the computational cost, only the first 10s of the startup are simulated, where most of the stochastic excitation was captured.

4.1. Numerical setup

Mechanical simulations presented here are performed within the Workbench suite with Ansys Mechanical 19.0. The numerical setup consists of the full-runner with 13 identical blades, including the fillet, the band and the crown. The retained mesh (mesh A, shown on Figure 9) has a size of 560k nodes (quadratic elements), with curvature-based refinement, as well as specified refinement on the fillets and on two specific blades corresponding to the measurements location. Each strain gage is refined as well very locally. It is meshed in order to obtain a light setup, considering the overall cost of time-dependent mechanical analysis. A second mesh (mesh B) is also used (with around 780k nodes) with supplementary refinement on the two blades of interest, and particularly in the higher deformation regions.

A zero-displacement boundary condition is specified at the coupling flange, while gravity and centrifugal forces set with the calculated rotating speed are specified as loads. Pressure loads varying over time are applied on the crown and band external side based on average value from the CFD results.

Finally, the main pressure load calculated from CFD is applied on the blades, as well as on the crown and the band inside walls. All calculations conducted on the mechanical side are time-dependent with a different pressure load for each timestep mapped through an external data module. Interpolation from the CFD mesh to the mechanical mesh is done with the default triangulation profile preserving from Ansys. In a previous paper [4][2], an external app was used to apply pressure loads, which was not really efficient memory-wise. The current approach also has its limitation, since it takes an equivalent amount of time to load the pressure loads than to do the calculation in itself. With all that being said, it appears that in the end, time-dependent structural simulations, with thousands or so of loads required to get the stochastic behaviour, are still not so affordable. It requires about a day on 32 cpus to simulate 10s with 4000 loads with the sparse matrix solver, and approximately as much to import the loads first and to import the results at the end as well.

In the hope to accelerate future structural simulations, additional tests were performed involving subsampling of the available CFD data with a selected number of timesteps. Those lighter simulations involved just half or a quarter of them, cutting the total processing time proportionally. To put this into perspective, this translates into a numerical sampling rate of 100 to 400Hz which compares to the experimental strain gage signal at 2500Hz. While lower, it is believed that the numerical sampling should cover most of the energetic modes for this runner that could be involved in the stochastic process.
4.2. Numerical and experimental comparisons

Figure 10 shows the maximal principal strain amplitude for the different simulations along with the experiments. As fatigue failure reliability is the main purpose of this study, the strains compared here are taken as peak to peak amplitude value of the maximal principal strain. The value is obtained during the first 10s of the startup at the leading edge (LE) and trailing edge (TE) on two different blades. We can see that there is a lot of spread in the experimental amplitudes from one startup to another and that overall, the experimental values are higher than the numerical ones by a factor of 2 to 10. While still too low, numerical strain fluctuations seem to be better estimated at the LE.

![Figure 10. Max principal deformation on four sites, comparing five different mechanical simulations with two repetitions of the startup.](image)

![Figure 11. Principal deformation over time on one LE site, comparing four selected mechanical simulations.](image)

Numerical signal is also shown on the Figure 11 for four selected simulations. It shows how subsampling the load enables to reduce the computational cost of such simulations because the frequency content of the pressure load is still well captured, even at 100Hz. The highest resolution still delivers the largest amplitude though. The influence of time integration, i.e. the inertial effects, is really small as quasi-static results mostly matches transient ones. Not surprisingly, results were independent of the numerical damping (not shown here), which gave similar results even when reduced by an order of magnitude. Mesh B which is more refined locally even reduced the largest cycle while showing approximately the same time-signature.

Looking back at Figure 10, it is obvious that the whole fluid-structure model described here does not predict adequately the deformations. The two experimental repetition of the startup show
maximum principal strains ranging from 500 to 1200 µS while their numerical counterparts show a
range from 50 to 200 µS. It is worth mentioning that a single peak at the TE manage to double the
amplitude observed. A different look at those data is presented in Figure 12 and Figure 13. Only one
simulation (Δt = 0.0025s) is presented versus experiments at the two strain gage locations on blade
#4.

![Figure 12. Max principal strain over time for LE gage on blade 4, exp versus simulation.](image1)

![Figure 13. Max principal strain over time for TE gage on blade 4, exp versus simulation.](image2)

Even though the amplitudes are off on both the leading edge and trailing edge, the curves show a
slightly different story. During most of the acceleration part, the trend is well captured at the LE site,
with about half the amplitude. This was to be expected as the trend was similar on the pressure
measurements at the intrados. It leads us to believe that getting the forcing pressure accurately would
translate quite directly on improvement on the structural side. It is true except at the very beginning,
where the hydraulic load seems too strong before 2s. Also noteworthy is the mismatch in fluctuations
starting around 8s. From this moment, the stochastic fluctuations rapidly die off in the simulations
while they keep growing in reality.

At the TE site, the mechanical simulation predicts about two orders of magnitudes smaller
fluctuations. Basically, it completely misses the deformation of the blade at this location. It follows
the trend for the first 3s, and then stays about flat in the simulation while it keeps growing in the real
startup. This correlates quite well with the pressure measurements on the extrados which are located
closer to the trailing edge. Both set of experimental data (pressures and strains) seem to indicate high
frequency vibrations of the blade at the trailing edge.

Needless to say, something is amiss. First, due to stochastic nature of the phenomena at play, we
know that a totally deterministic approach is not sufficient, and it is well illustrated just by looking at
the range of amplitudes from the two startup repetition. Second, we are looking at signals with quite
different temporal resolution, lacking some richness in the simulations. It was expected, due to the
choice of timesteps and meshes on both the fluid and the structure, as well as the turbulence modelling,
which filter out a part of the content. Still, the leading edge results show that when you get the load
about right, there is some hope to predict the trend. A preliminary analysis of different signals from
the experiment, not shown here, has provided some indications that all the signals from an individual
blade and the adjacent hydraulic channel are synchronized in phase over the stochastic transient event.
It could be explain by a stronger than anticipated coupling between the fluid and the structure. If it is
the case, it is virtually impossible to get the load right independently from the structure calculation as
it has been done here.

5. Conclusion
This paper presents a first attempt to predict numerically stochastic strains in a transient regime, herein
a startup, and to compare the results with what was experimentally measured on a medium head
Francis turbine. As expected from the beginning, this task is by no mean trivial. For life assessment, it remains crucial to determine the amplitude of the largest cycles on such events to determine the real operating cost of a startup, either to incorporate it into production so it can be operated accordingly or eventually to mitigate such a hydraulic and mechanical behaviour.

Scale-resolved turbulence modelling (SAS-SST) was used to predict by CFD the pressure loads on the runner during a startup. A complete setup is necessary to get the load over the entire runner, but a smaller setup with a reduced number of blades is helpful to perform a sensitivity analysis of the various numerical parameters necessary to simulate such a transient. This startup revealed itself quite challenging to model. Even with the inclusion of many advanced features such as labyrinth seals modelling, roughness and variable speed among other things, results remain mitigated with an overestimation of 10% of the speed no load velocity. The stochastic loads were also only partially captured at the leading edge whereas almost nothing was found at the trailing edge.

Starting with dynamic content of the hydraulic loads that was under-predicted, it was not expected to reproduce adequately the mechanical stochastic response. Still, a 1-way FSI approach was used to evaluate the strains and compare them with experimental results. Many structural models, including quasi-static and transient subsampling, were studied to get a first overview on how to model stochastic deformations during a transient event like a startup. In the end, without the need for stronger coupling, inertial effects remain of small importance and all models provided similar answers. The two sites compared for the strains show a different story. At the leading edge, while the largest cycles are not fully captured, the stochastic behaviour is present and the trend during the accelerating portion of the startup is good. At the trailing edge however, very low levels of fluctuations was found numerically while high levels were experimentally measured. This shows basically that the limitations in the fluid prediction translate directly to the mechanical model. Said otherwise, to get the stochastic mechanical behaviour one must probably get the fluid load right, and that might involve many things.

With the tools and computers available today, tackling transient and stochastic at the same time remains an important challenge and more development will be needed, as well as simplification in the approach once the physics is well understood. As mentioned previously, this paper is a first step on FSI stochastic during transient event and many actions are considered to improve on the results presented here. Hinted by what is happening at the trailing edge, where a high level of both pressure and strain fluctuations was measured, this could indicate the need for a stronger coupling between the fluid and the structure. That would mean essentially testing a two-way FSI approach. Another important question that has not been answered is the role of cavitation in the stochastic behaviour during the startup. From preliminary analysis, it should be present at different instant and thus could influence the flow dynamic in the runner over the whole startup. This field obviously has strong ties with turbulence modelling and other numerical issues. On the mechanical side, a decomposition of the hydraulic forces into modal basis could be quite helpful to understand which modes are being excited and contribute to this strong dynamic response. It could also accelerate the whole mechanical process significantly. Finally, there is a limit to what we can do with a deterministic numerical model so we will have to find a way to feed a statistical model with simulations or to add a statistical layer to physics-based simulation to get stochasticities prediction more useful in the future.

References

[1] Cote P, Dumas G, Moisan E, Boutet-Blais G 2014 Numerical investigation of the flow behavior into a Francis runner during load rejection, 27th IAHR Symp. on Hydraulic Machinery and Systems, Montréal, Canada
[2] Gagnon M, Nicolle J, Morissette JF, Lawrence M, 2016 A look at Francis runner blades response during transients, 28th IAHR Symp. on Hydraulic Machinery and Systems, Grenoble, France
[3] Glaessgen EH, Stargel DS 2012 The digital twin paradigm for future NASA and US air force vehicle, AIAA 53rd Structure, structural dynamics and materials conference
[4] Morissette JF, Chamberland-Lauzon J, Nennemann B, Monette C, Coutu A, Giroux AM, 2016 Stress predictions in a Francis turbine at no-load operating regime, 28th IAHR Symp. on Hydraulic Machinery and Systems, Grenoble, France

[5] Mössinger P, Jester-Zürker R, Jung A 2017, Francis-99: transient CFD simulation of load changes and turbine shutdown in a model sized high-head Francis turbine J. of Physics Conf. 782 012001

[6] Nennemann B, Morissette JF, Chamberland-Lauzon J, Monette C, Braun O, Melot M, Coutu A, Nicolle J, Giroux AM, 2014 Challenges in dynamic pressure and stress predictions at no-load operation in hydraulic turbines, 27th IAH Symp. on Hydraulic Machinery and Systems, Montréal, Canada

[7] Nicolle J, Morissette JF, 2016 Simulation of air admission in a propeller hydroturbine during transient events, 28th IAHR Symp. on Hydraulic Machinery and Systems, Grenoble, France