Hydraulic turbine start-up: a fluid-structure simulation methodology

Matthieu Melot$^{1,2}$, Maxime Coulaud$^1$, Joël Chamberland-Lauzon$^2$, Bernd Nennemann$^2$ and Claire Deschênes$^1$

$^1$ LAMH, Mechanical Engineering Department, 1065 avenue de la médecine, Université Laval, Québec G1V 0A6, Canada
$^2$ Andritz Hydro Canada Inc., 6100 Transcanada Highway, Pointe-Claire, Québec H9R 1B9, Canada
E-mail: matthieu.melot.1@ulaval.ca

Abstract. In the present work, a one-way fluid-structure coupling approach is proposed with a focus on the fluid modelling. Main engineering quantities are compared against the thorough laboratory measurements coming from the BulbT project which is led by the Consortium on Hydraulic Machines and the LAMH (Hydraulic Machine Laboratory of Laval University). Good agreement is obtained with main engineering quantities as well as pressure sensors and strain gauges on the runner blades. Moreover, pressure sensor signals on the runner blades and the hydraulic computations results allow to give a description of the flow topology in the runner during the start-up and speed-no-load operating points.

1. Introduction

The increased share of Renewable Variable Energies (RVE), namely wind and solar, in power grids across the world, imposes drastic changes in the way traditional power plants, including hydropower, are operated. As an example, countries with installed share of RVE above 20% like Denmark, Germany or Portugal can see several days in the year where almost 100%, if not more, of their electricity demand is exclusively covered by RVE [1]. The consequence for hydropower plants is the necessity for a much more flexible way of operation [2] because they need to balance at all time the intermittent nature of the RVE. This results in more frequent start-up sequences.

Unfortunately, those start-up sequences can provoke high stresses in the runner, which are detrimental for its life expectancy as reported by Gagnon et al. [3] or Sick et al. [4]. That is why utilities and turbine manufacturers have a pressing interest in knowing the stresses experienced by the runner.

One avenue to get the start-up runner stresses is to use modern numerical tools, such as a one-way coupling of CFD (Computational Fluid Dynamics) and FEA (Finite Element Analysis) simulations. Due to the rigidity of the runner it is assumed that the deflection of blades have negligible effects on the flow patterns. The merits of this approach are a reduced cost and faster answer compared to in-situ strain gauge measurements. Additionally, those computations would be made during the design phase. The risk of not being able to lower stresses enough even by optimizing the guide vane opening law would be reduced because runner geometry modifications would still be possible. Finally due to the nature of CFD, hydraulic phenomenon occurring during start-up are much easier to unveil than during prototype measurements. This
an additional piece of information will help designers in optimizing runner geometries for start-up sequences.

Several groups have reported investigations of start-up simulation methodologies and the related load rejection transient operation which consists in an abrupt disconnection of the generator from the electrical grid. Both start-up and load rejection share several CFD challenges due to the moving guide vanes and the varying runner rotation speed. IREQ (Research Institute of Hydro Québec) focused their studies on the minimal combination of CFD domains to include in the simulations [5, 6] and stated that the draft tube is required for such simulations. More recently they focused on the modelling of air admission in a propeller turbine [7] during start-ups and load rejection. This modelling was necessary to match the runner acceleration measurements because the machine was admitting air at those conditions. This comes with several challenges in terms of computational time and the difficulty to obtain high quality measurements. Mössinger and Jung [8] focused on the coupling of the inlet boundary condition with a one dimensional hydroacoustic model that modulates the inlet condition to represent upstream waterways and the pressure waves associated with them. The compressibility in the CFD domain comes from the cavitation vapor phase. For the load rejection case, the runner acceleration was extremely overestimated without the hydroacoustic modelling of the upstream waterways.

2. BulbT test case presentation

BulbT project is led by the Consortium on Hydraulic Machines and the LAMH (Hydraulic Machine Laboratory of Laval University). One topic of interest established at the project inception was the transient flow dynamics during start-up [9]. The description of the test stand operation for such transient regimes is described by Fraser et al. [10]. As can be seen in figure 1, a free water surface together with compressed air injection is used in the upstream tank to maintain the gross head imposed on the BulbT model during the start-up.

The experimental campaign associated with this configuration and the first results were presented by Coulaud et al. [11]. Three start-up sequences were tested with three guide vane opening rates: $5^\circ/s$, $10^\circ/s$ and $20^\circ/s$. Those opening rates were chosen with dimensionless analysis to be representative of opening rates seen in typical prototype size machines.

The eddy current break used for steady state operating points was electrically disconnected during the start-up measurements and no breaking power was applied. The small leakage flow on closed guide vanes was enough to create a runner rotational speed of 100 RPM. After 10s at that regime, guide vanes were opened at the specified opening rate. After the guide vanes reached an opening of $30^\circ$, the runner was left at the speed-no-load operating point (1000 RPM) until reaching 30s.
For each of the three guide vane opening rates test cases, at least 50 repetitions were made. Those repetitions were carefully synchronized, i.e. for each one of them, the guide vane opening started at 10s exactly. This allows to compute ensemble averaged quantities over repetitions during transient operations:

$$\bar{\chi}(t) = \frac{1}{N} \sum_{i=1}^{N} \chi_i(t)$$

Where:
- $\chi_i(t)$ is the value of the quantity of interest at time $t$ during the $i^{th}$ repetition.
- $N$ is the number of repetitions for the given start-up sequence.

The data acquisition system used 64 channels recorded at 2.5 kHz. Sensors plugged in those channels were found both in stationary and rotating frame of reference. A partial list of recorded quantities included (1) Head, net and gross, (2) Runner shaft rotational velocity, (3) Guide vane opening in degree, (4) Runner torque converted from shaft torque as explained by [12] (5) Instantaneous flow rate obtained from a TR-PIV (Time Resolved Particle Image Velocimetry) measurement in a pipe section upstream of the upstream tank – only 15 repetitions were made with this measurement activated – (6) 26 pressure sensors flush mounted on blade 1 (7) A uniaxial strain gauge on blade 3 that records in a direction parallel to the blade pivot axis.

The pressure sensors are shown with named black dots in figure 7. The strain gauge location was situated between pressure sensors PS_04 and PS_07, at the root of the blade and appears as a red dot labelled Rosette. Note that in this figure blades are mirrored with respect to the figure published in [11] for instance. This is due to Andritz Hydros’ CFD tools that require clockwise rotating runner instead of the tested anti-clockwise.

3. Experimental signal processing
In this work we used exclusively the 20°/s test case that contains 63 start-up repetitions. There were more than 60 channels recorded at 2.5 kHz for 30s each. Data of around 2.3 GB in total were stored in the binary hierarchical HDF5 format. To limit the size occupied in memory, only 7.5s (from 7.5 to 15s) were read for each signal.

Then for each time step and each sensor, ensemble averages were calculated over the repetitions. A 95% confidence interval was computed by assuming that variables were following the Student’s t-distribution. This distribution can be seen as the normal distribution corrected by a factor $\nu = n - 1$, the number of degrees of freedom computed from the number of observations $n$. The advantage of this distribution is that the uncertainty increases if the sample size is small. If the sample size is high enough, the uncertainty decreases and this distribution becomes equivalent to the normal distribution.

The number of degrees of freedom was set for each time step and each variable as being the number of repetitions that had an actual value. Certain signals, the flow rate in particular with 5s window measurements, had missing values for several repetitions near the end. Following the property of the t-distribution, the uncertainty increases.

Results of this process are presented in figures of section 5 for the various experimental signals where blue lines show the ensemble averages and the grey shaded areas represent the 95% confidence interval. The main engineering quantities are in figure 4, the blade pressure sensors in figure 5 and figure 6 and the strain gauge in figure 8.

4. Simulation setup
4.1. CFD setup
For this work Ansys CFX 18.2 has been used for all computations.
4.1.1. Numerical parameters  The three CFD domains, 360° guide vanes, 360° runner and draft tube, that were used in CFD simulations are represented in figure 2. Following the experience of [6, 7] for low head Francis and propeller turbines respectively, the asymmetric domain upstream of the guide vanes, the intake for the case of bulb turbine, was not modelled. It was deemed unnecessary due to the low guide vane openings involved during a start-up. The small openings act as a hydraulic filter and smooth-out all asymmetry.

The runner had its shroud and hub blade gaps modelled with GGI (general grid interfaces) interfaces in their middle in order to simplify the meshing process. The runner was connected to the distributor and the draft tube with transient rotor stator interfaces. Total pressure corresponding to the slightly varying gross head was imposed at the inlet and a static pressure of 0 Pa was imposed at the outlet. The inlet flow was imposed to be normal to the inlet.

The unsteady solver was used with the CFX high resolution scheme for the advection term and a second order backward implicit Euler scheme for the temporal term. The chosen turbulence model was SAS-SST (Scale Adaptive Simulation – Shear Stress Tensor) with automatic transition from boundary layer resolution to wall function treatment depending on the mesh.

The computations were started with the guide vanes opened at 0.1°. An artificial swirling flow tangential to the small guide vane opening was imposed in the whole domain. The gross head was progressively imposed between the time the guide vane was supposed to open between 0 and 0.1°.

Two computations were launched to study mesh convergence. The first one is referred to as the base case. The time step was $5 \times 10^{-4}$s which corresponds to $3^\circ$ of runner rotation per time step at speed-no-load speed. Mesh sizes are reported in table 1. The second computation, the refined one, was chosen so that the time step and the mesh were refined by a factor of 1.25 in each direction. Because meshes are three dimensional, meshes had 1.25$^3 = 1.95$ more nodes. Due to the multiblock nature of the structured meshes, it was not always possible to get exactly a 1.25 refinement of a block in a given direction. Great care has been taken to be overall as close as possible to this factor.

Table 1. Mesh sizes (in millions of nodes) used in CFD domains.

| Domain     | Base Case | Refined |
|------------|-----------|---------|
| Distributor| 2.98      | 5.82    |
| Runner     | 1.61      | 3.16    |
| Draft tube | 0.85      | 1.62    |
| Total      | 5.44      | 10.60   |

Figure 2. CFD domains: guide vane domain in green, runner domain in red and draft tube domain in blue.
4.1.2. Runner acceleration  The runner acceleration modeling had to be programmed with a user Fortran routine. This routine solved Newton’s second law of motion in rotation along the axis of rotation:

\[ I \frac{d\omega}{dt} = \sum T \]  \hspace{1cm} (2)

Where \( I \) is the moment of inertia of the whole shaft line (runner, shaft and eddy current break). The moment of inertia value was estimated from the CAD model. \( T \) are the various torques acting on the shaft line: hydraulic blade torque and friction torque in the bearings and seals. Equation (2) can be discretized with a first order or a second order backward Euler scheme [13]:

\[ \omega_{n+1} = \omega_n + \frac{\sum T}{I} \Delta t + \mathcal{O}(\Delta t) \]  \hspace{1cm} (3)

\[ \omega_{n+1} = \frac{1}{3} \left( 2\frac{\sum T}{I} \Delta t + 4\omega_n - \omega_{n-1} \right) + \mathcal{O}(\Delta t^2) \]  \hspace{1cm} (4)

Both schemes were tested in this study, but the difference in the results were hardly visible and have been left out. We also included an estimated friction torque that depends on the runner rotational speed based on bearing supplier calculation tools.

4.1.3. Moving mesh  The second aspect needing a custom user Fortran routine was the moving mesh for the guide vanes. The opening sequence was dictated by the experimental sequence of 20°/s. The strategy in this work was to use the in-house structured hexahedral meshing tool to generate meshes on demand during the CFD computation. Typically, three to five seconds are sufficient to generate a guide vane mesh at an arbitrary opening.

The more or less 50 mesh parameters for each guide vane opening required during the CFD computation were prepared before the computation launch. The rapidity of our in-house mesh generator allowed for the use of the gradient based, parallel optimizer Nomad [14] to chose adequate parameter values. Only a subset of guide vane openings, called key meshes were optimized. Some of the key meshes are shown in figure 3. All parameters for meshes between key meshes were linearly interpolated based on guide vane openings. The key mesh parameters were stored in a simple CSV file that was used as an input of the custom user Fortran routine.

Being able to control the meshing process instead of relying on CFX mesh deformation resulted in much better mesh quality. For instance, as can be seen from step (c) to (d) on the

![Figure 3. Guide vane meshes at different openings. 0.2° (a), 2.0° (b), 5.0° (c), 15.0° (d) and 30.0° (e).](image-url)
Mesh nodes were moving on the vane profile when considering a frame of reference fixed with respect to the vane. Additionally, changes of topology are required for large opening ranges, from time to time, when cells become too stretched as a consequence of mesh deformation. In those cases, an interpolation is required and introduces numerical shocks.

### 4.2. Finite Element Analysis (FEA) setup

The rigidity of the runner implies that the blade displacements are small enough to consider a one-way coupling between the fluid and structure solutions, meaning that the runner deformations and vibrations are not considered large enough to influence the flow. The absolute pressure field, corresponding to the loading on the blade, was saved at each time step of the CFD computation. The transient dynamic system, solved with FEA software Ansys mechanical, incorporated the solid structure as well as the surrounding water. More details are given in [15]. The results of this analysis were strains computed at the same location as the experimental strain gauges.

### 5. Simulation Results

#### 5.1. Mesh refinement

As explained in section 4.1, two mesh configurations, "base case" and "refined" were evaluated. All results in the following sub sections will show the two mesh configurations. While not perfectly mesh converged, the majority of the results were close enough from each other to say that in practice, the base case is probably enough for such a simulation.

#### 5.2. Main engineering quantities

Experimental and CFD results for the main engineering quantities are shown in figure 4. The three major quantities that are a result of the computations, namely the runner rotational speed, the flow rate, and the runner torque, showed, overall, very good agreement with the experimental results. In more details, the start-up phase from 0 s up to 1.25 s had an almost perfect prediction for the runner rotational speed and the flow rate. For those two engineering quantities, some deviations were observed after 1.5 s which corresponds to the end of the guide vane opening motion. In particular, the final speed-no-load runner rotational speed was over-estimated by both CFD computations by a factor in the order of 5%.

The torque prediction followed the trend of the experimental torque but there was an overestimation of the peak value reached at 0.8 s of around 35%. Due to the inertia of the rotor, there was a delay before seeing the runner rotational speed becoming higher than the experimental measurement. After that period of overestimation, the torque went back to 0 Nm.
as expected for the speed-no-load condition. It is noteworthy that the predicted fluctuation level of the speed-no-load torque is in the same order of magnitude as the experimental one.

5.3. Pressure signals

Figure 5 and figure 6 show the various pressure signals on pressure side and suction side respectively. Once again, there is an good overall agreement between the predictions and the experimental results. The tendencies were well captured, which was expected knowing that the runner torque tendencies were relatively well predicted also.

The first part of the signals when the runner torque increased, were very well predicted by CFD. The biggest differences appear near the end when reaching speed-no-load regime. The largest discrepancy was on the sensor 15 on the pressure side. The sensor was located on the leading edge and had a very low pressure that comes from the misalignment of the mean flow.

![Figure 5. Pressure side pressure signals. Same legend as figure 4.](image-url)
angle on the blade at that operating point. The stagnation point was actually on the suction side and the pressure side leading edge had a strong detachment and was cavitating. The CFD prediction is not as dramatic. But when looking at figure 7, showing the pressure field during speed-no-load, the low pressure area in dark blue is just next to sensor PS_15. This slight shift of the stagnation point and the associated detachment is something frequently seen in CFD simulations.

The other discrepancies with the sensors can be explained in the same way. For instance on the suction side of the blade, there was a diagonal low pressure zone potentially impacting pressure sensors SS_01, SS_02, SS_03, SS_07, SS_10 and SS_11. This area is the trace of the strong recirculation typical of a speed-no-load regime. To obtain a zero torque on the runner, some parts of the blades have to produce a positive torque and some other parts have to produce the same opposite, i.e. negative torque. Negative torque means that the pressure must be higher on the suction side than on the pressure side, which happens near the hub (lower part of the blades shown in 7). This is due to the inversed flow, near the hub that comes from the draft-tube and goes into the runner. It mixes with the turbine flow near the shroud and produces an intense recirculation that lowers the pressure locally. This recirculation is not stationary. Its location fluctuates randomly in time due to the highly turbulent flow. Any shift in the exact positioning of this recirculation will have an impact on the accuracy of the prediction of the

**Figure 6.** Suction side pressure signals. Same legend as figure 4.
pressure signals. CFD is known to have difficulties in predicting accurately flow detachment and recirculation controlled by turbulence.

5.4. Strain gauge signals
The signals recorded by the strain gauges are displayed in figure 8. As for the other main engineering signals, the prediction resulting from the coupled CFD and FEA analysis showed that the runner acceleration period was well captured. When arriving in the speed no-load-regime the mean strain value was correctly predicted, but as already identified in [16], the accurate prediction of the strain fluctuation levels is challenging. In this case, fluctuation levels were missed by a factor two.

This fact can be explained by turbulence. All of those fluctuations are coming from the strong recirculation that generates a turbulence cascade. This cascade is only partially resolved by the SAS-SST turbulence model. In contrast, during the first stages of the start-up, the physical process is governed by inertial effects of the flow. Turbulence plays a very minor role. Not surprisingly, CFD performs relatively well in those conditions.

6. Conclusion
A combined CFD-FEA methodology to predict runner stresses during start-up has been presented and applied on the BulbT test case. This methodology uses CFX and Ansys Mechanical to predict the fluid and structural solutions respectively. The guide vane mesh motion is handled with an in-house meshing tool that generates high quality meshes throughout the simulation.

Based on rotor inertia, guide vane opening sequence and the head, this approach was sufficient for predicting the runner acceleration within 5% accuracy and the blade strains. The highest discrepancies between the prediction and the experimental results were almost always in the speed-no-load regime. Many of the physical phenomena occurring at speed-no-load are a consequence of turbulence, whereas in the first stages of a start-up, flow inertial effects dominate, and the CFD is very well suited to predict those phenomena accurately.
Figure 8. Strain gauge signals. Same legend as figure 4.

7. Acknowledgement
The authors would like to thank the participants to the Consortium on Hydraulic Machines for their support and contribution to this research project: Andritz Hydro Canada Inc., Électricité de France, GE Renewable Energy Canada Inc., Hydro-Québec, Laval University, Polytechnique Montréal, Voith Hydro Inc. Our gratitude goes as well to the Canadian Natural Sciences and Engineering Research Council and the Fonds de Recherche du Québec - Nature et technologies who provided funding for this research.

References
[1] REN21 2017 Renewables 2017: global status report. Tech. rep.
[2] Cochran J, Miller M, Zinaman O, Milligan M, Arent D, Palmintier B, O’Malley M, Mueller S, Lannoye E, Tuohy A, Kujala B, Sommer M, Holtinnen H, Kiviluoma J and Soonee S K 2014 Flexibility in 21st Century Power Systems Tech. rep. National Renewable Energy Laboratory (NREL), Golden, CO.
[3] Gagnon M and Thibault D 2015 6th IAHR International Meeting of the Workgroup on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems, 9-11, Ljubljana, Slovenia September 2015.
[4] Sick M, Oram C, Braun O, Nennemann B and Coutu A 2013 HYDRO 2013, 7-9 October, Innsbrück, Austria
[5] Nicolle J, Morissette J F and Giroux A M 2012 IOP conference series: earth and environmental science 15 062014
[6] Nicolle J, Giroux A M and Morissette J F 2014 IOP Conference Series: Earth and Environmental Science 22 032021
[7] Nicolle J and Morissette J F 2016 IOP Conference Series: Earth and Environmental Science 49 072008
[8] Mössinger P and Jung A 2016 IOP Conference Series: Earth and Environmental Science 49 092003
[9] Deschénes C, Houde S, Aeschlimann V, Fraser R and Ciocan G D 2014 IOP Conference Series: Earth and Environmental Science 22 022013
[10] Fraser R, Coulaud M, Aeschlimann V, Lemay J and Deschénes C 2016 IOP Conference Series: Earth and Environmental Science 49 062006
[11] Coulaud M, Fraser R, Lemay J, Duquesne P, Aeschlimann V and Deschénes C 2016 IOP Conference Series: Earth and Environmental Science 49 062024
[12] Coulaud M, Lemay J and Deschenes C 2018
[13] Fortin A 2001 Analyse numérique pour ingénieur, 2e édition (Montréal: Presses internationales Polytechnique)
[14] Le Digabel S 2011 ACM Transactions on Mathematical Software 37 1–15 ISSN 00983500
[15] Chamberland-Lauzon J, Monette C, Nennemann B, Melot M, Birk S and Ruchonnet N 2018 28th IAHR Symposium on Hydraulic Machinery and Systems, Kyoto, Japan
[16] Nennemann B, Morissette J F, Monette C, Braun O, Melot M, Coutu A, Nicolle J and Giroux A M 2014 IOP Conference Series: Earth and Environmental Science 22 032055