Validation of simulation strategies for the flow in a model propeller turbine during a runaway event

M Fortin, S Houde, C Deschênes
Hydraulic Machines Laboratory LAMH, Faculty of Science and Engineering, Laval University, Québec, Canada
melissa.fortin.2@ulaval.ca, sebastien.houde@gmc.ulaval.ca, claire.deschenes@gmc.ulaval.ca

Abstract. Recent researches indicate that the useful life of a turbine can be affected by transient events. This study aims to define and validate strategies for the simulation of the flow within a propeller turbine model in runaway condition. Using unsteady pressure measurements on two runner blades for validation, different strategies are compared and their results analysed in order to quantify their precision. This paper will focus on justifying the choice of the simulations strategies and on the analysis of preliminary results.

1. Introduction
Transient conditions, such as start-up or runaway, reduce the life expectancy of hydraulic turbines [1][8]. They generate high amplitude pressure fluctuations in all the components of the turbine which accentuate the effect of normal fatigue. Researches into transient flow regimes could yield a better knowledge of the strain fields induced in the turbine thus allowing a better prediction and optimization of its life expectancy. Numerical and experimental studies of the flow dynamics in hydraulic turbines during transient events were, until recently, few and spaced but the increasing demand for renewable energy sources based on wind and solar power has put the problematic in the foreground of hydraulic turbine research.

Runaway is a transient event occurring during a load-rejection through the failure of a turbine security mechanisms, leaving the runner free to reach its maximum rotation speed. Although this is a rare occurrence, it is the most damaging event that can happen to a hydraulic turbine. Running the turbine at its runaway speed is mandatory test at the turbine commissioning. So far, few studies have focused the flow dynamics during runaway [2][4][5][6], either through experimental measurements or numerical simulations.

Using a dual approach combining pressure measurements on a propeller turbine blade and Unsteady Reynolds Averaged Navier Stokes (URANS) simulations, a study was initiated at the Hydraulic Machines Laboratory (LAMH) in Laval University, Canada, to develop strategies to perform transient simulations during runaway conditions [2]. The measurements were performed within the framework of the AxialIT project which was tailored, among other goals, to provide experimental and geometrical databases for numerical validation. Based on published results in other context of hydraulic machine...
unsteady flows [3], compressible and incompressible simulations were performed and benchmarked against the measurements.

This paper presents both numerical methodologies and preliminary results from the simulations. The first part presents the AxialT test case. The next section gives an overview of the selected numerical methodologies and the global simulation strategies. Finally, the preliminary results from compressible and incompressible URANS simulations are presented and discussed.

2. AxialT turbine
The AxialT turbine model has a semi-spiral casing, a distributor with 24 stay-vanes and guide-vanes, a 6 blades propeller runner with a 0.8 specific speed, and a single pier bended draft tube (figure 1a). The scale model was set up on the test rig of the LAMH at Laval University. This test rig consists of a classical closed-loop hydraulic facility with flow rate up to 1 \( \text{m}^3/\text{s} \), head up to 50 m, and a maximal net power output of 170 kW. A vacuum pump is installed on the downstream tank to control the overall pressure in the test loop and thus perform cavitation study. The test rig is configured to perform tests according to the IEC 60193 standards.

The runaway was initiated from a steady-state operating condition, shown in figure 1b, corresponding to a \( N_{11}/N_{11\text{ref}} \) of 1.0 and a \( Q_{11}/Q_{\text{ref}} \) of 0.9868 where “ref” refers to the value at the best efficiency point. In order to avoid damaging the test bench, the head of the turbine was kept low, at 2.5m, which is slightly below the IEC testing norm but still provided the expected behaviour during runaway. To mimic the runaway condition on the test bench, the eddy current break was put off-line with the guide vanes at a fixed angle while the test bench pump was slaved to the head in order to keep it constant. Figure 2 presents the test bench parameters acquired at 5000 Hz. They show that the turbine accelerates to its runaway speed, 80% above its synchronous speed, in about 10 seconds with about half of that speed increase in 1 second. This rapid acceleration corresponds to a sharp drop in torque to its free run value close to 0. During that period, the head drops momentarily while the test bench pump is accelerating in order to keep this value constant. The flow rate appears to increase out of phase with the others values, but this behaviour is linked with the 2 seconds integration time of the flow meter.
Prior to and during the entire runaway event, dynamic pressures on 3 sides of 2 blades were acquired at 5000 Hz with 31 miniature pressures sensors disposed as shown in figure 3 [2]. The pressure sensor signals were transmitted through a telemetric sensors to the test bench operating parameters. Only one run was performed in that particular transient condition so, as such, that test was unique. The comparison with simulations will have to be on the global variations and amplitude of the pressure fluctuations. Newer experiments done in transient regime at the LAMH are designed to be repeated many times to assess the number of samples required for statistical convergence of such flow conditions.

Houde et al. [2] have shown that during the AxialT runaway, the dominant frequencies are sub-synchronous with some of the highest amplitude fluctuations occurring within the first 8 seconds. Cavitation was present at the end of the accelerating phase but can be neglected for the first few seconds. This indicates that:

1- the flow physics require simulations with the entire domain, including the semi-spiral casing and the draft tube since low frequency fluctuations are often attributed to draft tube flow dynamics;

2- validating the numerical strategies over the first 8 seconds should be sufficient to cover the main fluctuations without having to resort to two-phase simulations to account for cavitation.
3. Numerical Methodology

3.1. Geometry and meshes
The numerical domain consists of the semi-spiral case, the distributor, the runner and the draft tube, as shown in figure 4. The walls were modeled without rugosity while the extensions at the inlet and outlet are slip walls only satisfying the impermeability condition. The runner blade tip gaps were neglected as well as the guide vanes overhang gap.

![Figure 4. Numerical domain and interfaces between domain parts]

An unstructured mesh for the spiral casing and the distributor was generated with Numeca Hexpress-Hybrid for a guide vane opening of 33°. Structured hexahedral meshes were generated with ANSYS ICEM CFD HEXA for the runner and the draft tube. Table 1 provides an overview of the meshes parameters and quality. All meshes satisfy strict quality criterion with a minimal angle between the element faces of 20°, a maximal aspect ratio of 300 and a minimum determinant of 0.5.

Table 1. Mesh characteristics.

|                      | Number of elements | Minimal angle (°) | Maximal aspect ratio | Determinant |
|----------------------|--------------------|-------------------|----------------------|-------------|
| Spiral case and distributor | 4 M                | 23°               | 14                   | > 0.5       |
| Runner               | 1 M                | 27°               | 41                   | 0.5         |
| Draft tube           | 2 M                | 27°               | 55.5                 | 0.55        |

Care was taken to construct the meshes in the distributor, the runner and the draft tube so that the resolution in the radial and circumferential directions on each side of their common interface was similar. This requirement was necessary to allow the wakes or the pressure waves to pass through the interface without damping. Finally, the length of the free-slip walls extension at the draft tube exit was extended for the compressible simulation in order to provide a uniform pressure outlet where a non-reflective boundary condition could be applied.

3.2. Numerical strategies
ANSYS CFX 14.5 was the flow solver used in the present study. The URANS simulations were performed using the k-ε turbulence model with scalable wall functions. The advection scheme for steady simulation is a 2nd order upwind formulation, corresponding in CFX to a Blend Factor of 1. The transient simulations use the High Resolution advection scheme.
Table 2 illustrates the different simulations and their links that make up the calculation strategy. Basically, a RANS frozen-rotor simulation is performed to initialize URANS simulations in steady-state with constant flow rate applied at the inlet. Once the monitored pressures on the blade and the torque start fluctuating around average values, the simulation is stopped and the total pressure at the intake is extracted, providing a new inflow condition. This total pressure is then used as inflow condition for a final URANS simulation in steady-state regime giving a flow close to the experimental values. Once again, when the monitored pressure and the torque fluctuate around a fixed value, the transient regime URANS simulations can be initiated.

This procedure was developed because:

1- the experimental flow rate cannot be used in transient regime since the flow meter had too long integration time;

2- the head provided by the RANS Frozen Rotor simulations is different from the head calculated by the URANS simulations. Hence starting a transient regime URANS simulation with a RANS simulation induces a sudden unphysical change in flow rate.

| Inlet boundary condition | RANS Steady = Initial solution | RANS solution = Initial solution | URANS Steady = Initial solution | URANS solution = Initial solution | URANS Transient = Initial solution | URANS Transient = Initial solution |
|--------------------------|--------------------------------|---------------------------------|---------------------------------|---------------------------------|---------------------------------|---------------------------------|
| Inlet boundary condition | Q (kg/s)                       | Q (kg/s)                        | P_{stat}=0 kPa                  | P_{stat}=0 kPa                  | P_{stat}=0 kPa                  | P_{stat}=0 kPa                  |
| Outlet boundary condition| P_{stat}=0 kPa                  | Extract resulting inlet total pressure | Transient rotor-stator         | Transient rotor-stator         | Transient rotor-stator         | Transient rotor-stator         |
| Interface                | Frozen rotor                    |                                 |                                 |                                 |                                 |                                 |
| Rotation speed           | Constant                        |                                 |                                 |                                 |                                 |                                 |

For the runaway simulations, some boundary conditions were modified to mimic the experimental data. Hence the head and the rotation speed were varied at each time step. To do so, a Fortran user routine was developed to feed the relevant data to ANSYS CFX. This routine calculates the value of the head and the runner rotation speed with the polynomial equations at each time step and imposes the calculated values as the new inlet boundary condition and the new runner rotation speed. The goal of this study being the validation of the flow dynamics to fix URANS requirements (turbulence model time step, advection scheme, residual levels), the rotation speed was taken from the measurements and not evaluated through an inertia model. This has the advantage of eliminating one potential source of error that can have a dramatic effect on the flow simulations.

The polynomials were built by splicing the experimental data in sections. Each section is represented by a polynomial equation of degree varying between 2 and 5. These equations are built to form a C^1 continuous curve representing the variation of the head and the runner rotation speed as show in the figure 5. Since the losses between the experimental and the numerical model are different, the head curve was scaled to give an initial flow rate close to the experimental measurements.
For all simulations, the RMS residuals must reach a value of $10^{-5}$ for time steps between $1/4^\circ$ ($\Delta t = 0.00008s$) and $1^\circ$ ($\Delta t = 0.00032s$) of runner rotation. This range of time step is judged sufficient to resolve the dynamic effects of interest since the maximum turbine acceleration is approximately 4.52 rot/s². It's important to keep in mind that the level of accuracy of these simulations is not the same than the one required to determine turbine performances.

3.3. Compressibility

In most of numerical transient analysis, the water is considered incompressible [4][5][6][7]. In fact, water is weakly compressible. The modification of the inlet boundary condition at each time step will cause a flow rate variation that travels as pressure waves through the domain at the speed of sound. The displacement of the pressure waves between two time steps will be small comparatively to the model dimensions. In fact, for a time step of $1.5 \times 10^{-4}$ second and a speed of sound of 1000 m/s, a pressure wave will travel 15 centimeters while the mode turbine length is of the order of 1 meter. So, the flow rate variations will take about 10 time steps to travel from inlet to outlet in a continuous manner during the runaway. This could have an impact on the measured pressure.

So the simulations will be performed by considering both compressible and incompressible fluid in order to assess the impact of the weakly compressibility of water on the dynamic behaviors of the flow. In CFX, water compressibility is introduced as a new material with same characteristics as water but with a variable density. The density was calculated using a barotropic law:

$$\rho = \rho_0 + \frac{P_{rel}}{a^2}$$

(1)

Where $\rho_0$ is the reference water density, $P_{rel}$ is the relative pressure, and $a$ is the speed of sound assumed constant. Unfortunately, the speed of sound in the LAMH test bench is unknown. It was set at 1000 m/s, based on the affirmation of Yan et al. [3] stating that pressure fluctuations propagate in hydraulic machinery with a speed of sound of 900 to 1500 m/s.

For the first compressible simulations, spurious wave reflections appeared inside the domain. In fact, the pressure boundary conditions imposed on the artificial boundaries of the truncated domain caused the reflection of the outgoing waves. The option “Nonreflective” for the acoustic reflectivity was afterwards selected, only to the outlet boundary condition, in order to avoid the spurious wave reflections.
4. Results

In this section, the preliminary results from incompressible and compressible simulations are presented. Simulations are still ongoing to optimize some solver parameters in order to improve the solution quality. For example, the compressible simulations show an unphysical pressure distribution at the outlet surface with the non-reflective conditions. Although this behaviour does not appear to have a significant influence far upstream, it still introduces some uncertainties that can probably be cleared out with appropriate solver parameters.

4.1. Unsteady results for a steady-state operating condition

The first step before launching the transient regime simulations was to validate the boundary conditions and solver setup in the steady regime conditions. Those URANS simulations with constant boundary conditions were also used to fix the mesh quality, the required time steps, the non-reflective conditions and the required advection scheme.

Figure 6 shows the difference in mean pressure between the experimental data, the compressible and incompressible simulations. The time-averaged pressure differences are referenced and normalized to a sensor on the same blade side through the following procedure:

\[
\text{Pressure mean variation (\%) } = \left( \frac{\bar{p} - \bar{p}_{\text{ref}}}{\bar{p}_{\text{ref}}} \right) \times 100, \tag{2}
\]

where “\( \bar{p} \)” is the time averaged value and the subscript “\( \text{ref} \)” indicates the reference sensors.

Figure 6. Pressure mean relative to a sensor on the blade side. Incompressible numerical results in blue, compressible numerical results in green and experimental data in red.

The average pressures coming from the URANS simulations in steady state show a rather good match with the experimental data, both simulations giving results falling below 1% in difference. Both the compressible and the incompressible simulations yield similar average values.
A Fourier spectral analysis is presented in figures 8 and 9 to evaluate the ability of the simulations to capture the dominant frequencies present in the model. The frequencies are normalized to the synchronous frequency associated with the runner rotation speed ($f_{\text{rot}}$). Numerical data for only 8 runner revolutions were available to perform the Fast Fourier Transform. Hence, in order to gain some resolution on the frequency, the signal from the simulations was duplicated in a periodic manner to a continuous signal over 30 sec. Based on the time step, the cut-off frequency of the numerical data is 3333 Hz, close to the Nyquist frequency of the experimental data and way higher than the maximum frequency of interest which corresponds to the guide vanes/runner interactions first harmonic. As can be seen in figures 8 and 9 the dominant frequencies are present for both simulations and their amplitudes close in relative term to the experimental values. For sensors 3 and 24, the main fluctuations are associated with the synchronous component ($f_{\text{rot}}$) through an imbalance in the flow at the distributor-runner interface, as shown in figure 7 where the circumferential component of the flow velocity is presented in function of the circumferential position. The guide vanes/runner interaction ($f/f_{\text{rot}} = 24$) is also well predicted.

**Figure 7.** Numerical circumferential component of the velocity at the interface between the distributor and the runner.

The difference in amplitude between the experimental and numerical values can be attributed to a number of reasons. First, the temporal size of the numerical signal will affect the precision of the amplitude evaluation; therefore simulations are still underway to provide a more accurate evaluation of the amplitude of the dominant frequencies. Secondly, the pressure signal do incorporate the energy from the turbulent fluctuations while those are absent from the URANS pressure signal.
The study of the pressure mean and the frequency analysis shows good agreement between numerical and experimental data. The converged incompressible and compressible solutions are considered precise enough to be used as initial solution for a transient calculation.

4.2. Preliminary transient results for an incompressible flow
An incompressible transient calculation, initiated from a converged steady-state regime URANS is still on-going. At the time of writing, 5 seconds was simulated. This time interval is sufficient to compare the evolution of some quantities with the experimental data. The calculations were performed with at least 64 processors on Intel/Linux Clusters of Compute Canada rated within the TOP500 at the time of publishing. Each second of flow simulation takes approximately one week to calculate. Figure 10 a) shows the evolution of the torque and the mass flow rate in percentage of their initial values. Overall, the head and rotation speed calculated from the Fortran user routine are taken into account adequately by CFX. This led to a variation in Q and T that are consistent with a runaway condition. The numerical torque decreases more slowly that the experimental one. This can be attributed to some damping associated with the head measurements. Further simulations are required to actually evaluate
the amplitude of the delay. Eventually, the head drop within the first second would have to be modified to produce a better torque prediction.

![Figure 10](image)

**Figure 10.** a) Evolution of the torque, in red, the mass flow rate, in black, the head, in green, and the runner rotation speed, in blue, during a transient simulation. b) Evolution of pressure fluctuations on sensor 4, sensor 19 and sensor 23 during a transient simulation. Experimental data in blue and numerical data in red.

The figure 10 b) compares the experimental and numerical pressure signal for 3 sensors. Again, qualitatively, the results are in good agreement showing a trend similar to the experimental data and the appearance of increased fluctuation levels. Nevertheless, the simulations seem to underestimate the fluctuations amplitude which might be attributed to the RANS turbulence treatment. Eventually simulations should be performed with a hybrid model, such as DES or SAS, able to tackle part of the turbulent fluctuations. The incompressible simulations are still on-going and will be stopped around 8 seconds. Compressible transient simulations are also running but at the time of writing, the physical time reached is still too short for any analysis. With the complete results at about 8 seconds, the experimental and numerical transient pressure signals will be analysed and compared with a wavelet method [9].

5. Conclusion
In this paper, a methodology to simulate the flow within a propeller turbine during a runaway event was presented. The methodology is based on incompressible and compressible URANS simulations validated through experimental pressure measurements on runner blades. To mimic the runaway behaviour of the turbine, the inlet boundary conditions and runner rotation speed are changed in function of time during the simulation by using a Fortran routine in ANSYS CFX. These variations of the simulation conditions follow the experimental measurements.

Preliminary results for the steady operating condition show the ability of the numerical model to reproduce the pressure variations. The FFT analysis shows that most of the dominant frequencies are reproduced numerically and the numerical average pressure for each sensor is close to the experimental values. The five first seconds of the transient regime incompressible simulation shows coherent behaviour between experimental and numerical variations of the torque, the mass flow rate and pressures. The compressible and incompressible transient regime simulations are still on-going in order to reach a solution covering the first 8 seconds of the measured runaway.
6. Acknowledgement
The authors would like to thank the participants of the Consortium on Hydraulic Machines for their support and contribution to this research project: Alstom Renewable Power, Andritz Hydro LTD, Hydro-Quebec, Laval University, NRCan, Voith Hydro Inc. Our gratitude goes as well to the Canadian Natural Sciences and Engineering Research Council who provided funding for this research and to Compute Canada who provided the required computing resources.

References
[1] Gagnon M, Tahan S A, Bocher P and Thibault D 2010 Impact of startup scheme on Francis runner life expectancy 25th IAR Symp. On Hydraulic Machinery and Systems (Timisoara, Romania, October 2010) p 8

[2] Houde S, Fraser R, Ciocan G D and Deschênes C 2012 Experimental study of the pressure fluctuations on propeller turbine runner blades: Part 2, transient conditions 26th IAR Symp. On Hydraulic Machinery and Systems (Beijing, China, August 2012)

[3] Yan J, Koutnik J, Seidel U, Hübner B and Scherer T 2009 Compressible Simulation of Rotor-Stator Interaction in Pump-Turbine 3rd IAHR International Meeting of the Workgroup on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems (Brno, Czech Republic, October 2009) pp 261-270

[4] Liu S, Zhou S, Liu D, Wu Y and Nishi 2010 Runaway transient simulation of a Kaplan turbine 25th IAHR Symp. on Hydraulic Machinery and Systems (Timisoara, Romania, 2010) p 8

[5] Li J, Yu J and an Wu Y 2010 3D unsteady turbulent simulations of transients of the Francis turbine 25th IAHR Symp. on Hydraulic Machinery and Systems (Timisoara, Romania, 2010) p 9

[6] Cherny S, Chirkov S, Bannikov D, Lapin V, Skorospelov V, Eshkunova I and Avdushenko A 2010 3D numerical simulation of transient processes in hydraulic turbines 25th IAHR Symp. on Hydraulic Machinery and Systems (Timisoara, Romania, 2010) p 9

[7] Nicolle J, Morissette JF and Giroux AM 2012 Transient CFD simulation of a Francis turbine startup IOP Conf. Ser.: Earth Environ. Sci. 15 062014

[8] Yuanfang H, Guangning L and Shiying F 2012 Research on prototype hydro-turbine operation (Beijing, China, 2012) Foreign Languages Press CO. Ltd p 395

[9] Torrence C and Compo GP 1998 A Practical Guide to Wavelet Analysis Bulletin of the American Meteorological Society 69 pp 61-78