MOC-CFD coupled model of load rejection in hydropower station

S Mandair¹, J F Morissette², R Magnan², B Karney¹

¹ Department of Civil Engineering, University of Toronto, Toronto, M5S 1A4, Canada
² Institut de recherche d’Hydro Québec (IREQ), Varennes, J3X 1S1, Canada

E-mail: sharon.mandair@mail.utoronto.ca

Abstract. A modelling study investigates the consequences of transient flow conditions due to a turbine load rejection. The case study considers a large hydropower station with a long penstock. A three-dimensional (3D) Computational Fluid Dynamics (CFD) model is used to represent the spiral casing, guide vanes, runner, and draft tube. A one-dimensional (1D) Method of Characteristics (MOC) solver simulates water hammer in the penstock. The two models are coupled, to simulate a full load rejection. The results are compared with reference to field measurements and a pure 1D solver, combining the penstock and a turbine model based on machine and conveyance characteristics. A comparison of the high level data (head, flow, torque and rotational speed) reveals the two models reproduce the field data reasonably well. The exception being rotational speed toward the zero torque region, where both models underestimate speed. The model predicts high cycle pressure fluctuations on the turbine blade, which would produce serious mechanical loading. The source of the fluctuations is determined to be unstable vortices within the runner.

1. Introduction

With a growing demand for grid flexibility, hydropower stations are increasingly being used in dynamic operating conditions. This inevitably leads to more rapid material degradation, not just from typical wear and tear, but in extreme cases, major damage. Of specific interest are the transient operating conditions, such as load rejection, speed-no-load, start-up, shut-down, and overspeed conditions.

In the case of sudden load rejection, and subsequent turbine shut down, the generator ceases to apply a resisting torque to the turbine runner. The resulting (torque) imbalance accelerates the runner, and the guide vanes begin to close, cutting off flow to the runner before it reaches runaway conditions [1]. The combined effects of reduced flow and high speed eventually create a negative hydraulic torque, progressively slowing the machine. However, the consequence of closing the guide vanes rapidly causes a pressure surge. Both the overspeed conditions and the surge pressures create a flow regime that is not well understood. The overspeed conditions introduce in the runner swirling flows, flow separation, and continuous formation and destruction of large eddies [2]. The runner blades are thus subject to high-cycle fatigue, vibration and crack propagation [3, 4]. The pressure surge, on the other hand, amplifies high pressures upstream and low pressures downstream of the wicket gates. The former runs the risk of overloading the penstock, while the latter can cause cavitation [5].
Computational Fluid Dynamics (CFD) models are widely used to investigate these details internal to the flow within a turbine. The diversity of CFD studies matches the diversity of hydropower stations and operating conditions. For cases with long penstocks, models typically begin at the spiral casing and fall short on their representation of water hammer. Hence, a better representation of the inlet boundary condition is necessary. In the present study, this is done by way of a coupled interface with a one-dimensional (1D) Method of Characteristics (MOC) model.

The present work models the fluid behaviour in a medium head Francis turbine during a full load rejection toward shut down. The turbine from the spiral casing to the outlet of the draft tube is modelled using the commercial CFD software, ANSYS CFX. To capture the water hammer effects in the penstock, the 3D CFD is coupled with a simple 1D MOC model. Further, a pure 1D MOC simulation of the penstock and turbine is presented for comparison. The models are compared to field measurements obtained during commissioning tests.

2. Models

2.1. 1D MOC Model

In linear hydraulic systems, MOC models are a common approach to simulating water hammer. This is fundamentally an acoustic phenomena, and as such the conservation mass and momentum are solved along the characteristic lines (C+ and C-), which are defined by the acoustic wave speed. In the present case, this is estimated to be 1300 m/s based on measurements. The governing equations takes the form of the equations below [6]:

\[ \frac{\partial H}{\partial t} + a^2 \frac{\partial Q}{gA \partial x} = 0 \]  \hspace{1cm} (1)

where \(a\) is the wave speed, \(x\) is the axial distance, \(t\) is time, \(H\) is head, \(g\) is acceleration due to gravity, \(Q\) is flow rate, and \(A\) is cross sectional area of pipe.

\[ \frac{\partial H}{\partial x} + \frac{1}{gA} \frac{\partial Q}{\partial t} + \frac{fQ|Q|}{2A^2D} = 0 \]  \hspace{1cm} (2)

where \(D\) is diameter, and \(f\) is the Darcy-Weisbach friction factor. The total lumped head loss in the penstock had been measured in previous data collection campaigns. From this the distributed friction factor, \(f\), is estimated. The governing equations are solved along the characteristic lines given by \(\pm a = dx/dt\).

2.2. 1D turbine model

The MOC equation presented above describe the penstock. The turbine can be represented either by CFD (as discussed below) or with the machine performance characteristics. The latter constitutes a 1D turbine model. Such a model must resolve the head, flow, torque and rotational speed of the turbine. The net head, \(H_n\), is the difference between the suction and discharge head:

\[ H_n = H_S - H_D \]  \hspace{1cm} (3)

where \(H_S\) and \(H_D\) are the head at the suction and discharge ends of the turbine, respectively. The second major turbine relationship represents the change in rotational speed as a result of the unbalanced torque, which is the difference between the torque applied by turbine runner, \(T\), and the resistive torque of the generator, \(T_g\). The resulting moment causes a change in rotational speed:

\[ I \frac{d\omega}{dt} = T - T_g \]  \hspace{1cm} (4)
where $T$ is turbine torque, $T_g$ is the resistive torque of the generator, $\omega$ is the angular velocity, and $I$ is the moment of inertia. In the MOC formulation, $T$ is replaced by the average torque over the time step; in other words, the average of the known torque at the beginning of the time step, and the unknown torque at the end.

Finally, Equations 3 and 4 are solved with the help of turbine characteristics in the form of Suter curves, using the scheme presented by Wylie [7].

2.3. 3D CFD Model

To account for the propagation of acoustic waves, the authors use a compressible 3D model. It is described by the continuity and Navier-Stokes equations shown below representing conservation of mass and momentum, respectively:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho v) = 0$$

(5)

and

$$\frac{\partial \rho v}{\partial t} + \nabla \cdot (\rho vv) = -\nabla P$$

(6)

where $\rho$ is density, $v$ is the 3D velocity vector, and $P$ is pressure. These equations are solved here using the commercial code Ansys CFX.

To capture the pressure wave propagation, which is a result of fluid compressibility, the fluid density is modelled as a function of pressure and wave speed:

$$\rho = \rho_I + a^{-2} \Delta P$$

(7)

where $\rho_I$ is the incompressible density.

Flow domains include the spiral casing, guide vanes, runner and draft tube. The geometry is presented in Figure 1, and the mesh statistics can be found in Table 1. The total size of the mesh is 20.9 million nodes; the simulation took 26 days on 512 processors. Due to the scale of the simulation study, a formal mesh convergence study was not performed but the mesh size and node distribution were informed from similar, uncoupled studies. The wall element thickness is based on what is appropriate for steady state flows. The authors are aware this will not fully represent the complexity of the physics in boundary layer during transient conditions. The validation of the model comes from the comparison to field measurements, namely pressure at various locations and rotation speed of the runner. These represent very coarse grained information relative to what is possibly accessible by CFD.

The CFD model is divided into several fluid domains. The spiral casing (including the stay vanes) is the largest component in terms of number of calculation nodes. Most are concentrated around the stay vanes. The inlet boundary condition models the acoustic behaviour in the penstock by way of coupling with MOC. An overlapped coupling scheme as presented by Zhang...
et al. is used [8]. The two models have the same time step, and share two interfaces where the flow conditions are exchanged.

An interpolation interface joins the stay ring and the guide vanes. Following load rejection, the guide vanes respond according to the closure law described in Figure 2 which is obtained from measured data during the load rejection; the guide vanes have a two-slope closure law. A moving mesh scheme is used, similar to that presented in [9], which takes advantage of the mesh deformation feature of CFX. Herein, meshes are generated using Numeca AutoGrid for a range of discrete guide vane openings (every 0.005°) from fully open (27.5°) to almost closed (2.75°). At the beginning of each time step, the mesh is updated. Beyond the minimum angle, the mesh quality becomes poor. Porosity cells could then be adopted to complete the closure, but in the present study the simulation terminates at this point.

The guides vanes and the runner are connected by a transient rotor-stator interface. The rotational speed of the runner is updated at the beginning of each time step based on the torque imbalance (Equation 4). Finally, the draft tube is joined to the runner by a transient rotor-stator connection. The model is single phase, so cavitation is not simulated, which would be a concern in the runner and draft tube. The outlet boundary condition is a pressure head, informed by field measurements.

The simulation uses $k - \epsilon$ turbulence model, with a low intensity at the inlet boundary condition, and a scalable wall function in all domains. The convergence criteria uses a maximum residual target of $1 \times 10^{-4}$. The blend factor is set to 0.75, and a second order backward Euler transient scheme is used. Internal iterations are limited to 15.
3. Results and discussion
Looking first at the high level quantities, the results of the coupled MOC-CFD model are presented in Figure 4. Net head and rotation speed are compared to the pure MOC solver and measured results. Flow rate and torque are only compared to the MOC model since no measurements are available. The turbine begins at full load, and at $t = 0\, \text{s}$ the generator disconnects. The flow rate and torque results reveal good agreement between the two models. Net head is notably different in the MOC model. In this model, dynamic head is neglected in the calculation, i.e., the net head used in Equation 3 is a function of the piezometric head at the inlet and outlet of the turbine. However, the Suter curves used to calculate the turbine operating points are derived from hill charts. These in turn are dependent on net head defined as the difference in total head. To date, the authors attempts to include dynamic head has resulted in an unstable solution. Nonetheless, this presents an avenue to improve the MOC approximation. As for the coupled model, the turbine net head is the difference in total head between the inlet and outlet of the CFD model. It agrees reasonably well with the measured results, particularly at the beginning.

With respect to the rotational speed, the two models reproduce the measurements for the first third of the transient; this roughly corresponds to when the runner is accelerating. They diverge as the runner reaches zero torque, i.e., where the rotational speed plateaus. The MOC model estimates this to be roughly a second sooner than the coupled model. Both models underestimate the maximum speed. The runner also decelerates at a faster rate in the MOC-CFD simulation. The present study excludes cavitation, which may explain why the MOC-CFD model underestimates the peak angular velocity. The presence of vapour in the fluid changes the pressure distribution on the blades, and therefore changes the applied hydraulic torque. A study by Cupillard simulates a turbine runaway with and without a cavitation model, showing considerable impact on the transient rotational speed [10]. With the inclusion of cavitation, the CFD model better represented the maximum speed. Moreover, it is possible the mesh and turbulence models are not adequate to capture the relevant physical phenomena near the zero torque region.

Further, in the 1D representation, the performance curves are the main input into the MOC turbine representation. Beyond the usual data provided by the manufacturer, the hill chart has been informed by measurements on a reduce scale model and some extrapolation from the BEP, notably in the overspeed region. Where the gate opening is small, the hill charts are extrapolated. It is clear from the transient results, these extrapolated curves do not sufficiently represent the turbine behaviour.

![Figure 4: MOC, MOC-CFD, and measured results over time, normalized by rated quantities](image)
3.1. Detailed CFD results

During the transient event, the machine is subject to dynamic pressures, which are of great interest given their consequences for mechanical fatigue. Though limited in its scope, the CFD model provides invaluable detail in this regard. Herein, this section explores the physical phenomena internal to the turbine, with emphasis on the runner. In Figure 5, pressure is observed at four points along the blade during the transient process:

(i) SLB: suction side, close to the leading edge and the band;
(ii) PTB: pressure side, close to the trailing edge and the band;
(iii) PTC: pressure side, close to the trailing edge and the crown;
(iv) SMC: suction side, mid point along the crown.

The pressure trace at point SLB shows high amplitude pressure oscillations. This is by far the most concerning output of the coupled model. The source of the pulsations are investigated first by looking at the inter blade flow behaviour.

3.1.1. Inter blade vortices  When considering the source of the high amplitude pulsation, the complexity of flow in the runner cannot be understated. The acceleration of the runner is accompanied by vortex formations in the inter blade regions, and a progressive change in pressure field. These vortices can be observed by the streamlines traced in Figure 6, which is taken at a mid-section of the runner for various instances throughout the simulation. Secondary flow begins soon after the load rejection in the form of small recirculations on the suction side (Figure 6 (a)). These vortices continue to grow, but do not coincide the high amplitude oscillations, which appear after $t = 5$ s. Figure 6 (b) is a snapshot at $t = 6$ s, when the oscillations are significant. A second vortex emerges in the inter blade region, and there is a lot of kinetic energy throughout. In Figures 6 (c), kinetic energy dissipates in a random manner. The streamlines vary in shape between different blade channels, indicating a more chaotic flow. Coincidentally, the pressure oscillations persist. They decay at roughly $t = 10$ s; the snapshot after this point is shown in Figures 6 (d). Not surprisingly, there is recirculating flow since the runner speed is 40 to 45% greater than initial condition. Interestingly, the two vortices persist, having found a more orderly flow pattern. From blade channel to blade channel, there is far more consistency and similarity.
3.1.2. Blade leading edge  The point SLB is not necessarily representative of the behaviour along the entire leading edge. It is located at an extremity (next to the band), in a region where the blade shape is filleted. SLB is compared to a series of points along the leading edge, on the suction side in Figure 7. The points are not monitored continuously during the simulation. They are instead informed by transient results, which were saved at a frequency of 22 per second of simulation. Consequently, the fine grain detail is not available, and the pressure traces may be missing peaks and troughs. On the left, the geometry of the blade is shown, along with the locations of five points of interest. On the top-right, the head trace for the full transient simulation is shown; the bottom-right shows the same data between 5 and 11 s.

For the first half, the pressure along the leading edge rises almost uniformly for the first 5 s. The behaviour near the band stands out, whereas elsewhere along the blade the pressure converges. High amplitude pressure pulsations begin at \( t = 5 \) s. The region near the band appears to see the highest amplitude fluctuations. Interestingly, these pressure pulsations are in phase along the entire edge. After \( t = 8 \) s, the pulsations desynchronize. Specifically, points SL2 and SL3 toward the middle of the edge fall out of phase with those at the extremities (SL1, SL4 and SLB). The amplitude also increases near the crown at points SL1 and SL2. Behaviour of this sort is likely to aggravate mechanical loading on the blade.

A similar pattern emerges when observing the velocity components at the leading edge. Figure 8 traces radial velocity in the upper graph, circumferential velocity on the bottom. The velocity components are in the rotating frame of reference. Note, that the observed points are not identical to those in Figure 7; they are at the same elevation but at a small distance upstream. The radial velocity is uniform along the span until \( t = 9 \) s, when there is a divergence between the extremities and the mid-span. Similar to the pressure pulsations, the velocity oscillations become out of phase. An interesting tendency appears in the time series for the
between $t = 6$ and $9$ s, the amplitude of oscillations is very high. This may be explained by interactions between the blade edge and the vortices. Recall from Figure 6 (d), at $t = 6$ s each blade channel has a well-formed inter blade vortex near the leading edge. This is not the case in the next snapshot at $t = 8.4$ s. During this time, the inter blade vortices are passing from one blade channel to the next, causing the observed oscillations in circumferential velocity. This behaviour coincides in both time and location with the pressure oscillations at SLB, indicating a strong causal relationship between the unstable vortices and the high amplitude pressure loading.

To fully appreciate the complexity of flow in this region, the profiles of the velocity vectors are plotted in Figure 9. The graph displays radial and circumferential velocities along the span of the blade leading edge at a series of time instants throughout the transient. The top-left shows the location of the span, as well as the point SLB for reference. The top-right graph shows the velocities for the first third of the load rejection. At this time, the radial velocity decreases smoothly and uniformly along the leading edge. The circumferential velocity increases as the runner accelerates and the guide vane angle diminishes. The flow conditions for the second third of the transient can be seen in the bottom-left graph. During this period, velocities become non-uniform. At no point does the runner experience an overall pumping flow rate, but there
are nonetheless instances of backflow. The vortices are highly unstable between $t = 5$ and 10 s, with no discernible patterns emerging. A vortex may roll over the blade edge near the band, at mid-span, or near the crown. During the last third of the transient simulation ($t > 10$ s), the behaviour stabilizes as shown in the graph on the bottom-right. The backflow concentrates at the mid-span of the blade. Coincidentally, the pressure at the mid-span is lower than at the extremities, as observed in Figure 7. Further, the runner is decelerating at this time, concurrent with a deceleration in circumferential velocity as seen here.

Though the flow conditions predicted by the coupled model are quite complex, it is still a simplified version of reality. First, the velocity components indicate very complex behaviour, particularly in the near wall region; the boundary layer is not well represented by the coarse mesh. Second, the pressure oscillations appear to diminish as the runner decelerates. However, the rotational speed predicted by the coupled model falls short of the findings from the field measurements (Figure 4). The model predicts deceleration roughly 2 s earlier than the field measurements. It may be that high amplitude loading persists much longer in the real machine. That said, it is unclear what causes the runner vortices to stabilize. Is it more closely related to the flow rate being reduced; or the transition from runner acceleration to deceleration? In energetic terms, the load rejection is a process of dissipation. When accelerating, part of this energy goes into adding kinetic energy in the rotor. When the turbine decelerates, some of that rotational energy is pumped back. At the same time, the energy carried by the flow also varies. There is less flow, but more head. Thus, it is a complex balance. Third, it is likely the coupled model underestimates maximum runner speed because of the exclusion of cavitation. Vapour is expected to form in the low pressure regions at the centre of a vortex. On a macro scale,
it would change the distribution of pressure on the blade and thus the applied torque. More locally, it is not clear whether it would exacerbate the pressure oscillations observed here. In general, the question of cavitation is significant, and requires further attention.

With respect to the consequences for mechanical loading on the machine, there are still many uncertainties. The amplitude of the pressure oscillations seems very high, and it is difficult to validate the results. The four locations monitored continuously through the simulation coincide with regions of high strain in the real machine. This in combination with the model results can only suggest that inter blade vortices play a role in fatigue loading. Further investigation would be needed to detail and quantify a causal relationship.

4. Conclusion
A full load rejection is studied with a coupled MOC-CFD model, with reference to a pure MOC representation and field measurements. The long penstock creates acoustic transient behaviour, which is represented by a weak compressibility model of the fluid and an MOC model of the pipeline. At the macro level, both MOC and MOC-CFD model fall short in their estimation of maximum rotational speed. The CFD approximation would likely be improved with a better representation of cavitation and turbulence. This, however, would demand much greater computational resources from a model that is already very expensive.

Significant perturbations are observed in the runner, in the form of high cycle pressure fluctuations along the leading edge, near the band. This may indicate a source of high cycle fatigue on the blades leading edge. The origin of the fluctuations is determined to be unstable vortices in the runner. The load rejection causes the runner to accelerate, causing recirculating flow in the inter blade region. Two vortices develop and become unstable and disorganized. Along the leading edge, radial and circumferential velocity is observed to be non-uniform for a period of 5 s. As the vortices move from blade channel to blade channel, they create rapid oscillations in circumferential velocity, which translate to high amplitude pressure fluctuations. Finally, it is anticipated that the effect of the unstable vortices are inaccurately represented by the coupled model due to the limitations in scope. First, the mesh is too coarse to reliably capture the boundary layer physics. Second, vaporization in the inter blade region is expected to accelerate the runner more than simulated here, thus causing the real vortices to persist. Further work is needed to estimate the consequences for wear and tear of the runner structure.

References
[1] Zhang Q 2009 Numerical Modeling of Active Hydraulic Devices and Their Significance for System Performance and Transient Protection Ph.D. thesis University of Toronto
[2] Trivedi C, Cervantes M J and Gandhi B K 2016 Energies 9 ISSN 1996-1073 URL http://www.mdpi.com/1996-1073/9/3/149
[3] Gagnon M, Tahan S A, Bocher P and Thibault D 2010 IOP Conference Series: Earth and Environmental Science 12 012107 URL http://stacks.iop.org/1755-1315/12/i=1/a=012107
[4] Trivedi C, Gandhi B and Michel C J 2013 Journal of Hydraulic Research 51 121–132
[5] Nicolle J and Morissette J F 2016 IOP Conference Series: Earth and Environmental Science 49 072008 URL http://stacks.iop.org/1755-1315/49/i=7/a=072008
[6] Karney B W and McInnis D 1992 Journal of Hydraulic Engineering 118 1014–1030
[7] Wylie E B and Streeter V L 1978 Fluid Transients (McGraw-Hill Inc.)
[8] Zhang X and Cheng Y 2012 Journal of Hydrodynamics 24 595–604
[9] Nicolle J, Morissette J F and Giroux A M 2012 IOP Conference Series: Earth and Environmental Science 15 062014 URL http://stacks.iop.org/1755-1315/15/i=6/a=062014
[10] Cupillard S 2019 Modeling the rotor vibrations of a hydropower unit during a runaway regime Proceedings of the 10th International Conference on Rotor Dynamics – IFToMM (Mechanisms and Machine Science vol 63) ed K C and H W (Springer, Cham)