CFD evaluation of added damping due to fluid flow over a hydroelectric turbine blade

J P Gauthier¹,², A M Giroux², S Etienne¹ and F P Gosselin¹

¹Department of Mechanical Engineering, Ecole Polytechnique, Montreal, QC, Canada
²IREQ — Hydro-Quebec Research Institute, Varennes, QC, Canada

E-mail: jean-philippe.gauthier@polymtl.ca

Abstract. To estimate structural fatigue, vibrational response to realistic spectrum of excitations and associated equivalent damping are of paramount importance. In this paper, an approach to quantify flow-induced damping of a relatively heavy fluid on a vibrating hydraulic turbine blade using numerical simulations is presented. First, mode shapes and frequencies of the immersed structure are obtained by modal analysis using the finite element method. Then, forced oscillatory modal motion is prescribed on the structural boundary of unsteady Reynolds-averaged Navier-Stokes flow simulations. Damping is finally computed as the normalized work done by the resulting fluid load on the structure. Validation is achieved by comparing the numerical results with available experimental data for a steel hydrofoil oscillating in flowing water. For this case, the linear increase in the damping ratio with the flow velocity is reproduced within 10% of the experimental values. Application of the method to an actual hydroelectric propeller turbine blade yields a fluid damping value of around 15% of critical damping for its first vibration mode.

1. Introduction
Modern hydroelectric turbines are designed to achieve near perfect efficiency while keeping production costs as low as possible. One way to accomplish such a goal is by reducing the thickness of turbine blades, consequently lowering their stiffness and increasing their propensity to experience critical vibration levels. Moreover, the end-users’ growing need for power regulation due to the integration of intermittent renewable sources leads to more frequent off-design operations as well as machine starts and stops, for which dynamic mechanical stresses are significantly higher than at best efficiency point. In this context, precise knowledge of the turbine dynamic characteristics, especially damping, is essential in order to correctly estimate their fatigue life.

The primary function of water flow through a hydroelectric turbine is to impart rotational motion to the runner, but it affects the structure in multiple other ways. For example, the passage of turbine blades in front of guide vanes induce high frequency water pressure variations, which in turn could excite a vibration mode of the runner. This phenomenon, known as rotor-stator interaction (RSI), has been linked to very early cracking of Francis turbine blades [1]. This particular ability of the flow to act as an excitation force has received much attention in recent years and can now be predicted using computational fluid dynamics (CFD) with sufficient precision for design purposes [2, 3]. Many other phenomena can negatively affect the lifetime of a turbine; a review of fatigue mechanisms was recently published by Liu et al. [4].

Hydroelectric turbines are also susceptible to strong fluid-structure interactions (FSI) because of the high density of water relative to that of the runner material, generally steel. It is common to model FSI effects as fluid-added mass, damping and stiffness implemented in the structural equations of motion.
For a hydraulic turbine, added mass and added stiffness effects are well understood and relatively easy to model [7]. However, while added damping effects have been investigated for fluid conveying pipes, cables, bridges, cylinder arrays in heat exchangers [5, 6, 8] and also aerodynamic fan blades [9], little research effort has been devoted to water flow-induced damping in hydraulic turbines. Recent work has focused on the experimental determination of damping on simplified hydrofoil geometries [10, 11, 12], along with the development of analytical and numerical models able to reproduce and predict those results [13, 14]. Yet, to the authors' knowledge, these models have not been applied to a real turbine geometry.

Different approaches were considered in order to reproduce the experimental data and predict fluid damping on hydrofoils. Liaghat et al. [14] carried out two-way FSI simulations and related damping to the exponential decay rate of the immersed structure's response to an impulse. A slight dispersion of the numerical results both below and above experimental data was observed, but a good agreement with an average overestimation of 12% was found. Monette et al. [13] proposed a theoretical model based on kinetic energy transfer between the fluid flow and the moving structure. The implementation of this model in an in-house FE code yielded an excellent agreement with experiments, which was also observed when they used prescribed-motion CFD as detailed in this paper.

The main contribution of this work is the prediction of water flow-added damping on an actual hydroelectric propeller turbine blade using numerical simulations. Assuming that FSI do not affect linear independence and orthogonality between the runner's eigenmodes, fluid-added damping is computed for one mode at a time. The following modeling sequence to achieve this is proposed:

(i) obtain the mode shape and added mass using modal analysis;
(ii) obtain added stiffness using steady-state CFD;
(iii) compute the system's undamped natural frequency taking into account added mass and added stiffness;
(iv) compute the damping ratio as the normalized work done by the flow on the structure being subjected to prescribed modal oscillations using unsteady CFD.

This paper first introduces the physical vibration model considered. The added damping calculation method is then described. Subsequently, results for a simple hydrofoil which was previously studied experimentally [11, 12] are presented to assess the method. Finally, results for a real propeller turbine blade are given and discussed.

2. Physical model
Assuming independent and orthogonal eigenmodes, the dynamic response of a structure can be described using mode superposition. Furthermore, if only one mode is contributing, the entire structure can be modeled as a single degree of freedom (DOF) linear oscillator. Arguing that structural damping can be neglected as it is much lower than fluid damping [11], the equation of motion for this system is

\[ M_S \ddot{h} + K_S h = F_F(t), \]

where \( M_S \) is the structural modal mass, \( K_S \) the structural modal stiffness, \( F_F(t) \) the total modal force applied on the structure by the fluid and \( h \) the modal deflection of the structure. The overdot denotes time differentiation. Introducing the shape of the selected mode, \( \varphi(x, y, z) \), the structural modal mass and the total modal force can be expressed as follows:

\[ M_S = \iiint_\Omega \rho_S \varphi^2 \, dV, \]

\[ F_F(t) = \int_\Gamma \tau(t) \cdot \varphi \, dS, \]
where $\rho S$ is the density of the structure, $\Omega$ the volume of the structure, $\tau$ the total surface load induced on the structure by the flow and $\Gamma$ the fluid-structure interface.

Assuming that fluid-added mass, damping and stiffness depend linearly on acceleration, velocity and displacement, Eq. (1) can be rewritten as

$$
(M_S + M_F)\ddot{h} + C_F \dot{h} + (K_S + K_F)h = \tilde{F}_F(t),
$$

where $M_F$ is the fluid-added modal mass, $C_F$ is the fluid-added modal damping coefficient, $K_F$ is the fluid-added modal stiffness and $\tilde{F}_F(t)$ is the remaining part of the total fluid force, $F_F(t)$, i.e.

$$
F_F(t) = \tilde{F}_F(t) - M_F \ddot{h} - C_F \dot{h} - K_F h.
$$

The added mass can be interpreted as the mass of fluid accelerated due to the motion of the structure. The added stiffness describes the change in the flow-induced restoring force with the deflection of the structure. For example, the lift coefficient versus the angle of attack slope for a thin airfoil would correspond to a negative $K_F$ value. The added damping represents energy extracted from the structure as a result of work done by the fluid flow. The remaining force term, $\tilde{F}_F(t)$, may be interpreted as an external excitation, such as forces due to RSI in the case of a turbine.

Taking into account structural mass and stiffness as well as fluid-added effects, the undamped natural frequency of the system is

$$
\omega^2_n = \frac{K_S + K_F}{M_S + M_F},
$$

and the dimensionless ratio of the fluid-added damping coefficient to the critical damping coefficient is given by

$$
\zeta = \frac{C_F}{2\omega_n(M_S + M_F)}.
$$

3. Method

Every parameter needed to compute the fluid-added damping ratio from Eq. (7) can be obtained using standard numerical methods. The following section describes the three types of simulation required: modal analysis, steady-state CFD and unsteady CFD.

3.1. Modal analysis

In order to compute natural frequencies and mode shapes of a structure immersed in a fluid at rest, fluid-added mass effects can be taken into account using the finite element (FE) method. To achieve this, structural elastic elements are coupled with potential flow elements [15, 16]. Some FE codes have potential flow elements based on the Laplace equation, whereas others have acoustic elements based on the Helmholtz equation. Both types are suitable as the Laplace equation is the degenerate form of the Helmholtz equation for an incompressible fluid, which is a valid assumption for water. Using this method, simulation results on a Francis turbine reduced-scale model immersed in water showed a very good agreement with measured natural frequencies [7, 17]. Of course, the fluid in a running hydroelectric turbine is not at rest but the flow velocity does not seem to influence added mass effects for the problems addressed in this paper. More details can be found in [18].

An initial classical modal analysis, i.e. modeling only the structure will yield its natural frequency in vacuum, $\omega_v$, which depends only on structural parameters:

$$
\omega^2_v = \frac{K_S}{M_S}.
$$

A second structural-potential flow modal analysis will yield the natural frequency of the structure in fluid at rest, $\omega_{fr}$. The only fluid effect taken into account is the added mass:

$$
\omega^2_{fr} = \frac{K_S}{M_S + M_F}.
$$
Because a potential flow modeling approach is used, the added mass due to viscous effects is not taken into account. However, for low angles of attack, the wake behind a streamlined body is thin and viscous effects remain weak. Therefore, this approximation is acceptable for hydrofoils and turbine blades. The second modal analysis will also give the mode shape of the structure in still fluid, \( \varphi \), which is expected to be only slightly different from that in vacuum. For this purpose, both are considered equal, although further calculations will be made using the mode shape of the immersed structure. Additionally, it is assumed that the mode shape is not affected by the additional stiffness induced by the flow.

Using the discrete representation of \( \varphi \) at the nodes of the FE model, the structural modal mass, \( M_S \), can be computed by numerically integrating Eq. (2). The structural modal stiffness, \( K_S \), and the fluid-added modal mass, \( M_F \), are then obtained from Eqs. (8) and (9).

### 3.2. Steady-state CFD

Assuming the existence of a steady-state flow solution around a motionless structure, the fluid force model (Eq. 5) can be rewritten as the sum of two time-independent forces, one is a constant while the other is proportional to the modal deflection:

\[
F_F = F_0 - K_F h.
\]  

(10)

The fluid-added modal stiffness, \( K_F \), corresponds to \( -\frac{dF_F}{dh} \). Therefore, RANS flow simulations for at least two different values of \( h \) are required in order to quantify \( K_F \). In Eq. (10), it is assumed that the deflection remains small so that the linear approximation holds.

Interpolation of the mode shape onto the CFD mesh must be done in the likely case of non-matching meshes at the fluid-structure interface. To achieve this, it was found that using regression polynomials to obtain a continuous analytical representation of the discrete mode shape was a robust and simple way to implement the method. However, this may not be feasible when working with very complex mode shapes and geometries, in which case direct interpolation of displacement values on the fluid mesh nodes might be a better solution. Modern CFD software often includes mesh deformation algorithms that can easily and automatically manage mesh deformation.

### 3.3. Unsteady CFD

Knowing the fluid-added modal stiffness, the natural frequency of the fluid-structure coupled system, \( \omega_n \), can be computed from Eq. (6). Then, for a prescribed harmonic modal motion of the structure defined by

\[
h(t) = h_0 \sin(\omega_n t),
\]  

(11)

a URANS simulation of the surrounding fluid flow can be carried out. The average modal work done by the flow on the structure, \( W \), can be obtained by projecting the resulting modal force time signal, \( F_F(t) \), on the prescribed modal velocity, \( \dot{h}(t) \):

\[
W = \frac{1}{N} \int_{t_0}^{t_0 + 2\pi N/\omega_n} F_F \cdot \dot{h} \, dt.
\]  

(12)

In the above expression, \( N \) is an integer number of oscillation periods and \( t_0 \) an arbitrary time coordinate. Modal work can easily be evaluated by numerically integrating the unsteady CFD results. Assuming that the frequency spectrum of \( \tilde{F}_F(t) \) does not contain the system’s natural frequency and using a sufficiently high number of periods, substitution of Eq. (5) into Eq. (12) yields the following approximation:

\[
W = -C_F \pi h_0^2 \omega_n.
\]  

(13)

This shows that the net work associated with any forcing frequency different from the system’s natural frequency will vanish over time. Furthermore, added mass and stiffness effects do no net work, as they are conservative forces. The fluid-added modal damping coefficient, \( C_F \), can be obtained by comparing the average modal work computed from CFD results (Eq. 12) to the right-hand side of Eq. (13). The dimensionless fluid-added damping ratio, \( \zeta \), can ultimately be determined from Eq. (7).
4. Validation

The geometry of the hydrofoil used for validation is schematized in Fig. 1. The channel in which water flows has the same height as the mounting blocks. This test case was introduced by Seeley et al. [11] and Coutu et al. [12] as experimental work on three slightly different geometries denominated as H0, H1 and H3, for which damping characteristics were measured. Their exact dimensions cannot be divulged as they are protected by a non-disclosure agreement. For various flow velocities, piezoelectric actuators were used to excite the first eigenmode of the immersed structure. Assuming zero structural damping, fluid-added damping was determined from the amplitude of the resulting vibrations, which was measured using a laser vibrometer.

![Figure 1. Isometric view and cross-section of the hydrofoil. Not to scale: dimensions were altered to protect intellectual property.](image)

Figure 1. Isometric view and cross-section of the hydrofoil. Not to scale: dimensions were altered to protect intellectual property.

![Figure 2. Fluid-added damping ratio for the hydrofoil at various flow velocities: numerical results (○); experiments [11, 12] (×).](image)

Figure 2. Fluid-added damping ratio for the hydrofoil at various flow velocities: numerical results (○); experiments [11, 12] (×).

The fluid-added damping ratio associated with the first bending mode of the H0 hydrofoil was calculated numerically for various flow velocities and plotted in Fig. 2. The details on the implementation of the method can be found in [18]. Experimental damping values from the experiments of Seeley et al. [11] and Coutu et al. [12] are also shown for comparison. Numerical results compare well with experiments, showing 10% overestimation on the $\frac{d\zeta}{dU}$ value obtained by least squares linear regression for both data sets. This overestimation is mostly contained within the experimental error bars.

To explain the linear increase in damping with flow velocity, the modal motion of the hydrofoil can roughly be approximated by the transverse vibration of a rigid two-dimensional wing, described by the displacement $y(t)$. At first order, the angle of attack, i.e. the angle between the wing and the mean flow, is given by $-\dot{y}/U$. The lift force, being proportional to both the angle of attack and the square of the flow velocity, is ultimately proportional to $-U\dot{y}$, which describes a damping mechanism with a linear relationship to flow velocity.

Overall, it is observed that the proposed method is able to predict not only the general trend of damping linearly increasing with flow velocity, but the absolute value of the damping ratio as well. Minor disparities in geometry and material properties between the numerical model and the actual setup are practically unavoidable and might account for some of the error. Nevertheless, a 1-DOF linear model was able to successfully reproduce experimental results within a 10% error margin for a coupled fluid-structure system similar to a turbine blade. This is sufficiently promising to test the method on an actual hydroelectric turbine.
5. Propeller turbine
Investigation of fluid-added damping was conducted on an actual six-blade propeller turbine presently in operation in the Canadian province of Quebec. This particular unit was targeted primarily because it was recently instrumented for an important measurement campaign. An extensive database of pressure sensor and strain gauge readings is now available for various operating conditions. However, this raw data is not readily usable to determine the fluid damping associated with a particular vibration mode. Consequently, experimental data was used mostly to identify dominant frequencies in the turbine blades’ response. This was done to help select a vibration mode which was susceptible to be excited and thus for which the determination of fluid-added damping was relevant.

Damping was computed only for the turbine operating at full load, or maximum rated power. Under such conditions, the flow in the runner is relatively well aligned with the blades and globally smooth. For the selected turbine, this corresponds approximately to a water mass flow rate of $4 \times 10^5$ kg/s and a power of 70 MW. The radius of the runner, $R$, and the chord of the blades, $c$, are both around 3 m. The mean water velocity, $U_{\text{mean}}$, estimated from the mass flow rate and the circular area based on $R$ is of 14 m/s, from which a chord-based Reynolds number of 42 millions can be determined. To simplify the application of the method, it was decided to only consider modes having zero nodal diameter, i.e. modes for which the motion of every blade is in-phase. This way, the size of both FE and CFD models can be reduced, in this case by six, using cyclic symmetry boundary conditions.

Modal analysis of a single cyclically symmetric blade was done in ANSYS Workbench 15.0. A single unstructured mesh for both the blade and the surrounding water with shared nodes at the interface was built. The shape of the fluid domain was modeled as close as possible to reality using available drawings. However, to increase the stability of meshing algorithms, the radial gap of a few millimeters between the tip of the blade and the casing was suppressed by extending the blade. This slight elongation is not significant compared to the radius of the turbine and does not affect the natural frequencies of the structure in vacuum. However, the thereby neglected pressure in the tip gap can possibly result in erroneous evaluation of added mass effects. Fortunately, it was possible to reassess the added mass using the results of transient CFD, where the blade tip gap was modeled. The modal analysis results were used as a good initial guess.

Refinement of an initial mesh comprising approximately 65 k nodes up to a final mesh of around 500 k nodes showed good convergence of computed frequencies. The first mode with zero nodal diameter was retained. Its associated deflection is mostly blade bending, as illustrated in Fig. 3.

**Figure 3.** Total displacement field for the first mode with zero nodal diameter of the propeller turbine. Darker coloration represents higher displacement amplitude; color intensity is linear with respect to displacement.

**Figure 4.** Effect of static modal deflection on the steady modal force induced on the propeller turbine blade at full load. The constant added stiffness is the slope of the linear relation.
Steady-state CFD simulations were conducted in CFX 15.0 for various modal deflections using the standard $k - \epsilon$ turbulence model and scalable wall functions. Around 830 k, $1.4 \times 10^7$ and $1.8 \times 10^5$ hexahedral cells were used to mesh the entire draft tube, one sixth of the distributor and one sixth of the runner (one blade) respectively.

Circumferential averaging interfaces were used to convey flow variable information between the subdomains. Velocity components corresponding to operation at full load were prescribed at the distributor inlet while an average zero pressure condition was set on the draft tube outlet.

Results for the modal force on the blade are plotted against the modal deflection in Fig. 4. The observed linear behavior translates to a constant added modal stiffness, given in Table 1 along with the other modal parameters and the natural frequency of the turbine in flowing water.

| Table 1. Modal parameters and natural frequency for the first mode of the propeller turbine. |
|---------------------------------------------------------------|
| $M_S$ (kg) | $M_F$ (kg) | $K_S$ (N/m) | $K_F$ (N/m) | $f_n$ (Hz) |
| 793 | $2.64 \times 10^3$ | $1.54 \times 10^7$ | $2.99 \times 10^5$ | 10.8 |

The CFD pressure field for the non-deformed runner ($h = 0$) was interpolated on a FE mesh to use as a load in a structural static analysis. The resulting static deflection of the blade was found to be almost identical to the selected mode shape, with an amplitude of $h_{stat} = -17$ mm (negative sign due to the arbitrarily chosen convention for the mode shape deflection). To assess the validity of this one-way fluid-structure analysis, it can be pointed out that:

(i) the added modal stiffness is low compared to the structural modal stiffness ($K_F/K_S < 2\%$). Flow-induced stiffness therefore has little effect on the structural deflection;

(ii) the modal force for $h = -17$ mm is only 1.6% higher than for $h = 0$. This suggests that structural deflection does not significantly affect the flow.

Consequently, one-way coupling was deemed sufficient. To compute damping, modal oscillation of the blade was prescribed around this new equilibrium position. For the sake of simplicity, it was decided to take advantage of the similarity between the static and modal displacement fields, conveniently allowing the prescribed modal deflection to be expressed as:

$$h(t) = h_{stat} + h_0 \sin(\omega_n t).$$  (14)

Of course, if the static and modal displacement fields were different, it would be possible to simply superimpose the two displacement fields, as long as deformations remain small.

In order to reduce computational costs for unsteady RANS simulations, the distributor and the draft tube subdomains were dropped. An inlet velocity field and an outlet pressure field were applied as boundary conditions on the remaining turbine subdomain. These fields were extracted from the $h = h_{stat}$ steady-state solution at the distributor-turbine and turbine-draft tube interfaces, respectively. The flow in the turbine was simulated for vibration amplitudes, $h_0$, of $0.01$ mm, $0.1$ mm, $0.1$ mm and $3$ mm. This was done to study the sensitivity of fluid-added damping to this parameter, from very small oscillations to potentially catastrophic vibrations.

A sample of the time evolution of the modal force on the blade is plotted in Fig. 5. Two distinct frequencies are observable in each signal: the prescribed natural frequency as well as a slower $2.3$ Hz phenomenon. The former corresponds to the combined effects of added mass, damping and stiffness. The latter was linked to leading edge vortices near the runner hub using 3-D visualization of the flow solution. The force amplitude of the natural frequency component was found to be proportional to the prescribed vibration amplitude. On the other hand, the amplitude of the force attributable to vortex shedding showed
no significant dependency on the vibration amplitude. Consequently, for low $h_0$ values, the contribution of vortex shedding relative to the total force becomes higher than added mass, damping and stiffness effects.

![Figure 5](image1.png)  
**Figure 5.** Time evolution of the modal force induced on the oscillating turbine blade for various prescribed vibration amplitudes: 3 mm ( ); 1 mm ( ); 0.1 mm ( ); 0.01 mm ( ).

![Figure 6](image2.png)  
**Figure 6.** Effect of the number of simulated oscillations on the damping ratio of the turbine blade for various prescribed vibration amplitudes: 3 mm ( ); 1 mm ( ); 0.1 mm ( ); 0.01 mm ( ).

In figure 6, the fluid-added damping ratio is plotted against the number of oscillation periods used for its calculation. For every prescribed vibration amplitude, $\zeta$ converges to a stable value as more oscillations are simulated except for $h_0 = 0.01$ mm. It is suspected that this convergence problem is due to the amplitude of the motion-induced forces being lower than that of the vortex-induced forces, which can be interpreted as noise when computing the damping ratio.

Despite this, it is relevant to note that all results plotted in Fig. 6 are comprised inside $\zeta \in [0.143, 0.166]$. This is a relatively small range considering the factor of 300 between the highest and lowest prescribed vibration amplitudes. The sensitivity of the damping ratio to the amplitude of the motion is therefore deemed modest at most. A slight decrease in damping can be observed with increasing vibration amplitude.

6. Conclusion
Damping due to the water flowing around the blades of a hydroelectric turbine can significantly affect its vibrational behavior, which in turn is closely linked to its service life. To characterize this complex fluid-structure interaction problem, an essentially decoupled numerical method, as opposed to a bidirectional fluid-structure coupling, was proposed. Coupling between the fluid and the structure occurs only in the FE model used for modal analysis, where it is relatively easy to implement. Modal motion of the structure is then prescribed in unsteady RANS simulations of the flow and damping is extracted from the resulting forces. Complementarily, fully coupled fluid-structure simulations could be used to validate the results presented in this paper.

While the proposed approach is interesting for its relatively simple modeling, it is important to understand its main limitations:

- a single vibration mode is considered at a time, neglecting any possible interaction between modes;
- linear modeling of flow-induced forces might not be applicable in more chaotic flows, such as those found in turbines operating at partial load;
The method was validated using a test case consisting of a turbine blade-like hydrofoil oscillating in flowing water. For its first vibration mode, experimental results were numerically reproduced within a 10% margin of error over a range of flow velocities.

When the procedure was applied to the first mode with zero nodal diameter of a hydroelectric propeller turbine operating at full load, an added damping ratio of approximately 15% was determined. Sensitivity to the amplitude of the prescribed modal motion was investigated and found to be low. However, when prescribing small amplitudes, interference with vortex shedding seems to occur, hindering the convergence of the damping ratio over time to a stable value.

Through repeated application of the method for several vibration modes and operating points, a damping mapping of a turbine could be established and used to model its dynamic behavior more accurately.

Acknowledgments
Special thanks are due to Andritz Hydro Canada Inc. for greatly facilitating this research by sharing their work on hydrofoils. The authors would also like to thank the Natural Sciences and Engineering Research Council of Canada (NSERC) as well as the Fonds de recherche du Quebec - Nature et technologies (FRQNT) for funding this project.

References
[1] Coutu A, Proulx D, Coulson S and Demers A 2004 Dynamic Assessment of Hydraulic Turbines Hydrovision 2004 (Montreal, Canada)
[2] Nennemann B, Vu T C and Farhat M 2005 CFD prediction of unsteady wicket gate-runner interaction in Francis turbines: A new standard hydraulic design procedure HYDRO 2005
[3] Seidel U, Hubner B, Lofflad J and Faigle P 2012 IOP Conference Series: Earth and Environmental Science 15 052010 ISSN 1755-1307, 1755-1315
[4] Liu X, Luo Y and Wang Z 2016 Renewable and Sustainable Energy Reviews 54 1–14 ISSN 13640321
[5] Paidoussis M 1998 Fluid-Structure Interactions: Slender Structures and Axial Flow Fluid-structure Interactions: Slender Structures and Axial Flow (Elsevier Science) ISBN 978-0-08-053175-5
[6] Blevins R 2001 Flow-induced Vibration (Krieger Publishing Company) ISBN 978-1-57524-183-8
[7] Liang Q, Rodriguez C, Egusquiza E, Escaler X, Farhat M and Avellan F 2007 Computers & Fluids 36 1106–1118 ISSN 00457930
[8] Naudascher E and Rockwell D 2012 Flow-Induced Vibrations: An Engineering Guide Dover Civil and Mechanical Engineering (Dover Publications) ISBN 978-0-486-13613-4
[9] Vasananthakumar P 2011 Computation of aerodynamic damping for flutter analysis of a transonic fan ASME 2011 Turbo Expo: Turbine Technical Conference and Exposition (American Society of Mechanical Engineers) pp 1429–1437
[10] Roth S, Calmon M, Farhat M, Munch C, Bjoern H and Avellan F 2009 Hydrodynamic damping identification from an impulse response of a vibrating blade Proceedings of the 3rd IAHR International Meeting of the Workgroup on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems vol 1 (Bmo University of Technology) pp 253–260
[11] Seeley C, Coutu A, Monette C, Nennemann B and Marmont H 2012 Smart Materials and Structures 21 035027 ISSN 0964-1726, 1361-665X
[12] Coutu A, Seeley C, Monette C, Nennemann B and Marmont H 2012 IOP Conference Series: Earth and Environmental Science 15 062060 ISSN 1755-1307, 1755-1315
[13] Monette C, Nennemann B, Seeley C, Coutu A and Marmont H 2014 *IOP Conference Series: Earth and Environmental Science* **22** 032044 ISSN 1755-1307, 1755-1315

[14] Liaghat T, Guibault F, Allenbach L and Nennemann B 2014 Two-Way Fluid-Structure Coupling in Vibration and Damping Analysis of an Oscillating Hydrofoil *ASME 2014 International Mechanical Engineering Congress and Exposition* (American Society of Mechanical Engineers) pp V04AT04A073–V04AT04A073

[15] Zienkiewicz O C and Bettess P 1978 *International Journal for Numerical Methods in Engineering* **13** 1–16

[16] Brennen C E 1982 A Review of Added Mass and Fluid Inertial Forces. Tech. Rep. CR 82.010 Department of the Navy Port Hueneme, CA, USA contract Number N62583-81-MR-554

[17] Rodriguez C, Egusquiza E, Escaler X, Liang Q and Avellan F 2006 *Journal of Fluids and Structures* **22** 699–712 ISSN 08899746

[18] Gauthier J P, Giroux A M, Etienne S and Gosselin F P 2016 A numerical method for the determination of flow-induced damping in hydroelectric turbines (manuscript submitted for publication)