Investigation on the hydrodynamic damping using prescribed blade motion techniques

P Čupr¹, W Weber² and B Hübner²

¹ V. Kaplan Department of Fluid Engineering, Faculty of Mechanical Engineering, Brno University of Technology, Technická 2896/2, Brno, Czech Republic
² Voith Hydro Holding GmbH & Co. KG, Alexanderstraße 11, 89522 Heidenheim, Germany
cupr@fme.vutbr.cz

Abstract. Increasing requirements for a high efficiency in a wide operating range of hydraulic turbines lead to turbine designs being more susceptible for periodic and stochastic excitation forces. For periodic excitations like rotor-stator interaction, a large distance to runner natural frequencies is not always possible. And stochastic excitation may cause the runner to respond directly with one of its natural frequencies. Therefore, in both cases, the quantification of damping for runner mode shapes is necessary to predict dynamic stresses and to ensure the required lifetime. The main goal of this paper is to present numerical investigations of the hydrodynamic damping using unsteady CFD analyses with prescribed structural motion. The investigation is carried out on a simple hydrofoil, which is placed in a the cavitation tunnel test section. The natural vibration shape of the hydrofoil is prescribed as a periodic motion with the corresponding frequency. As a prerequisite, natural frequency and mode shape have to include the effects of water environment. Two different approaches (in time and frequency domain) have been applied and compared to 2-way fluid-structure-interaction analysis with respect to accuracy and calculation time. In addition numerical results are validated by comparison with experiments. A good agreement is observed for the applied calculation techniques.

1. Introduction

The excitation forces quantification and the correct dynamic response prediction are key factors for a successful design of hydroelectric turbines and its lifetime estimation. Due to the requirement of turbine operation in very broad range, turbines experience significant periodic as well as stochastic loading, especially in part load regimes. Beside rotor-stator interaction (RSI), periodic loading may be connected also with v. Kármán vortex shedding including cavitating vortices or with self-induced instabilities of swirling flow in the draft tube. Consequently, a large distance between runner natural frequencies and periodic excitation frequencies is not always possible. Under stochastic excitation, the runner may directly respond with one of its natural frequencies. Hence, the quantification of damping for runner mode shapes is necessary to predict dynamic stresses and to avoid runner fatigue failure.

The coupling of fluid dynamics inside the runner and structural dynamics of the runner have to be considered as a strong fluid-structure interaction (FSI) problem in which added mass, damping and stiffness need to be taken into account. With decreasing blade thickness of newly designed turbines to provide a high efficiency in a wide operating range, the influence of water on the blade dynamics is rapidly increasing. Considering a single degree-of-freedom (1-DOF) oscillator, the oscillations of a structure submerged in a liquid is described by the equation of motion
\[ m_s \ddot{x} + d_s \dot{x} + k_s x = F(t) , \]  

where \( m_s \) is the mass of the hydrofoil structure, \( d_s \) is the structural damping, \( k_s \) is the stiffness of the structure and \( F(t) \) is the fluid force acting on the hydrofoil. The equation can be rewritten by

\[ (m_s + m_w) \ddot{x} + (d_s + d_w + d_f) \dot{x} + (k_s + k_w + k_f) x = 0 . \]  

The following added effects are present: \( m_w \) is the added mass of still water, \( d_w \) is damping from viscous effects, \( d_f \) is hydrodynamic damping from momentum exchange between flow and structure due to the oscillation, \( k_w \) is the added stiffness due to compressibility effects and \( k_f \) is the added stiffness due to the flow. Added mass [1, 2] and added stiffness effects [3] have been studied widely. However, hydrodynamic damping effects are not yet completely understood. Monette et al. [4] presented an analytical approach for hydrodynamic damping estimation. Experimental investigations of hydrodynamic damping [5, 6, 7] as well as numerical investigations [8, 9, 10, 11] have been presented by various authors. The numerical approaches are mostly based on 2-way fluid-structure interaction analyses of an oscillating hydrofoil. However, the main disadvantage of this approach is the high computational effort leading to the need to develop more efficient numerical techniques, as presented in [3], which are based on the assumption of small linear oscillations with a chosen natural frequency and the corresponding natural mode shape. These techniques use a CFD solver with prescribed mesh motion, without the need of coupling to a structural solver leading to a more efficient solution.

The main focus of this paper is to compare two different approaches for determining hydrodynamic damping effects based on such efficient techniques using the example of the first bending mode of a simple hydrofoil presented in the next section. The results are compared to a 2-way FSI solution [8] and validated by experimental results, obtained from measurements in a high speed cavitation tunnel.

2. Hydrodynamic damping investigation

2.1. Geometry

The tests are carried out at a simple hydrofoil geometry with blunt trailing edge being placed in a high speed cavitation tunnel with square test section, see Figure 1. The hydrofoil is made of stainless steel and fully fixed on both sides. The fluid domain used for numerical calculations corresponds to the test section dimensions. The simulations were carried out using ANSYS 18.1 (CFX and Mechanical).

![Figure 1. Sketch of the hydrofoil geometry in the water tunnel, taken from [8].](image-url)
2.2. Method

The presented hydrodynamic damping investigation carried out by prescribed harmonic blade motion in a flow solver is restricted to following assumptions:

- The hydrofoil performs small linear periodic oscillations.
- The oscillation frequency is equal to the chosen natural frequency (first natural frequency of the submerged hydrofoil).
- The hydrofoil deformation during oscillations is determined by the corresponding mode shape.
- The mode shape is damped weakly such that effect of damping on frequency and mode shape is negligible.
- The numerical model assumes hydrofoil oscillations with a constant amplitude $U_{MAX}$.
- When the hydrofoil reaches its maximum deflection during the periodic motion, the total energy $E_{TOT}$ of the oscillation (potential and kinetic energy) is accumulated into deformation, i.e. $E_{TOT}$ is equal to the elastic strain energy.
- The hydrofoil deformation during the oscillation is elastic, and structural damping is neglected.

The “Modal Work Approach” is utilized for the estimation of the damping ratio $\zeta$:

$$\zeta = \frac{W_{DIS}}{4\pi E_{TOT}}$$ (3)

Herein, $W_{DIS}$ is the modal work exchanged between the hydrofoil and fluid, and $E_{TOT}$ is the total energy of the oscillating structure.

The main steps are summarized in the diagram, shown in Figure 2. The first natural frequency of the submerged hydrofoil that is analyzed in detail within this paper is obtained by finite element modal analysis. To include the added mass effect, acoustic finite elements apply for the fluid domain. The mode shape corresponding to the first natural frequency is normalized to the mass matrix and exported in two ways: First, the deformations $UX, UY$ and $UZ$ are exported only for nodes attached to the hydrofoil surface which is in contact with water (interface between solid and fluid). Secondly, the mode shape deformation is exported for all nodes including the nodes inside the hydrofoil volume. The first set of surface data is used to prescribe a harmonic motion in ANSYS CFX, while the second set of volume deformation is used to calculate the elastic strain energy in a static structural analysis.

![Figure 2. Hydrodynamic damping investigation diagram.](image-url)
In addition to transient analysis with prescribed hydrofoil motion, the ANSYS CFX 18.1 solver provides a new tool for performing harmonic analyses based on prescribed hydrofoil motion to obtain the flow solution in frequency domain. In both cases, the motion of the fluid-structure interface is prescribed in the fluid analysis according to

\[ u(x, y, z, t) = U_{\text{MAX}} \cdot \Phi(x, y, z) \cdot \sin(\omega \cdot t), \]

where \( u \) is the deflection, \( U_{\text{MAX}} \) is the hydrofoil amplitude (scaling factor), \( \Phi \) is the normalized mode shape deflection, \( \omega \) is the natural angular frequency, and \( t \) denotes the time.

Using the transient approach for determining the modal work, various amplitude values \( U_{\text{MAX}} \) in range of 0.5 - 2% of the chord length were tested in the simulation, without showing a significant influence (damping ratio difference is below 2%). The presented results are calculated with an \( U_{\text{MAX}} \) of approximately 1% of the chord length. 200 time steps per oscillation period are applied as a result of a time step study in the transient simulation. The dissipated energy \( W_{\text{DISS}} \) exchanged between hydrofoil and fluid (also called “modal work”) during a single blade oscillation period is given in Equation (5). Here, \( p \) is the pressure, \( \vec{n} \) is the surface normal vector (pointing into the fluid), \( \vec{t} \) is the wall shear stress vector, \( \vec{u} \) is the mode shape velocity vector, \( t \) denotes the time, \( A \) is the surface area, and \( T \) is the blade oscillation period.

\[ W_{\text{DISS}} = - \int_0^T \int_A (p\vec{n} + \vec{t}) \cdot \vec{u}(t) \, dA \, dt \]  

The frequency-domain approach provided by ANSYS CFX assumes harmonically varying flow. This pressure-based procedure applies the Harmonic Balance / Time Spectral Method, see [12, 13]. The flow variation in time is represented by a Fourier series for a prescribed fundamental frequency. The \( n \) harmonics (modes) are retained in the Fourier series for this approximation (usually \( n = 1, 3, 5 \)). Compared to the transient simulation, this approach can save most of the simulation time when a small number of harmonics \( n \) is used [14]. The presented results are calculated with the same maximum deflection as used in the transient analysis and a harmonic order of \( n = 3 \). The dissipated energy \( W_{\text{DISS}} \) follows from Equation (5) as in the transient analysis.

Beside the work of the fluid on the structure, the total energy is required for the calculation of the damping ratio using Equation (3). Considering a 1-DOF system, the total energy is given by the sum of potential and kinetic energy. When the oscillation reaches its maximal deflection, the total energy is equal to the potential energy beeing accumulated in terms of elastic strain energy which is calculated by a static analysis based on nodal displacements scaled with the mode shape amplitude \( U_{\text{MAX}} \). Alternatively, the energy can be obtained in terms of kinetic energy via the modal mass and the natural frequency, as described in [11].

3. Results

3.1. Acoustic modal analysis

The acoustic modal analysis was carried out using 106812 quadratic elements (447327 nodes). The hydrofoil made of stainless steel (\( \rho_s = 7850 \text{ kg/m}^3, E = 210000 \text{ MPa}, \mu = 0.3 \)) is fully fixed on its both sides. The water domain (\( \rho_f = 1000 \text{ kg/m}^3, \text{ speed of sound } c = 1487 \text{ m/s} \)) is modeled with acoustic finite elements. Zero acoustic pressure applies at inlet and outlet of the test section. The “Acoustic fluid-solid interface” is defined on the hydrofoil surface, which is in contact with water. Figure 3 shows the mode shape under investigation.
3.2. Transient and harmonic CFD analyses

Transient and harmonic CFD analyses, both with prescribed harmonic blade motion, use the $k$-$\epsilon$ turbulence model to calculate the hydrodynamic damping. The computational mesh is built using ICEM CFD and consists of 952000 hexahedral elements. No-slip wall boundary conditions with prescribed harmonic mesh motion apply on the hydrofoil surface. At the test section inlet, the flow velocity is varied, while zero static pressure applies at the outlet.

In Figure 4, the results of both methods are compared to the corresponding experimental data. Additionally, the results obtained from full transient 2-way FSI simulations [8] are also included.

![Figure 3. Analyzed mode shape](image)

![Figure 4. Comparison of different methods for determining hydrodynamic damping.](image)

4. Conclusions

Different methods to determine hydrodynamic damping have been investigated and results have been compared to experimental ones. Overall, the results obtained by the methods with prescribed blade motion are in very good agreement with the experiments. Compared to the fully coupled 2-way FSI simulation, the calculation times are much lower.
Acknowledgement
This paper has been supported by the EU project Computer Simulations for Effective Low-Emission Energy funded as project No. CZ.02.1.01/0.0/0.0/16_026/0008392 by Czech Republic Operational Programme: Research, Development and Education, Priority axis 1: Strengthening capacities for high-quality research and the collaboration. The first author would like to express special thanks to the company for the internship opportunity at the Methods & Science department of Voith Hydro Holding GmbH & Co. KG in Heidenheim, Germany.

References
[1] Escaler X, De La Torre O, Farhat M, 2015, Review on parameters influencing the structural response of a submerged body under cavitation conditions, Journal of Physics: Conference Series 656 012150
[2] Liang Q, Rodrigue C G, Egusquiza E, Escaler X, Farhat M, Avellan F, 2007, Numerical simulation of fluid added mass effect on a francis turbine runner, Computers & Fluids 36(6) 1106–1118
[3] Nennemann B, Monette C, Chamberland-Lauzon J, 2016, Hydrodynamic damping and stiffness prediction in Francis turbine runners using CFD, IOP Conference Series: Earth and Environmental Science 49 072006
[4] Monette C, Nennemann B, Seeley C, Coutu A, Marmont H, 2014, Hydro-dynamic damping theory in flowing water, IOP Conference Series: Earth and Environmental Science 22 032044 ISSN 1755-1307, 1755-1315
[5] Roth S, Calmon M, Farhat M, Münch C, Hübner B, 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, Brno, 253-260
[6] Seeley C, Coutu A, Monette C, Nennemann B, Marmont H, 2012, Characterization of hydrofoil damping due to fluid–structure interaction using piezocomposite actuators, Smart Materials and Structures 21(3) 35027
[7] Bergan C W, Solemslie B W, Ostby P, Dahlhaug O G, 2018, Hydrodynamic damping of a fluttering hydrofoil in high-speed flows, International Journal of Fluid Machinery and Systems 11(2) 146-153
[8] Hübner B, Seidel U, Roth S, 2010, Application of fluid-structure coupling to predict the dynamic behaviour of turbine components, IOP Conference Series: Earth and Environmental Science 12 012009
[9] Liaghat T, Guibault F, Allenbach L, 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, V04AT04A073–V04AT04A073
[10] Gauthier J P, Giroux A M, Etienne S, Gosselin F P, 2017, A numerical method for the determination of flow-induced damping in hydroelectric turbine, IOP Conference Series: Earth and Environmental Science 69 341-354
[11] Tengs E O, Bergan C W, Jakobsen K-R, Storli P T, 2019, Numerical simulation of the hydrodynamic damping of a vibrating hydrofoil, IOP Conference Series: Earth and Environmental Science 240 062002
[12] Hall K C, Thomas J P, Clark W S, 2002, Computation of unsteady nonlinear flows in cascades using a harmonic balance technique, AIAA Journal 40 (5) 879-886
[13] Gopinath A, Jameson A, 2005, Time spectral method for periodic unsteady computations over two and three-dimensional bodies, 43rd AIAA Aerospace Sciences Meeting and Exhibit, Reno, AIAA Paper 2005-1220
[14] ANSYS CFX Solver Theory Guide, v 19.1, 2019