FSI analysis of francis-99 hydrofoil employing SBES model to adequately predict vortex shedding

P Čupr¹, D Štefan¹, V Habán¹ and P Rudolf¹
¹V. Kaplan Department of Fluid Engineering, Faculty of Mechanical Engineering, Brno University of Technology, Technická 2896/2, Brno, Czech Republic
cupr@fme.vutbr.cz

Abstract. The added effects from the fluid on a structure submerged in water significantly affect its dynamic response. Since the hydraulic turbine runner is geometrically complex and involves complicated flow phenomena, the research on simple hydrofoil offers a unique opportunity to investigate added effects and mutual interaction of the elastic structure and vortical flow. For this purpose, the fluid structure interaction of Francis-99 hydrofoil was analysed using the Stress Blended Eddy Simulation (SBES). Advantage of this hybrid RANS-LES turbulence model over RANS models is shown by its enhanced ability to represent vortex shedding. The results of modal sensitivity analysis showed, that fillets of the fixed hydrofoil have negligible influence on the natural frequencies of the hydrofoil and therefore the simplified geometry was used. The modal analysis of fully fixed hydrofoil both in the air and submerged in water were carried out to investigate the added mass effect. Moreover, the hydrodynamic damping for various flow velocities was also investigated for the first bending mode. Overall results are complemented by sensitivity analysis of time step size and mesh for both structural and fluid domains. The results showed that the computed damping ratio above the lock-in and vortex shedding frequency at lock-in are largely underestimated. Therefore, the geometry with blunt trailing edge was additionally tested.

1. Introduction
Current trends on the energy market lead to the extension of turbine operating range, i.e. the turbine operation under off-design conditions is more and more requested. Consequently, the turbine components are exposed to the extreme flow conditions from almost no-load to overload experiencing high-frequency rotor stator interaction, low-frequency vortex rope, inter-blade vortices, cavitating structures and repeated start up and shut down, resulting in both periodic and stochastic excitation forces. The coincidence between the periodic rotor-stator interaction or stochastic excitation and the runner natural frequencies is unavoidable. In order to avoid some runner failure and potential fatigue problems, the runner dynamic response investigation with focus on hydrodynamic damping is necessary [1], [2]. The dynamic response of the hydroelectric turbine structural components is strongly influenced by the occurrence of the fluid flow and its added effects (added mass, damping and stiffness). The prediction of added mass effect causing the natural frequency shift has been widely documented in the literature [3], [4], [5], [6], [7]. However, the prediction of hydrodynamic damping and added stiffness both occurring due to the flow has only recently and only partially been treated in the domain of hydraulic turbines.
In recent years both the experimental and numerical hydrodynamic damping investigation has been studied by researchers. Kimber studied the response of cantilevered beams both in the air and submerged in water [8]. Yao investigated the influence of the trailing edge (TE) shape on the hydrodynamic damping using a simple hydrofoil [9]. Coutu and Seeley used the piezoelectric actuators made of Macro Fiber Composites (MFC actuators) to excite the hydrofoil and to measure its dynamic response [10], [11]. Various techniques have been applied and developed to predict the hydrodynamic damping in the field of numerical simulations. The full two-way fluid structure interaction approach was used in several cases and was applied on a simple hydrofoil [12], [13] as well as on the whole turbine runner [14]. However, this approach is very time consuming and the proper analysis setting is not easy and straightforward. Recently, the alternative methods of “Modal Work Approach”, based on the one way coupled FSI simulation, have been presented [15], [16], [17], [18].

The hydrodynamic damping prediction in hydraulic turbines has not been totally understood yet. Therefore, the experimental damping investigation was done at NTNU Trondheim in Norway both for the high head Francis turbine runner and a simple hydrofoil [19], [20]. Since the hydraulic turbine runner is geometrically complex and connected with different complicated flow phenomena, the research on a simple hydrofoil, as a canonical case, is reasonable for study of dynamic response of elastic structure submerged in flowing water.

This paper presents the results of hydrodynamic damping estimation in terms of one-way fluid structure interaction using a “Modal Work Approach” applied on Francis-99 hydrofoil geometry (figure 1). Results on this geometry pointed out the difficulty of accurate prediction of vortex shedding frequency and consequently accurate estimation of hydrodynamic damping. The main reason was linked to the difficulty in onset of boundary layer separation on a round TE with no exactly defined separating point [21]. Consequently, the results of vortex shedding and hydrodynamic damping are in parallel investigated for another hydrofoil with modified geometry, where the original TE, which has one side rounded, is replaced by a blunt TE of thickness 4.5 mm (see, figure 2). The location of boundary layer separation on hydrofoil with sharp blunt TE is fully determined and thus expected vortex shedding frequency might be easily captured in CFD simulation.

Figure 1. Longitudinal cross-section through test section geometry of Francis-99 hydrofoil.

The paper is structured as follows: Section 2 describes Francis-99 hydrofoil test case, the results of modal analysis for both hydrofoil geometries are summarized in Section 3, the Section 4 describes in details CFD simulation and its set up, Section 5 refers about vortex shedding and results of computed hydrodynamic damping analysis with compared with provided measurements are documented in Section 6.

2. Francis-99 hydrofoil test case
The hydrofoil designed especially for the Francis-99 research project has a chord length of 0.25 m and its maximal thickness is 0.012 m. After 0.15 m from the leading edge, the thickness was tapered down
to 4.5 mm at TE, before being chamfered and rounded on one side. The hydrofoil test section of a squared 0.15 m × 0.15 m internal cross-section was milled from a single piece of aluminum alloy (7075 T651) and mounted into the test rig.

Two MFCs actuators from PI Ceramic were mounted on each side of the foil at the widthwise center, close to the trailing edge. These patches were used for hydrofoil excitation in a sinusoidal pattern phase-separated by 180°, resulting in a bending action induced in the blade. The two main results are available: i. Damping ratio versus discharge, ii. Natural frequency versus discharge.

The measurements were done by Bergan et al. [19]. For more details about Francis-99 workshop and experimental results see [22].

3. Modal Analysis
Considering an oscillating hydrofoil submerged into water, its dynamic response is altered by the added effects. This can be modelled as a single degree of freedom (1-DOF) oscillator. The hydrofoil oscillations can be described by equation of motion (1)

$$m_s \ddot{x} + d_s \dot{x} + k_s x = F(t)$$

(1)

where $m_s$ is the mass of the hydrofoil, $d_s$ is the structural damping, $k_s$ is the stiffness of the structure and $F(t)$ is the force acting on the hydrofoil from the fluid. The equation can be rewritten in another form (2).

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

(2)

The following added effects are present: $m_w$ is the added mass of still water, $d_w$ is damping resulting from viscous effects, $d_f$ is damping resulting from momentum exchange between the flow and the structure due to the oscillation, $k_w$ is added stiffness due to the compressibility effects and $k_f$ is added stiffness due to the flow.

The added mass effect $m_w$ controls the frequency of oscillations and causes the shift of natural frequencies compared to the values measured in the air. The typical approach to investigate the added mass effect is the acoustic modal analysis. Carrying out the modal analysis of the hydrofoil both in the air and submerged in water, the natural frequency drop can be investigated and the added mass of water can be calculated using equation (3), where $m_w$ is the mass of the added water, $m_s$ is the mass of the hydrofoil, $f_{water}$ and $f_{air}$ are the hydrofoil natural frequencies calculated in water and air respectively.

$$m_w = m_s \cdot \left( \frac{1}{\left( \frac{f_{water}}{f_{air}} \right)^2} - 1 \right)$$

(3)

Since two different geometries were provided, i.e. the hydrofoil with and without fillets, the modal analysis was carried out for both of them, including the test section walls (figure 1). Additionally, the modal analysis of hydrofoil with blunt TE and without fillets was also calculated.

All three geometries were made of the same material (aluminium alloy 7075 T651). The boundary conditions are set in respect to the experimental setup. The inlet and outlet cross-sections are fully fixed. The computational mesh was created using 99 000 quadratic structural elements SOLID 186 (including fillets) and 200 000 acoustic elements in case of the hydrofoil submerged in water. The material properties are summarized in table 1. The results of modal analysis for all three geometries are presented in the three following tables (table 2 -table 4). The first mode shape of the Francis-99 hydrofoil is shown in the figure 2. Only the symmetric half of the mode shape is shown to provide better look into the test section. The maximum displacement is located in the middle of the trailing edge.

The modal analysis results showed that only negligible difference is between the geometries with and without fillets. Therefore, the first natural frequency of the hydrofoil without fillets (641.78 Hz) is
taken into account for further steps. This value shows a good agreement with the experimental value (622.8 Hz; difference ca. 3%).

**Table 1.** Material properties of the structure.

| Material          | Density (kg/m³) | Young Modulus (MPa) | Poisson Ratio |
|-------------------|-----------------|---------------------|---------------|
| Aluminium alloy   | 2810            | 71700               | 0.33          |

**Table 2.** Results of modal analysis of Francis-99 hydrofoil without fillets.

| Mode | Natural frequency Air (Hz) | Natural frequency Water (Hz) | Ratio water/air (%) | Added mass (kg) | Added mass/hydrofoil mass (-) |
|------|-----------------------------|-----------------------------|---------------------|----------------|-------------------------------|
| 1    | 1711.1                      | 641.78                      | 37.5                | 162.8878       | 6.109                         |
| 2    | 1791.5                      | 1063.80                     | 59.4                | 48.9596        | 1.836                         |
| 3    | 1912.7                      | 1083.40                     | 56.6                | 56.4474        | 2.117                         |
| 4    | 2222.8                      | 1456.50                     | 65.5                | 35.4402        | 1.329                         |
| 5    | 2452.3                      | 1469.70                     | 59.9                | 47.5752        | 1.784                         |
| 6    | 2645.2                      | 1762.20                     | 66.6                | 33.4184        | 1.253                         |

**Table 3.** Results of modal analysis of Francis-99 hydrofoil with fillets.

| Mode | Natural frequency Air (Hz) | Natural frequency Water (Hz) | Ratio water/air (%) | Added mass (kg) | Added mass/hydrofoil mass (-) |
|------|-----------------------------|-----------------------------|---------------------|----------------|-------------------------------|
| 1    | 1743.3                      | 672.76                      | 38.6                | 152.4401       | 5.715                         |
| 2    | 1793.3                      | 868.88                      | 48.5                | 86.9553        | 3.260                         |
| 3    | 1968.2                      | 1127.00                     | 57.3                | 54.6826        | 2.050                         |
| 4    | 2224.1                      | 1457.10                     | 65.5                | 35.4744        | 1.330                         |
| 5    | 2551.5                      | 1469.20                     | 57.6                | 53.7770        | 2.016                         |
| 6    | 2645.4                      | 1744.50                     | 65.9                | 34.6656        | 1.300                         |
Table 4. Results of modal analysis of hydrofoil with blunt TE without fillets.

| Mode | Natural frequency Air (Hz) | Natural frequency Water (Hz) | Ratio water/air (%) | Added mass (kg) | Added mass/hydrofoil mass (-) |
|------|---------------------------|-------------------------------|--------------------|----------------|-----------------------------|
| 1    | 1702.08                   | 637.05                        | 37.4               | 163.7213       | 6.139                       |
| 2    | 1791.39                   | 1066.90                       | 59.6               | 48.5202        | 1.819                       |
| 3    | 1897.48                   | 1077.50                       | 56.8               | 56.0382        | 2.101                       |
| 4    | 2222.60                   | 1458.00                       | 65.6               | 35.3078        | 1.324                       |
| 5    | 2453.08                   | 1471.30                       | 60.0               | 47.4697        | 1.780                       |
| 6    | 2645.14                   | 1749.30                       | 66.1               | 34.3115        | 1.286                       |

Figure 2. The first mode shape of Francis-99 hydrofoil (symmetric half).
4. CFD simulations
In this section the methods used for CFD analysis are summarized. Geometry simplification, computational mesh, solver set-up and turbulence modelling are discussed in particular subsections.

4.1. Geometry
The physical geometry of hydrofoil includes fillets creating the intersection of hydrofoil and side walls. This geometrical feature might pose difficulties for building of structured hexahedral grid thus in case of turbine runners is often neglected without significant impact on the results [23], [24]. As shown in Section 3, the fillets exclusion does not have any significant effect on the modal shapes and relevant natural frequencies. After several preliminary simulations it was deduced that the fillets might be neglected also for the estimation of hydrodynamic damping, thus further only geometry without fillets is considered.

In order to limit the influence of inlet and outlet boundaries, the geometry for mesh has total length of 1.56 m in streamwise direction with hydrofoil placed approximately in the middle (figure 3 a).

![Figure 3](image_url)

**Figure 3.** Computational domain for numerical CFD simulation (a), mesh resolution on hydrofoil surface (b), Detail of mesh resolution near the leading edge (c), Detail of mesh resolution near the trailing edge (d).

4.2. Mesh
Computational mesh was built using ICEM and consists of only hexahedral elements. Two different mesh sizes were tested in order to estimate the influence on the vortex shedding frequency. In order to employ turbulence models with low-Re near wall formulation, the near wall mesh resolution respects the $y^+ < 1$ for the maximal mean velocity in test-section. Two mesh sizes were tested prior to final simulation of Francis-99 hydrofoil geometry. The initial mesh (Mesh #1) consists approximately 3.7 mil. elements and the refined mesh (Mesh #2) with approximately 1.6 times more elements, see table 5 and figure 3 b-d.
Table 5. Mesh sizes for Francis-99 hydrofoil geometry.

| Francis-99 geometry | Mesh #1 | Mesh #2 |
|---------------------|---------|---------|
| Elements            | 3 657 600 | 5 976 000 |
| Nodes               | 3 750 292 | 6 114 290 |

As shown in figure 4 the maximal $y^+$ on hydrofoil surface at $v = 25$ m/s is $y^+ = 1.29$ on the leading edge (LE). The trailing edge (TE) is well resolved with $y^+$ value around 0.75.

![Figure 4. Contours of $y^+$ on hydrofoil surface, LE view (left) and TE view (right).](image)

The mesh for hydrofoil geometry with blunt TE (see figure 5) was created with lower resolution, approximately equal to resolution of Mesh #1, see table 6. It was found that even using this relatively coarse mesh the vortex shedding and consequently hydrodynamic damping were well predicted.

Table 6. Mesh size for hydrofoil geometry with blunt TE.

| Geometry with blunt TE |        |
|------------------------|--------|
| Elements               | 3 221 100 |
| Nodes                  | 3 304 392 |

![Figure 5. Mesh resolution downstream of hydrofoil for geometry with blunt TE.](image)

4.3. Turbulence models
Several turbulence models were tested prior to the final simulations. The main reason was, that it was impossible to capture the Karman vortex street using two-equation model either Standard k-ε based on the wall function approach or SST k-ω based on the Low-Re approach. Consequently, the Scale-Resolving Simulation (SRS) models were employed. Using the coarse mesh (consists of 3.66 mil. elements), the Scale-Adaptive Simulation (SAS) model [25] had difficulties and only the Stress Blended Eddy Simulation (SBES) model was able to easily capture unsteadiness of vortex shedding. Thus for the further simulations only the SBES model was employed. SBES is a hybrid RANS-LES model using the shielding/blending function to automatically switch between RANS and LES solution. Contrary to the original Detached Eddy Simulation (DES) model family this model features a much improved shielding function to protect RANS boundary layers. The SBES model transitions much quicker and on much coarser grids from RANS to SRS mode in separating shear layers than classical DES [25].

4.4. Solver
For all presented CFD simulations the Ansys CFX v19.1 solver was used. The unsteady simulations started from the result of steady-state solution for particular flow rate \( Q \). The sensitivity study was done in order to choose appropriate time-step size \( dt \) for chosen turbulence model and maximal \( Q \) based on the change of vortex shedding frequency \( f_s \).

It was found that the time-step size \( dt \) plays important role on the vortex shedding frequency. Nevertheless, even using very small time-step \( dt = 1e^{-6} \) s the simulated vortex shedding frequency \( f_{s, CFD} \) underestimates experimental value by 11% at resonance \( f_{s, EXP} = 630 \) Hz for \( Q = 0.248 \) m\(^3\)/s (\( v = 11 \) m/s).

All unsteady simulations were running using Second Order Backward Euler transient scheme and High Resolution scheme for turbulence. While for k-\( \varepsilon \), SST k-\( \omega \), and SAS models the High Resolution scheme was selected for advection, the Bounded Central Difference scheme was preferred for SBES model.

5. Vortex Shedding
The accurate simulation of vortex shedding behind the object placed in the fluid stream is the challenging task. Especially if the object has a rounded geometry without sharp edges and thus the exact point of boundary layer separation is unknown. In case of narrow hydrofoil with symmetrical profile the main part of interests is the trailing edge (TE). The shape of TE might either accelerate or delay the boundary layer separation which consequently leads to the vortex shedding in a form of von Kármán vortex street. This flow phenomenon periodically acts on the hydrofoil structure and if the shedding frequency is in resonance with the natural frequency of hydrofoil the strong structural load may lead to cracks. This is very important aspect for turbomachinery design since the Francis turbine runner is composed of several blades resembling hydrofoil-like flow. Nowadays, for turbines working in extended operating ranges the high blade load might frequently appear if the vortex shedding frequency fits the blade natural frequency [1].

For above mentioned reasons the accurate prediction of vortex shedding frequency is very important. As previously mentioned we encountered difficulties to correctly capture vortex shedding frequency and consequently the hydrodynamic damping in simulations of Francis-99 hydrofoil geometry. Thus the additional geometry with modified TE was tested. Both hydrofoil geometries are described in following subsections.

5.1. Francis-99 hydrofoil geometry
The Francis-99 hydrofoil geometry as shown in figure 6 has a one side of the trailing edge rounded which resembles the so called Donaldson trailing edge [27].
Figure 6. Francis-99 hydrofoil geometry.

The vortex shedding behind this hydrofoil is shown in figure 7 in a form of vorticity iso-surface. For lock-in ($v = 11 \text{ m/s}$) the simulated vortex shedding frequency was $f_s = 560 \text{ Hz}$ which is 11\% lower than the measured one.

Figure 7. Vortex shedding at $v = 11 \text{ m/s}$ visualized by isosurface of vorticity component perpendicular to the mean flow ($-1000 \text{ s}^{-1}$ in blue, $1000 \text{ s}^{-1}$ in red), Francis-99 hydrofoil geometry.

5.2. Hydrofoil geometry with blunt TE

As mentioned in introduction the accurate prediction of boundary layer separation is challenging task in case of hydrofoil geometry with one side of TE rounded (as Francis-99 hydrofoil geometry). For this purpose, the hydrofoil geometry with blunt TE is created in order to have known point of BL separation. This yields only small modification of original Francis-99 hydrofoil geometry where the total length of the hydrofoil remains and the rounded corner was replaced by the material to create symmetrical sharp blunt TE, see figure 8.
The vortex shedding behind this hydrofoil is shown in figure 9 in a form of vorticity iso-surface. For the same velocity \( v = 11 \) m/s of Francis-99 hydrofoil lock-in the simulated shedding frequency of this hydrofoil was \( f_s = 495 \) Hz.

6. Hydrodynamic damping estimation

The first six mode shapes and corresponding natural frequencies were calculated using the (acoustic) modal analysis. However, the hydrodynamic estimation presented in this paper was performed only for the first mode shape. This enables to use simplified way of hydrodynamic damping estimation in a form of one-way FSI.

For this purpose, the unsteady simulation with prescribed mesh motion is carried out. The hydrofoil surface and the test section walls move according to the mode shape and its frequency. This satisfies the assumption of hydrofoil oscillation on the first natural frequency and corresponding mode shape. The prescribed amplitude 0.001 m was selected with 350 time steps per mode period.

Consequently, the “Modal Work Approach” is used for estimation of damping ratio \( \zeta \) according to the following equation

\[
\zeta = \frac{W_m}{2\pi M_w \omega^2 u^2}
\]  

(4)
where, $W_m$ is the modal work exchanged between structure and fluid, $M_w$ is the modal mass in water, $\omega$ is modal natural angular frequency and $u$ is the modal reference amplitude. The $W_m$ might be computed using equation (5).

$$W_m = -\int_0^T \int_A (p\vec{n} + \vec{\tau}) \cdot \vec{u}(t) \, dA \, dt$$

(5)

where $p$ is pressure, $\vec{n}$ is surface normal vector (pointing into the fluid), $\vec{\tau}$ is wall shear stress vector, $\vec{u}$ is mode shape velocity vector, $t$ denotes time, $A$ is surface area and $T$ is period of one blade oscillation [16], [17], [18].

Using CFX the area integral of variable Wall Power Density (W) over moving walls of prescribed mode shape might be monitored during one period $T$ of mode cycle. Using the time integration, the modal work $W_m$ is then calculated in same way as described by (5).

The results of damping ratio $\zeta$ estimated from CFD simulation and compared with measurements are shown in figure 10 for both Francis-99 hydrofoil and hydrofoil with blunt TE.

![Figure 10. Results of simulated damping ratio vs experimental measurements.](image)

7. Conclusions
Presented study showed, that the numerical estimation of hydrodynamic damping on Francis-99 hydrofoil geometry might not be a straightforward and easy task. While for flow velocities below the lock-in the damping ratio is in good agreement with measurements, the simulated results above the lock-in are largely underestimated (around 50%). Another disagreement was found for estimation of vortex
shedding frequency. The measured shedding frequency in lock-in ($f_{s, \text{exp}} = 630 \text{ Hz}$) was underestimated in simulation by 11%, although relatively fine mesh (6 mils. elements), small time step ($dt = 1e-6 \text{ s}$) and enhanced turbulence modelling (SBES model) were used.

On the other hand, interesting comparison between simulation and experiment was achieved for geometry with blunt TE. The damping ratio for lower velocities is underestimated by around 35%, but results for higher velocities ($v > 16 \text{ m/s}$) follow the measurements with error less than 5%.

The interesting fact is also, that the lock-in for hydrofoil with blunt TE was moved towards higher flow rate (somewhere between $v = 16 – 17 \text{ m/s}$) with vortex shedding frequency around $f_s = 740 \text{ Hz}$. All this was caused only by relatively small geometrical change of hydrofoil TE from one side rounded to the fully blunt. For the turbine manufacturer this fact is very important since only small change of TE geometry on the turbine blade might cause large change in vortex shedding dynamics and consequently in added parameters influencing structural performance and lifespan.

Authors would like to acknowledge effort of the workshop organizers to provide open experimental data, which are necessary prerequisite to finetune the computational methods and tools for further tackling of even more complicated FSI problems connected with hydrofoil loading by collapsing cavitation clouds and vortices [13], [28], [29].

**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.

**Nomenclature**

1-DOF One degree of freedom
$A$ Area ($\text{m}^2$)
BL Boundary Layer
CFD Computational Fluid Dynamics
DES Detached Eddy Simulation
d_{s,w,F} Damping (N s m$^{-1}$)
$F$ Force (N)
$f_{\text{air}}$ Natural frequency in air (Hz)
$f_s$ Vortex shedding frequency (Hz)
FSI Fluid Structure Interaction
$f_{\text{water}}$ Natural frequency in water (Hz)
k Turbulent kinetic energy ($\text{m}^2 \text{s}^{-2}$)
k_{s,w,F} Stiffness (N m$^{-1}$)
LES Large Eddy Simulation
$m_{s,w}$ Mass (kg)
p Pressure (Pa)
$Q$ Flow rate ($\text{m}^3 \text{s}^{-1}$)
RANS Reynolds Averaged Navier-Stokes
Re Reynolds number
SAS Scale Adaptive Simulation
SBES Stress Blended Eddy Simulation
SRS Scale-Resolving Simulation
SST Shear Stress Transport
t time (s)
TE Trailing Edge
$u$ Modal reference amplitude
$v$ Flow velocity ($\text{m s}^{-1}$)
$W_m$ Modal Work (J)
$x$ Deflection
$y^+$ Non-dimensional wall distance
$\varepsilon$ Dissipation of turbulent kinetic energy ($m^2 s^{-2}$)
$\zeta$ Damping ratio
$\tau$ Shear Stress (Pa)
$\omega$ Angular velocity (rad s$^{-1}$)

References
[1] Zouhar J, Obrovsky J, Feilhauer M and Skoták A 2016 Case study and numerical analysis of vibration and runner cracks for the Lipno I hydroelectric project IOP Conference Series: Earth and Environmental Science 49 072011
[2] Weber W, Locquenghien F, Conrad P and Koutnik J 2017 Dynamic stresses in a Francis model turbine at deep part load Journal of Fluids and Structures, 81 012014
[3] Rodriguez C G, Egusquiza E, Escaler X, Liang Q W and Avellan F 2006 Experimental investigation of added mass effects on a Francis turbine runner in still water Journal of Fluids and Structures 22 699–712
[4] De La Torre O, Escaler X, Egusquiza E and Farhat M 2013 Experimental investigation of added mass effects on a hydrofoil under cavitation conditions Journal of Fluids and Structures 39 173–187
[5] Escaler X, De La Torre O and Farhat M 2015 Review on parameters influencing the structural response of a submerged body under cavitation conditions Journal of. Physics: Conference Series 656 012150
[6] Liang Q, Rodrigue C G, Egusquiza E, Escaler X, Farhat M and Avellan F 2007 Numerical simulation of fluid added mass effect on a francis turbine runner Computers & Fluids 36 1106–1118
[7] Presas A, Valentín D, Egusquiza E, Valero C and Seidel U 2015 On the detection of natural frequencies and mode shapes of submerged rotating disk-like structures from the casing Mechanical Systems and Signal Processing 60–61 547–570
[8] Kimber M, Lonergan R and Garimella S V 2009 Experimental study of aerodynamic damping in arrays of vibrating cantilevers Journal of Fluids and Structures 25 1334–1347
[9] Yao Z, Wang F, Dreyer M and Farhat M 2014 Effect of trailing edge shape on hydrodynamic damping for a hydrofoil Journal of Fluids and Structures 51 189–198
[10] Coutu A, Seeley C, Monette C, Nennemann B and Marmont H 2012 Damping measurements in flowing water IOP Conference Series: Earth and Environmental Science 15 062060
[11] Seeley C E, Coutu A, Monette C, Nennemann B and Marmont H 2013 Determination of hydrofoil damping due to fluid structure interaction using MFC actuators 54th AIAA/ASME/ASCE/AHS/ASC Structures, Structural Dynamics, and Materials Conference 1–12
[12] Hübner B, Seidel U and 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
[13] Čupr P, Habán V and Rudolf P 2018 Numerical Investigation of Added Mass and Damping Effects on a Hydrofoil in Cavitation Tunnel Proceeding of 20. Internationales Seminar Wasserkraftanlagen (Vienna, Austria, 14–16 November 2018)
[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] Monette C, Nennemann B, Seeley C, Coutu A and Marmont H 2014 Hydro-dynamic damping theory in flowing water IOP Conference Series: Earth and Environmental Science 22 032044
ISSN 1755-1307, 1755-1315

[16] Nennemann B, Monette C and Chamberland-Lauzon J 2016 Hydrodynamic damping and stiffness prediction in Francis turbine runners using CFD IOP Conference Series: Earth and Environmental Science 49 072006

[17] Tengs E O, Bergan C W, Jakobsen K-R and Storli P T 2019 Numerical simulation of the hydrodynamic damping of a vibrating hydrofoil IOP Conference Series: Earth and Environmental Science 240 062002

[18] Nennemann B and Monette C 2019 Prediction of vibration amplitudes on hydraulic profiles under von Karman vortex excitation IOP Conference Series: Earth and Environmental Science 240 062004

[19] Bergan C W, Solemslie B W, Ostby P and 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

[20] Sagmo K F, Tengs E O, Bergan C W and Storli P T 2019 PIV measurements and CFD simulations of a hydrofoil at lock-in IOP Conference Series: Earth and Environmental Science 240 062006

[21] Fontanals A, Guardo A, Zobeiri A, Egusquiza E, Farhat M and Avellan F 2014 Boundary layer effects on the vortex shedding in a Donaldson- type hydrofoil IOP Conference Series: Earth and Environmental Science 22 032045

[22] https://www.ntnu.edu/nvks/f99-test-case3

[23] Vu C T, Gauthier M, Nennemann B, Wallimann H and Deschenes C 2014 CFD analysis of a bulb turbine and validation with measurements from the BulbT project IOP Conference Series: Earth and Environmental Science 22 0022008

[24] Štefan D, Houde S and Deschênes C 2019 Numerical investigation of flow in a runner of low-head bulb turbine and correlation with PIV and LDV measurements Journal of Fluids Engineering 141(9)

[25] Menter F R, Garbaruk A, Smirnov P, Cokljat D and Mathey F 2010 Scale adaptive simulation with artificial forcing, Progress in Hybrid RANS-LES Modelling, part of the Notes on Numerical Fluid Mechanics and Multidisciplinary Design, 111, 235-246

[26] Ansys CFX Solver Theory Guide, v 19.1, 2019

[27] Donaldson R M 1956 Hydraulic turbine runner vibration Journal of Engineering for Power 78 1141 – 1147

[28] Sedlar M, Komárek M, Rudolf P, Kozák J and Huzlík R 2015 Numerical and experimental research on unsteady cavitating flow around NACA 2412 hydrofoil IOP Conference Series: Earth and Environmental Science 72 022014

[29] Rudolf P, Štefan D, Sedlář M, Kozák J, Habán V and Huzlík R 2015 Spatio-temporal description of the cavitating flow behavior around NACA 2412 hydrofoil Journal of Physics: Conference Series 656 012168