ISSN 2176-5480

22nd International Congress of Mechanical Engineering
November 3-7, 2013, Ribeirão Preto, SP, Brazil

PARAMETRIC OPTIMIZATION OF A HIDRAULIC TURBINE STAY VANE CONSIDERING FLUID STRUCTURE INTERACTION

Renato Venturatto Junior
Escola Politécnica da USP – Av. Prof. Luciano Gualberto, Travessa 3 nº 380, Butantã, São Paulo, 05508-010, Brasil
renatovj@usp.br

Felipe Langellotti Silva
Escola Politécnica da USP – Av. Prof. Luciano Gualberto, Travessa 3 nº 380, Butantã, São Paulo, 05508-010, Brasil
felipe.langellotti@usp.br

Sandro Vatanabe
Escola Politécnica da USP – Av. Prof. Luciano Gualberto, Travessa 3 nº 380, Butantã, São Paulo, 05508-010, Brasil
sandro.vatanabe@usp.br

Rodrigo Sanomya
CESP – Companhia Energética de São Paulo – Av. N. Senhora do Sabará, 5312 – escrit. 32 CEP 04447-011/São Paulo/SP
rodrigo.sanomya@cesp.com.br

Emilio C. Nelli Silva
Escola Politécnica da USP – Av. Prof. Luciano Gualberto, Travessa 3 nº 380, Butantã, São Paulo, 05508-010, Brasil
ecnsilva@usp.br

Abstract. *Vibrations in hydraulic turbines may arise due to an inappropriate design of components such as stay vanes, guide vanes or blade runners. The objective of this study is to evaluate through optimization techniques the best format of the profile of a Francis Turbine stay vane in order to keep its natural frequencies far from the vortex shedding frequency of the flow interacting with the structure (phenomenon known as Fluid Structure Interaction – FSI). Essentially, the defined optimization problem has as objective function the minimization of the mass of the structure subjected to constraints such as stay vane natural frequency values in water. These frequencies must be out of the range of the vortex excitation frequencies resulting from the different conditions in which the turbine can operate. As design variables, the dimensions of the trailing edge and the leading edge of the stay vane shape are considered. The optimization problem is solved by using the Mode Frontier software. To obtain the natural frequencies in water, the software ANSYS Mechanical is applied by performing a modal analysis that includes the effects of surrounding fluid in the structure. Calculated resonance frequencies are compared with experimental data. At the end, Computational Fluid Dynamics (CFD) analysis is performed for the optimized result to verify its fluid-structure behavior and to make a comparison between original and optimized structural displacements. As a conclusion, the optimization method developed is able to eliminate resonances and to avoid excessive vibrations with just a few simple modifications in the original profile, as indicated by CFD simulations.*

Keywords: Hydraulic Turbines, Stay Vane, Parametric Optimization, Fluid-Structure Interaction, Finite Element Method.

1. INTRODUCTION

About 24050 MW or 34% of total hydraulic energy production in Brazil is currently generated in power plants with more than 20 years of commercial operation (EPE, 2008). In many cases, there is wear on the turbine components due to not only the effect of time, but also an inadequate design of the profile of some of these components.

In this context, the need to consider hydrodynamic loads due to the interaction with the fluid in the surroundings of those structures is increasingly important. The phenomenon of vortex shedding can occur, for example, in stay vanes with trailing edges of certain shapes. The Figure 1 shows a scheme where it is possible to see the stay vanes and its location in a hydraulic turbine. The set of stay vanes forms the stay ring, which is connected to the spiral case. The stay vane bases are fixed to the stay ring.

ISSN 2176-5480

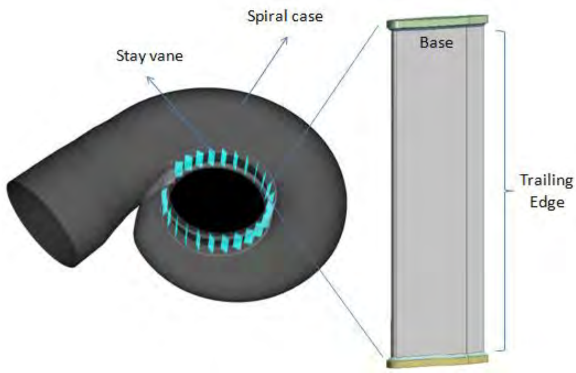

Figure 1. Spiral case and the set of stay vanes.

The focus of this work is the optimization of the stay vane profile of a Francis turbine (from Ilha Solteira Hydroelectric Power Plant). It is found that the inadequate geometry of the profile leads to dynamic problems (vibration) related to vortex shedding.

Regarding the natural frequency values in water of some turbine components, which is extremely important in the study of resonance phenomena, the work of Liang et al. (2006) shows that the effect of the inertia of the water can reduce a given natural frequency up to 40%, depending on the vibration mode, as compared to the same rotor blade in air. In a later work (Liang et al., 2007), experimental results are compared with computer simulations using the Finite Element Method (FEM). Vibration modes similar to the empirical modes are obtained, with a maximum deviation of 3.5% in natural frequencies.

The optimization of a rotor blade using genetic algorithm / CFD is discussed by Rodrigues Junior and Brazil (2009) in order to maximize the torque on the runner. The results obtained in that study are more accurate than the results obtained with methods based just on a simplified mathematical model.

The solutions currently applied to resolve the vortex induced vibrations are essentially empirical, which consists in narrowing the trailing edge by doing a chamfer by grinding and removing material. In general, it is common to reduce the trailing edge to dimensions smaller than 10 mm (Gissoni, 2005). Another widely applied procedure is to propose a geometry different from the original and evaluate vibrations experimentally with strain gauges (Kurihara et al., 2007). In this last case, this geometry is based on previous experiences.

Unlike the purely empirical approach, the present study looks for possible ways to optimize the profile of stay vanes. In other words, the objective is to parameterize the stay vane and to obtain, with the use of an optimization algorithm, what is the best geometry for regions that significantly affect the flow (such as leading and trailing edges on inlet and outlet, for example, see Figure 2).

In the formulation of the optimization problem in this work, the objective is to reduce the mass to a certain limit (i.e., make changes on the edges with a minimum interference). Adopted constraints intend to keep the natural frequencies of the structure immersed in water far from the frequency of shedding. After that, we compare the response / behavior of the original and optimized structure using CFD simulations considering fluid structure interaction (FSI hereafter - see item 3.2).

The method used here differs from the methods reported in the literature for performing optimization considering the interaction of the structure in the water, and not only the flow around rigid bodies. With these natural frequencies, the estimated frequency of vortex shedding can be used in the formulation of the problem constraints, which is based on geometric characteristics of the stay vane and the operating conditions of the turbine. CFD is used in a third and final simulation to evaluate the effect of the flow around the structure through FSI simulation, as discussed above. All the above simulations are performed using ANSYS Mechanical with CFX solver for CFD. The optimization is parametric using genetic algorithm. The software Mode Frontier performs the optimization analyses.

2. THEORETICAL FORMULATION

The first simulation performed in this work used FEM to determine the natural frequencies of the stay vanes in air. These FEM results are compared with analytical and experimental results (obtained from empirical tests at the plant). Thereafter, a similar procedure is carried out for the stay vane immersed in water. It is possible then to perform the optimization and, finally, to compare the displacements of the original and optimized structures by FSI simulation to evaluate the resonance.

In this section the focus is to present the analytical methods for calculating the natural frequencies of the stay vane in air and water. These equations are found on items 2.1 and 2.2. The FEM equations for the analyses in air and water are presented in section 2.3. Important for the optimization problem that is solved, the estimation of the vortex shedding

ISSN 2176-5480

frequency is presented in section 2.4. The computational modeling is described in section 3, and the optimization problem modeling in section 4, with results (section 5) and conclusion (section 6) following.

2.1 Natural frequencies analitycal calculation

To analytically estimate the natural frequencies of a stay vane in air, the theory of plates with reduction factor *f* was used. Considering a thin rectangular plate of constant thickness, the natural frequencies are given by (Blevins, 1995):

$$\omega_{ij} = \frac{\lambda_{ij}^{2}}{L^2}\sqrt{\frac{Eh^3}{12\rho h(1-v^2)}} \tag{2.1}$$

The value of λ_{ij}^{2} depends on the boundary conditions and the ratio L / b. For the case being modeled, the stay vane is treated as a thin plate vibrating in the thickness direction of the cross section (dimension h), being imbedded in the bases and free in the other two sides. L is the height of the stay vane (distance between bases, as shown in Figure 1) and b is the width of the cross section (Figure 2). The parameters E, ρ and v are properties of the material: elastic module, specific mass, and Poisson's ratio (see section 5.3).

In this paper, we focus the analyses on the 1st Bending Mode - Mode (11) and 1st Torsion mode - Mode (12). (Gissoni, 2005).

According to the ratio L / b of the stay vane (Figure 2), the values for λ_{ij}^{2} in Table 1 are adopted (Blevins, 1995). In order to improve the results obtained with the equation (2.1), the *f* reduction factor in the frequency is defined for a rectangular carbon steel plate:

$$f = \frac{1}{\sqrt{1-0.3434J\left(\frac{h}{L}\right)^2\lambda_{ij}^{2}}} \tag{2.2}$$

Where the constant J takes the following values:

Table 1. Values of λ_{ij}^{2} and J per mode.

Property	Mode (11)	Mode (12)
λ_{ij}^{2}	22	44
J	- 1.811	- 1.527

In order to extend the application of the theory of plates for non-rectangular sections (as the sections of stay vanes), Gissoni (2005) proposes to use equivalent bases and heights. Thus:

$$h_{eq} = \sqrt{\frac{12L}{A}} \tag{2.3}$$

$$b_{eq} = \frac{A}{h_{eq}} \tag{2.4}$$

2.2 Water reduction factor

Since the above equations are an estimate of the natural frequency in air, a useful equation to estimate the reduction of these frequencies due to the presence of the additional mass of the fluid is given by (Gissoni, 2005):

$$\frac{\omega_{água}}{\omega_{ar}} = \sqrt{\frac{m_{viga}}{m_{viga}+m_w}} \tag{2.5}$$

$$m_w = \frac{\rho_w \pi b^2}{4} \tag{2.6}$$

$$m_w = \frac{3}{8}\frac{\rho_w \pi b^2}{4} \tag{2.7}$$

ISSN 2176-5480

Where m_{viga} refers to the mass of the stay vane being modeled as a double cantilever beam while ρ_w is the density of water. The term m_w is the estimate of fluid added mass. For the 1st bending mode m_w is given by equation(2.6), and it expresses the additional mass per unit length of a thin plate vibrating in the thickness direction. Similarly, for the 1st torsion mode, the added mass is given by equation(2.7). The term b is the width of the cross section.

2.3 Finite Element Method

To numerically determine the natural frequencies of a structure immersed in water, the following equations are solved using finite element software (ANSYS, 2009b):

$$[M]\{\ddot{u}\}+[C]\{\dot{u}\}+[K]\{u\}=\{F\}\{F^{pr}\} \tag{2.8}$$

where $\{\ddot{u}\}$ is the acceleration and $\{\dot{u}\}$ the velocity related to the displacement $\{u\}$. $[M]$, $[C]$ and $[K]$ are the mass, damping and stiffness matrices, respectively.

The load vector of fluid pressure on the surface S, $\{F^{pr}\}$, is obtained by integrating the pressure on the surface area:

$$\{F^{pr}\}=\int_S \{N'\}P\{n\}d(S) \tag{2.9}$$

Where:

$\{N'\}$: shape functions used in the discretization of the displacements at each direction of the coordinate axes;

$\{n\}$: normal vector to the surface of the fluid.

More information about these equations are in reference ANSYS (2009). For details about the finite elements utilized, check section 3.1.

For the fluid, the equations that correlate flow velocity, pressure field and loads acting on structure are the momentum and continuity equations (for viscous inviscid fluid):

$$\rho\left(\frac{\partial(\{\mathrm{u}\})}{\partial \mathrm{t}}+\{u\}.\nabla\{u\}\right)=-\nabla p+\mu\nabla^2\{u\}+\{f\} \tag{2.10}$$

$$\frac{\partial(\rho)}{\partial \mathrm{t}}+\nabla.(\rho\{u\})=0 \tag{2.11}$$

where ρ is the fluid density; $\{u\}$ and p are the velocity and the pressure fields, respectively; μ is the fluid dynamic viscosity; and $\{f\}$ is the fluid body forces.

2.4 Vortex shedding frequency

One way to estimate the frequency of vortex shedding is through the dimensionless number known as Strouhal Number (S):

$$S=f_v d/V_e \tag{2.12}$$

By isolating f_v in above equation and by substituting d for the corresponding value of trailing edge thickness (δ_s) plus the average thickness of the displacement boundary layer (δ_d) ,on pressure and suction sides, we have:

$$f_v=V_e.S/(2*\delta_d+\delta_s) \tag{2.13}$$

Where:

$$\delta_d=b/138{,}8 \tag{2.14}$$

$$V_e=Q/\pi.DTi.L.cos\beta \tag{2.15}$$

ISSN 2176-5480

where Q is the discharge in the turbine operating condition, DTi is the diameter of the position where the stay vane is, L is the height of the water flow which is equal to the height of stay vane, β is the angle between the plate and a radial line passing the trailing edge, δ_s is the trailing edge stay vane thickness and b is the stay vane length in millimeters (Figure 2).

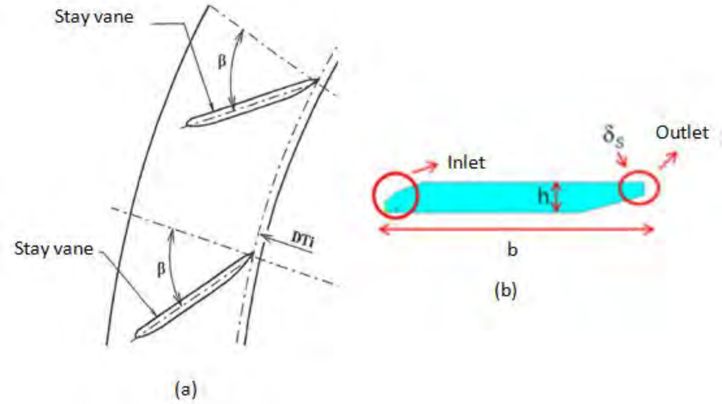

Figure 2. (a) Top view of a stay ring, with β =55 º and DTi = 9.4 m. (b) Stay vane cross section, with L=2140 mm, b=755 mm, and h=90 mm.

3. COMPUTATIONAL MODELING

The simulations in this work employed the finite element software Ansys Mechanical and CFX solver configured from the Workbench. The first simulations are modal analyses in air and then in water. These results are compared with the experimental data and the values obtained by the analytical methods for air and water. After validating the FEM results, we could estimate the vortex shedding frequency for certain operating conditions of the turbine, parameterize the structure and get the optimized structure. As a final analysis, the original and optimized structures are compared with respect to their displacement in a CFD / FSI simulation.

For the FEM case, a program in the APDL language (ANSYS Parametric Design Language) was developed, with the objective of creating the geometry and performing modal analysis in air and in water through an automatic process, allowing the coupling with the optimizer (the optimization changes the stay vane parameters at each loop, which changes natural frequencies for each case). For the CFD/FSI analysis, there is no need of such automation, since the simulation is carried out only twice (once for the original structure and once for the optimized one).

3.1 Finite Elements Utilized

The modal analysis is initially performed in ANSYS without the presence of fluid around the structure in order to know their behavior in air. The element used for modeling the structure in 2D domain is PLANE182. In the second stage, the geometry is extruded (see Figure 9) using the basic 2D code and the structure received the mesh with SOLID185 element.

For the fluid, the elements chosen are FLUID 29 and FLUID 30 (these elements are related with an acoustic analyses, see ANSYS, 2009). Similarly to the structural elements, the fluid is modeled first in 2D domain (with FLUID 29) and then extruded (using FLUID 30). Modeling the fluid around the structure with these elements becomes possible to take into account the effect of additional mass in the estimation of natural frequencies. In Figure 3, we can see the representation of the elements referred above.

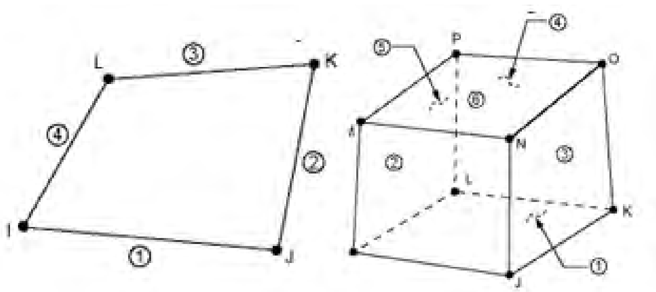

Figure 3: Finite elements utilized on structure and fluid mesh. 2D and 3D representation.

3.2 Fluid Structure Interaction

The fluid structure interaction (FSI) simulation is responsible for including the dynamic effects that the flow exerts on the stay vane. Loads (pressures) are transferred to the structural mesh in the form of forces on the knots. The structure, on the other hand, returns the displacements of its mesh to the CFD solver (which contains the flow information), restarting FSI cycle. The main goal of this analysis is to compare the original and optimized structures to verify if resonance occurs or not. This approach is used to observe the behavior of the structure due to a harmonic force applied to the structure playing the vortex role.

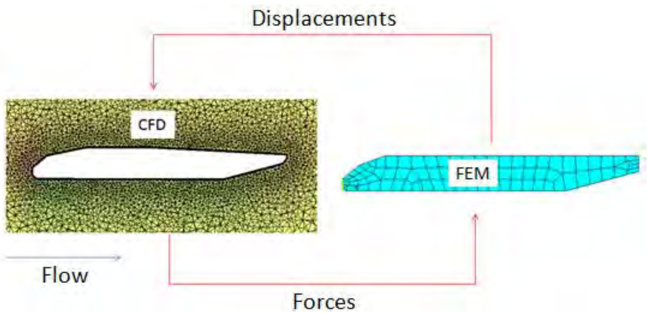

Figure 4. Interaction between two physics: structural and fluidic.

Starting from a geometry built in CAD, which encompasses not only the stay vane but also the fluid domain around, the mesh is generated by ANSYS Meshing and the boundary conditions, including the distributed harmonic force representing the vortices, are set. Subsequently, the mesh is generated for the fluid using also ANSYS Meshing.

Note that in this process it is necessary to create a layer of hexahedral elements in the region of contact between structure and fluid. This procedure permits a better discretization of the mesh in the region of the boundary layer. For this, the inflation method is applied. In this region, the maximum size of the layer is estimated with the aid of equation (2.14) and it is considered the value of 5. 5.10^{-3} m, with 8 layers and growth ratio of 1.1.

4. OPTIMIZATION

Due to the problem of vortex shedding and vibration of the stay vane turbines, methodologies for calculating natural frequencies of these structures in water were developed in the previous sections. In this section, it will be shown how that information will feed the software Mode Frontier in order to run the optimization and choose the best geometry parameters set to avoid these natural frequencies to be close to those estimated for the vortex shedding in a particular operating condition.

The optimization problem proposed is defined as follows:

Minimize M

such that

$$f_{1b} < f_s$$
$$f_{1t} > f_s$$

where:

M is mass of the structure, f_{1b} is the first bending mode, f_{1t} is the first torsion mode and f_s is the shedding frequency.

The objective here is to take the operational range out of regions where the shedding frequency is equal to the structure natural frequencies.

The optimization process flowchart of the methodology described above can be seen in Figure 5, which shows how the optimizer Mode Frontier, Matlab and Ansys relate. The optimiser, the first block to the left, generates the parameters (characteristic geometric dimensions of the part illustrated by P1 and P2) which enter the Matlab code

ISSN 2176-5480

responsible for generating the APDL script which calculates the natural frequencies in water. Thus, the output from ANSYS is read by Matlab and information of natural frequencies and mass of the structure are sent back to the optimizer that evaluates the objective function and constraints. Having optimal solutions, the problem ends. Otherwise, the process restarts.

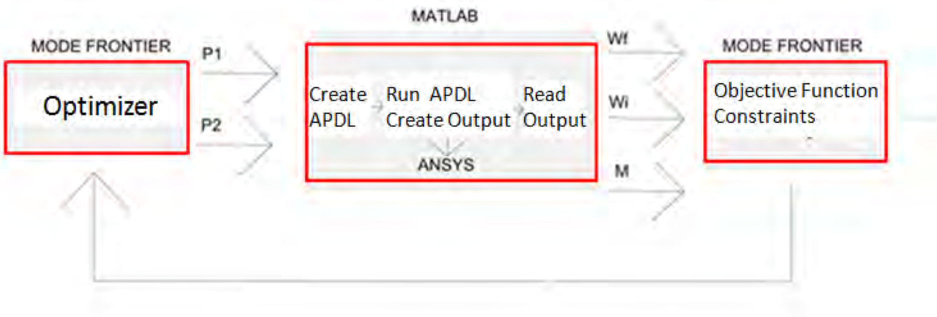

Figure 5. Computational optimization methodology flowchart.

5. RESULTS

In this section natural frequencies in water and optimization results are presented. After that, a comparison between the original and optimized structures will be shown. For the optimization to be valid, the separation of the flow must occur near the trailing edge. A CFD simulation will be presented (just for the flow, without considering FSI) to verify whether this hypothesis is valid.

5.1 Optimization problem characteristics

Using equation (2.13) for estimating the shedding frequency, we obtain the following chart, which relates the shedding frequency for different discharges with the natural frequencies of the stay vane with the original profile:

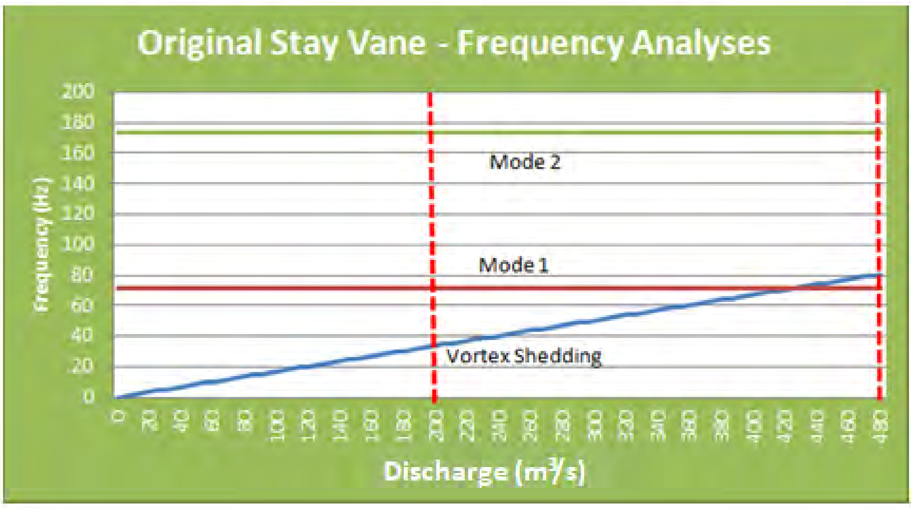

Figure 6. Shedding and natural frequencies for the original profile.

In the chart above, we have: Mode 1 - 71.2 Hz, Mode 2 - 174.2 Hz and the intersection of the curve Vortex and Mode 1 occurs for 423.4 m³ / s.

According to the hill chart below (Figure 7), we estimated a minimum operational discharge of 200 m³ / s and 480 m³ / s for the maximum discharge (this choice can vary and it depends on the turbine operation). Thus, the optimization defined in section 4 will seek for solutions that move the intersections of Vortex with Modes 1 and 2 to points outside this range of flows in which the turbine operates. As it can be seen in Figure 6, this intersection occurs for the original stay vane in 423.4 m³ / s.

ISSN 2176-5480

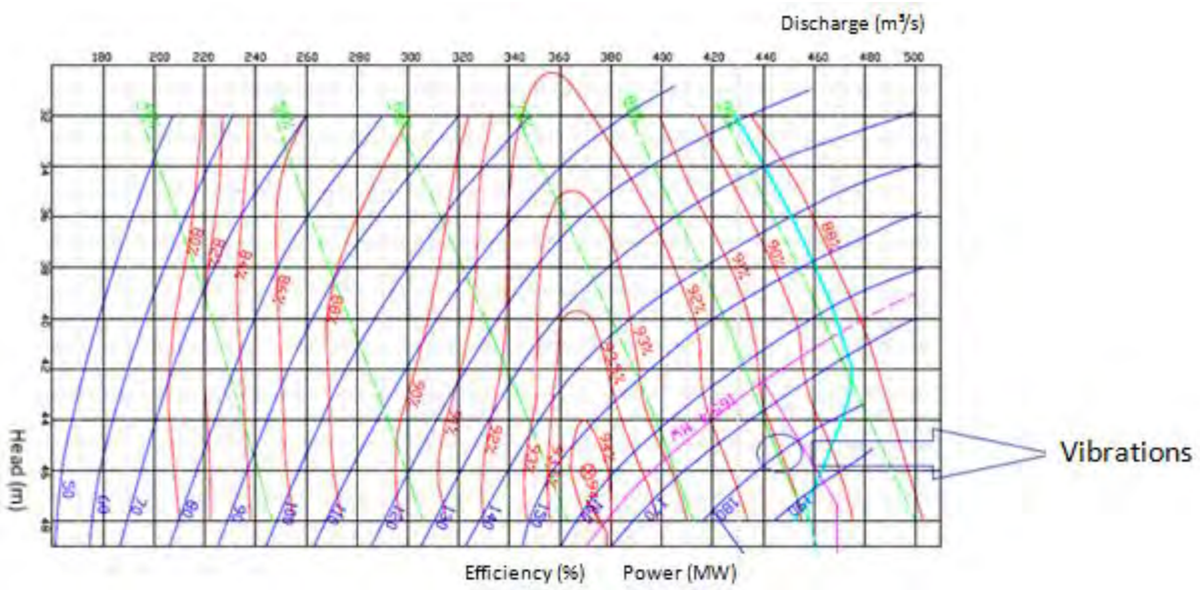

Figure 7. Hill chart for units from 05 to 20 at HPP Ilha Solteira. CESP,2012.

5.2 Mode Frontier Settings

The optimization algorithm used was the Genetic Algorithm (crossing over, selection and mutation probabilities: 50 %, 10 % and 10%, respectively) in association with the Sobol method, responsible for generating the initial population. By parameterizing the geometry with three parameters (c1, c2 and c3) the main change made is in the trailing edge (parameters c1 and c2 in the figure).To test the flexibility of the method, we inserted a third parameter c3, which contributes to the objective function (by removing part of the area of the section) and it also helps to smooth the leading edge to ensure the flow does not separate before the trailing edge. The ranges employed for each variable are: 300<c1< 400 mm; 2 <c2 <28 mm and 5 < c3<100 mm. The values of c1 and c2 help to keep – like c3 does for leading edge - the trailing edge as smooth as possible respecting the limits imposed for the original part.

Figure 8. Optimization parameters.

The parameter c3 performs an increment in the *x*-coordinate of point 1. This new point is connected with the previous point. The c1 and c2 parameters create two new points from point 2, which are then joined together to run the chamfer. The Point 2 no longer exists in the new geometry.

5.3 Natural frequencies in water

To perform modal analysis, we assumed that the sound speed in water was 1500 m/s and water density equal to 998 kg / m³. For the stay vane material (ASTM A36 Carbon Steel), the elastic modulus is 200 GPa, the specific mass is 7850 kg / m³, and the Poisson's ratio is 0.3.

The boundary conditions adopted are: cantilever on upper and lower surfaces – the bases (Figure 9a). For the fluid, we adopted zero displacement in the direction of the stay ring (z direction), and unit impedance on the boundaries of fluid domain. The unit impedance value is used for adding damping to the fluid. Thus, it simulates an infinite medium. For the mesh, a layer of fluid is built around the structure and the element FLUID 30 is set to interact with structure (making KEYOPT (2) = 0 for the element type 2). For the rest of the fluid domain FLUID 30 elements are not interacting with the structure (KEYOPT (2) = 1 for element type 3). The element size is defined according to each line of the profile, with more elements for longer lines, such as the horizontal lines in Figure 9b. The more type 3 elements are far from the type 2 elements, the more they become progressively larger.

ISSN 2176-5480

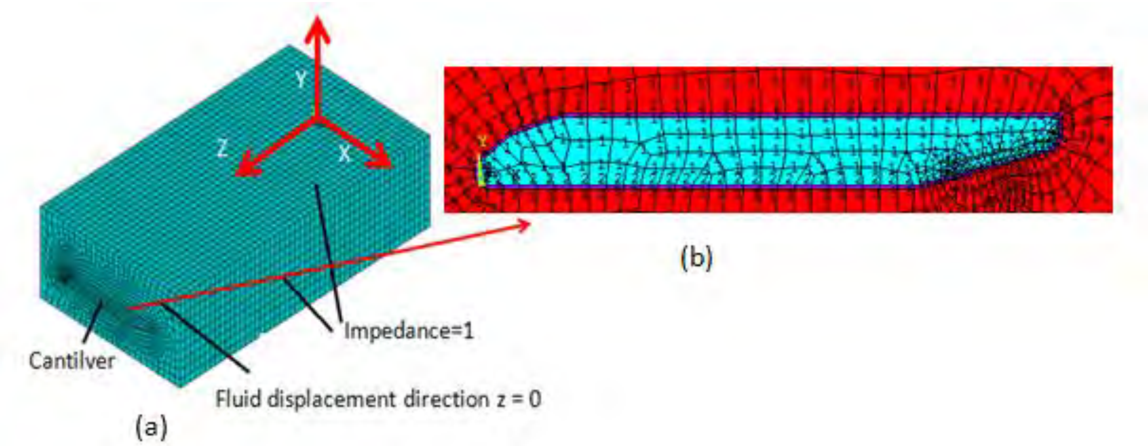

Figure 9. Boundary conditions and element types considered ("1" for structural and "2" and "3" for fluid).

The modal analysis results (air and water) are shown below. The FEA model for air is similar to the modal built for the fluid, except for type 2 and type 3 elements. The cantilever stay vane base is a common boundary condition. The first and second modes in air are shown at the top of Figure 9 (first to the left and second to the right). Below, the same goes for the first two modes in water.

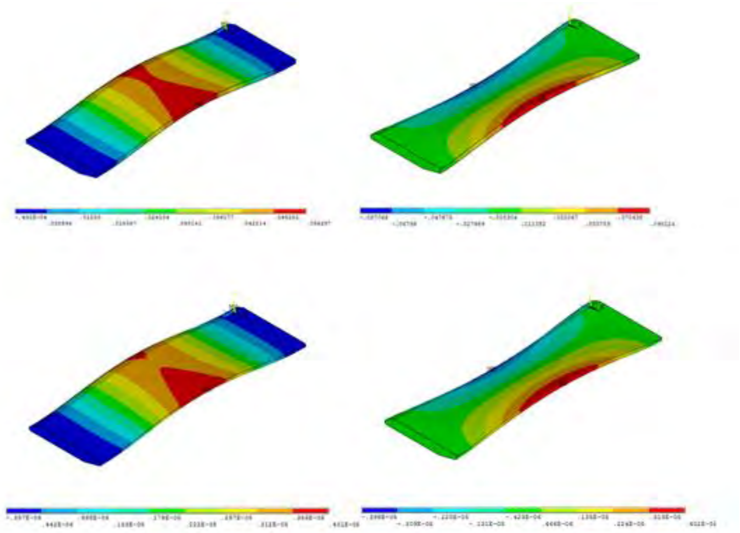

Figure 10. 1st and 2nd vibration modes in air (above) and water (below).

Table 2. Stay vane natural modes in air and water.

1st Mode air (bending)	2nd Mode air (torsion)	1st Mode water (bending)	2nd Mode water (torsion)
98.5 Hz	206.2 Hz	71.2 Hz	174.1 Hz

The empirical, numerical and analytical results are summarized in Table 3:

Table 3: Results summary.

	Empirical Freq. in Air [Hz]	Empirical Freq. in Water [Hz]	FEA Freq. in Air [Hz]	FEA Freq. in Water [Hz]	Analytical Freq. in Air [Hz]	Analytical Freq. in Water [Hz]
1st Mode	93.8	67.8	98.5	71.2	99.3	71.3
2nd Mode	207.0	181.0	206.2	174.1	191.9	165.0

Comparison of reduction ratios for resonance frequencies in Water / Air for analytical, empirical and FEM methods can be seen in Table 4:

Table 4: Ratio of natural frequency values in water and air for the stay vane 12 of UG 19 from HPP Ilha Solteira.

	Empirical	FEM	Analytical
	Water/Air	Water/Air	Water/Air
1st Mode	72.3 %	72.3 %	71.8 %
2nd Mode	87.4 %	84.4 %	86.0 %

Analytical results are calculated by using the theory of thin plates and the method of the added mass. Source of experimental results: CESP , 2012.

5.4 Optimization

The optimization problem defined in section 4 is solved with Mode Frontier, which offers several optimal solutions. The combination of the parameters c1, c2 and c3 for each solution corresponds to a given set of natural modes. The best choice depends primarily on the conditions the turbine operates, leaving it to the operator of the plant to choose.

The values chosen are: c1 = 400 mm, c2 = 8 mm and c3 = 49 mm. The figure below shows the geometry of the optimized stay vane.

Figure 11. Optimized stay vane.

5.5 CFD analyses

A test before the implementation of FSI simulation consists of performing CFD analysis in steady state. The main objective is to verify the separation of the flow in the region of the trailing edge, because only under these conditions the equation (2.13) is valid.

In Figure 12 a detail of the velocity field around the stay vane profile for original and optimized situations can be seen.

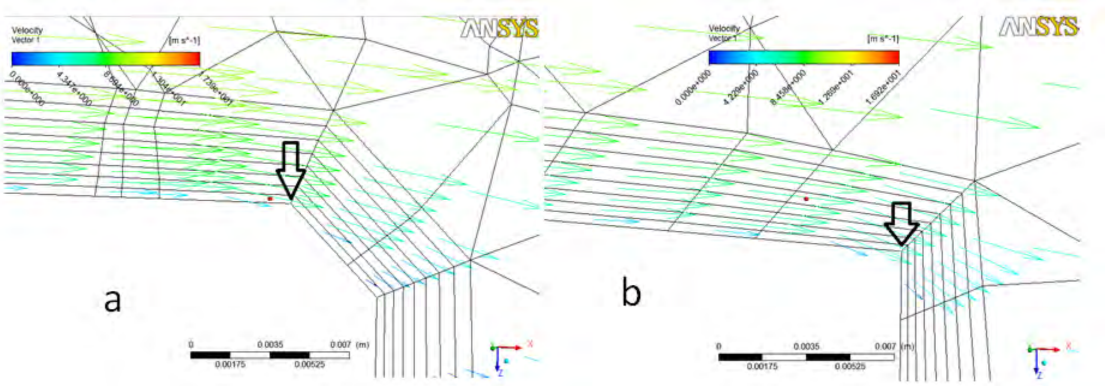

Figure 12. Separation of the flow around the stay vane. a) Original b) Optimized.

The arrows indicate the point where the velocity vector begins to have their module reduced, as is expected when the pressure gradient in the direction of the flow begins to be positive (this is a necessary condition for the flow separation, provoking flow reversion. For more information, see the work of Neto, 2007). The point indicated by the arrow in the Figure 12a is the beginning of the fillet that connects to the trailing edge. Similarly, for the optimized structure (Figure 12b), there is a separation in the region of trailing edge also for the optimized structure.

Being valid the estimation of vortex shedding frequency, the graph in the figure below can also be considered valid. It can be confirmed that the objective of the optimization is achieved: the first intersection is 193.483 m³ / s (less than

ISSN 2176-5480

200 m³ / s, as imposed) and the second intersection is 482.668 m³ / s (more than 480 m³ / s, as imposed), thus out of the range in which the turbine operates. Mode 1 is at 67 Hz, while the second mode is at 167.14 Hz.

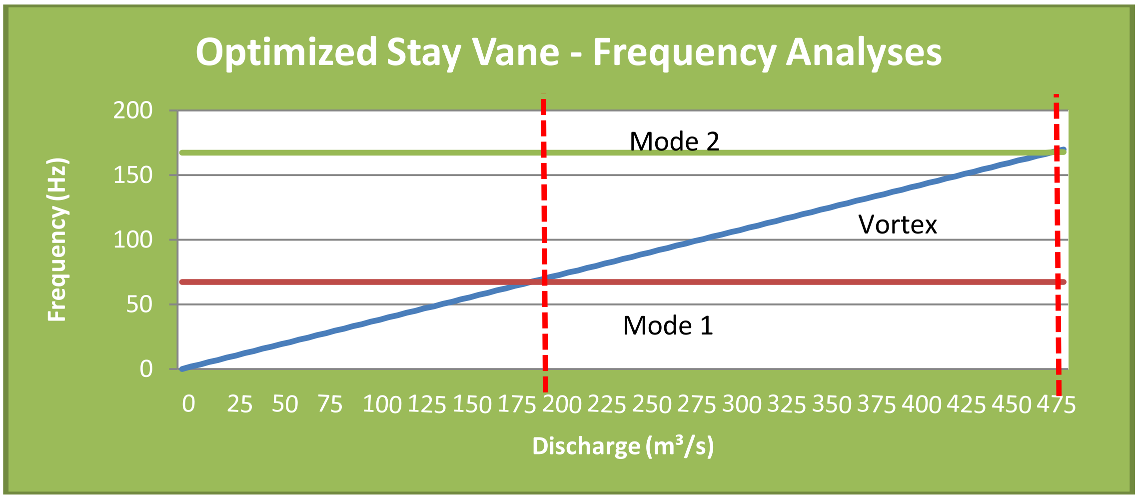

Figure 13. Frequency characteristics for optimized stay vane.

5.6 Transient Fluid Structure Interaction

We performed a transient simulation with 0.12 s total time. We used as excitation force a sine wave distributed along an edge of geometry and in the same frequency as the 1st natural frequency of the first mode for original stay vane. To reconstruct the wave, we used the criterion of a sampling frequency 10 times higher than that of interest (Mode 1 to 71.2 Hz). We adopted a force magnitude of 5000N, which is sufficient amplitude to cause displacements in the same order of magnitude as the experimentally measured for the stay vane in resonance, 10^{-4} m. For the transient simulation is necessary to adopt a value β representing the damping associated with the damping matrix, according to the reference ANSYS (2009b). The value used satisfies the relationship $\beta = 2\zeta / \omega_i$, and = 2.5% (value as Gissoni, 2005), and f is the frequency of Mode 1 in Hertz. The chart below shows the comparison of the original structure (above) with the optimized structure (below). The maximum Total Mesh Displacement (TMD) in the interface region between fluid and stay vane and in the direction of application of the forces representing the vortex is plotted.

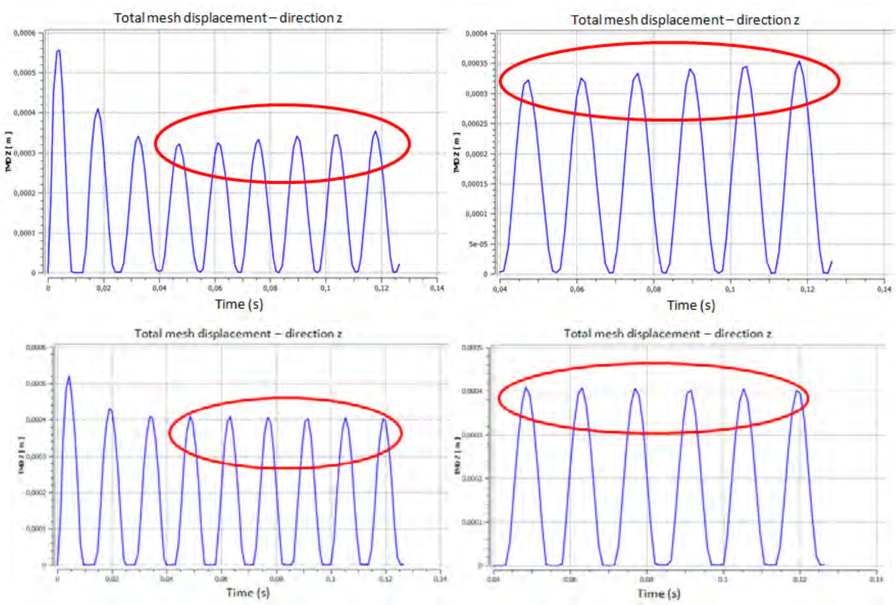

Figure 14. Maximum Total Mesh Displacement in Z direction (original structure above and optimized below). Highlighted in red from 0.04 s.

Clearly, when observing Figure 14, it is possible to note the stabilization of displacements after the instant of impact (maximum deformation at the beginning of the simulation - when the water hits the structure). In contrast, the displacements in the original structure always grow from a given instant. This is a good indication that the optimized stay vane is not in resonance, as expected.

6. CONCLUSION

The optimization methodology is successfully developed. It is worth noting the possibility that the designer can parameterize the structure as he desires. Changing leading/trailing edges and making chamfers is just one of the possible ways.

Likewise, simulations made in order to obtain natural frequencies in water gave results with minimal deviations with respect to experimental and analytical methods (approximately 5%).

The transient FSI simulation proved to be useful for comparing the original and optimized structures, since it is possible to observe the occurrence of resonance in the former, unlike what occurred in the latter. Furthermore, the isolated CFD analysis - which is the CFD analysis separated from structural - assisted in checking the flow separation in the region of the trailing edge, which is a prerequisite for the analytical vortex shedding frequency estimation to be valid.

It would be interesting to find ways to engage the flow separation information (point on the stay vane where it occurs) on the optimization methodology in future work, making the method more robust.

7. ACKNOWLEDGMENTS

The last author is thankful for the financial support received from CNPq, n.o 303689/2009-9.

8. REFERENCES

ANSYS. “Theory Reference for the Mechanical APDL and Mechanical Applications”. Release 12.0. [S.l.]: Ansys Inc., 2009b.

BLEVINS, R.D. “Formulas for natural frequency and mode shape”. Malabar:Krieger Publishing Company, re-edition 1995.

CESP. Personal communication with Rodrigo Sanomya, CESP Engineer, April 10 th, 2012.

EPE. Nota Técnica DEN. “Considerações sobre repotenciação e modernização de usinas hidrelétricas”.Brasilia, 2008.

GISSONI, H.C. “Análise mecânica de vibrações em travessas do pré-distribuidor de turbinas hidráulicas”, Dissertação (Mestrado). São Paulo: EPUSP, 2005.

KURIHARA, F., H. POLL, D. DIAS, e H. GISSONI. “Diagnóstico de Máquinas para a eliminação da origem de trincas em pré-distribuidores:a otimização da UHE Capivara.” XIX SNPTEE. Rio de Janeiro, 2007.

LIANG, Q.W., C.G. RODRÍGUEZ, E. EGUSQUIZA, X. ESCALER, M. FARHAT, e F. AVELLAN. “Numerical simulation of fluid added mass effect on a francis turbine runner.” Computers & Fluids, 6 de 07 de 2007: 1106-118.

LIANG, Q.W., C.G. RODRÍGUEZ, E. EGUSQUIZA, X. ESCALER, M. FARHAT, e F.. AVELLAN. “Experimental investigation of added mass effects on a Francis turbine runner in still water.” Journal of Fluids and Structures, 07 de 2006: 699-712.

NETO, A. D'A. “Estudo de Vibrações induzidas por escoamento em pré-distribuidores de turbinas hidráulicas utilizando CFD”. São Paulo: Trabalho de Formatura - Escola Politécnica da Universidade de São Paulo - Departamento de Engenahria Mecânica, 2007.

RODRIGUES, A.P.S.P, e A.C.P BRASIL JUNIOR. “Numerical Simulation of the Optimized Hydraulic Turbine Blades.” 20th International Congress of Mechanical Engineering, Gramado - RS, 2009.

9. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.