Development of a Methodology for Evaluation of a Structural Damage in Turbine Blades from Hydropower Generators

Edvaldo Sobrinho, edivaldo.silva@cesp.com.br
Rodrigo Sanomya, rodrigo.sanomya@cesp.com.br
Rui Ueda, rui.ueda@cesp.com.br
Hamilton Tiba, hamilton.tiba@cesp.com.br
Companhia Energética do Estado de São Paulo

Marcos de Sales Guerra Tsuzuki, mtsuzuki@usp.br
Julio Cezar Adamowski, jcadamow@usp.br
Emilio Carlos Nelli Silva, ecnsilva@usp.br
Ronny Calixto Carbonari, ronny@usp.br
Flavio Buiochi, fbuiochi@usp.br
Escola Politénica da Universidade de São Paulo

Abstract. *It is very important that the turbine blade hydraulic profile must be very close to the hydraulic profile defined during the design and adjusted during the tests with the experimental model, because this fact minimizes the effects of cavitation and vortex. In this work it is studied a viable alternative to solve such problems by using a measuring equipment with computerized software for surface reconstruction, which will create a digitized model of the current status of the blade, allowing the comparison with the mathematical model of the original surface. However, the original geometry is not always available and it must be recreated. Thus, as an alternative, a blade with few damages is selected and a smoothing algorithm is applied to determine the geometry. The proposed smoothing algorithm uses a probabilistic heuristic named simulated annealing. The smoothing algorithm searched for curves that approximate the measured points. The desired curve must be very close to the measured points and must have minimal length. This work develops a structural evaluation methodology and lifetime estimation for a hydrogenerator component using the ANSYS software. The main objective is to identify load regions and critical parameters to be inspected in the component. It predicts the mechanical components structural behavior subjected to different load conditions and the repair influence on the lifetime, in order to optimize the maintenance schedule, increasing its reliability by reducing the need of experimental tests. Optimization techniques are applied to evaluate the turbine wear. Some results are shown with data obtained from Jaguari Hydroelectric power station.*

Keywords: *CAE, curve optimization, reverse engineering, blade profile reconstruction, dynamic analysis, finite element*

1. INTRODUCTION

The cavitation affects the blades of the power generation units turbine of Ilha Solteira hydroelectric plant, among others. In the case of Ilha Solteira, during cavitation repair, it is deposited an average of 3.000 kg of weld material and the control of the profile is performed manually by using templates. This methodology of dimensional control is slow and imprecise. The imprecision in the dimensional control is very hazardous to the power generation. Currently, a viable alternative to solve such problem is the use of a measurement equipment with computer software for surface reconstruction, which will create a digitized model of the current geometry of the blade, allowing the comparison with the mathematical model of the original surface. In addition, an accurate measurement of the surface of the blade can be used to check the amount of material needed to complete the gap with welding. Finally, one can check the quality of repairs, by applying the methodology again.

However, it is common the fact that the original model of the geometry of surfaces is not available. An alternative to achieve a geometry of the designed surface is to make measurements of a turbine blade that is only slightly damaged by using a computerized measuring equipment and software for reconstruction. Problems of vibration, cracks and breakage of components of generating units (mainly in hydroelectric plants built prior to the 80s) have required a reassessment of the various structural elements of machines to maintain the security, reliability and availability of the equipment. It is known that the structural model through the finite element method is a powerful simulation tool for performance evaluation of the influence of wear and repair of the component during its lifetime (Wikström, 1997; Rodriguez et al., 2006; Richter, 2003; Dubas and Schch, 1987; Liang et al., 2007).

In this work, it is developed a methodology to recreate the mathematical model of the designed surface of a turbine blade, to make the structural evaluation and to estimate the lifetime for a component of power generators using finite element, with the objective of identifying regions, loads and critical parameters to be inspected in the component. This will assess and predict the structural behavior of mechanical components subjected to various loading conditions, and determine the influence of repairs during the lifetime in order to optimize the gaps between maintenances by increasing its reliability because it reduces the need for experimental tests.

The implementation of this methodology allows the rapid achievement of results and the possibility to test various options of material and repair geometry. The best alternative that meets the conditions of maintenance can be chosen, providing significant reductions in cost and time. The use of computer simulation to optimize the maintenance of the components is an under explored area of research in literature, since most work is directed to the area of design of power generators. Therefore, the study and development of work in this area bring significant contributions to the area of power generators. The software ANSYS has been used to perform the computer finite element simulations (ANSYS Tutorials, 2007).

The methodology proposed in this work follows the sequence: a) construction of CAD solid model of the power generators component from data obtained with a machine for measuring coordinates; b) export the CAD model to CAE (Computer Aided Engineering) model by discretizing it using solid elements (3D), c) to perform a linear static finite element analysis considering the loads (pressure, centrifugal force, weight and reaction forces) d) assess the stiffness of the component and critical points where the stress concentration occurs, e) evaluate the lifetime of the component (fatigue), f) sensitivity analysis of geometrical characteristics at critical points g) use of optimization techniques to evaluate the material wear limit.

2. CAD MODEL CREATION

The CAD model is created through several steps: turbine blade measurement, curve optimization and surface reconstruction. The measured points are obtained using a Sigma Portable Arm from Romer. The space available is small and several ways to prepare the measurement are studied. The measured points represent curves that are drawn on the turbine surface. The representation of those curves are obtained using the optimization method Simulated Annealing. Finally, the surface is reconstructed from the determined curves using Irit (Elber, 2001), a free CAD software.

2.1 MEASUREMENT PREPARATION

Several ways to prepare the measurement are studied: use of a flexible wire network with no stiffness, use of a flexible network with low stiffness and use of lines drawn on the turbine blade. The use of artifacts in the measurement is intended to reduce the measurement time. The acquired points are defined by the following rules:

- Divide the blade surface in curves;
- Divide the curve into a set of points.

Each point is labeled by tags, facilitating their identification during the measurement. The problems found are the following:

- The sequence of points should follow the geometry of the blade, i.e., regions of large curvature should have a larger number of points;
- Most of the points is not located exactly on the mesh, and the acquisition should be performed taking into account some visual approximations and access difficulties for positioning the equipment.

The best method for the acquisition of blade points is drawing a line with chalk which approximates the blade geometry (see Fig. 1).

2.2 PROCESSING OF ACQUIRED POINTS

The measured points are read by a CAD system. Fig. 2.(a) displays a set of points for pressure and suction surfaces. The lack of physical space to reach the point with the equipment measuring arm, made impossible to acquire all points at their drawn position. The modification of the acquired position created tensions on the curve (see Fig. 2.(b)). The tension does not exist on the designed or measured surface, but it comes as a consequence of the difficulty in correctly positioning the equipment arm during the measurement.

This difficulty is inherent to the measurement process, so a proposal is studied to determine the control points of a B-Spline curve (Bartels et al., 1986), aiming the determination of a curve that approximates the measured points and that has no tension. The fact that the measured curve is three-dimensional (i.e., there is no plane that contains the measured curve) makes the manual procedure adjustment impossible.

2.2.1 Simulated Annealing

The simulated annealing (Kirkpatrick et al., 1983) is a probabilistic meta–heuristic algorithm used in this work, which presents local exploration. At each iteration only one solution of the problem is analyzed. The cost of this solution is


(a) (b)

Figure 1. (a) Engineer is performing the measurements in a turbine blade. (b) The measuring equipment.

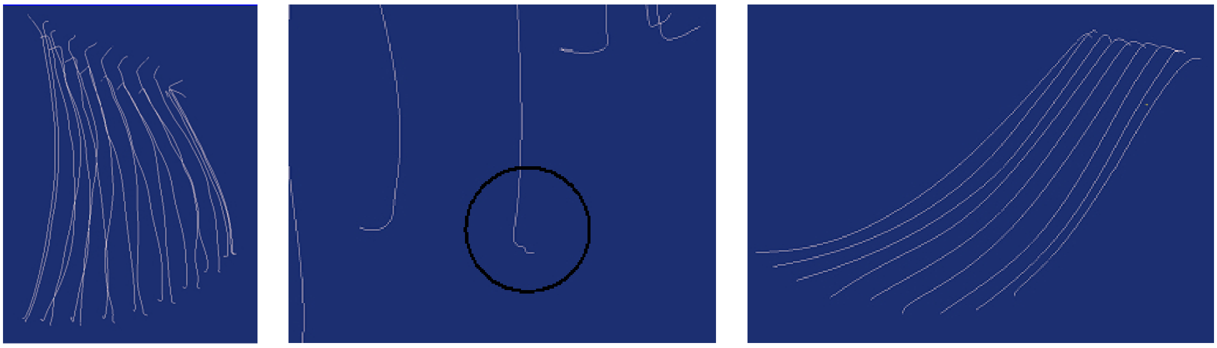

(a) (b) (c)

Figure 2. (a) Set of points acquired in the suction and pressure surfaces. (b) Defect representing a tension in the curve. (c) Final optimized curves displayed in three dimensional space.

compared with the cost of the preceding iteration solution, and based on the heuristic rules, a new solution is generated for the next iteration.

2.2.2 Function to be Minimized

The objective is to find a curve with a reduced number of points of control that approximates a set of measured points. The objective is represented by a cost function that sums the distance of the measured points to the curve. However, there are many curves that satisfy the condition of minimum distance. The cost function is improved by adding the length of the curve. Thus, the function should minimize two factors: distance to the measured points and length.

The results show that there are two forces to be controlled during the optimization process: interpolation and smoothing forces. The interpolation force objective is to reduce the distance of the measured points to the curve, and the smoothing force reduces the total length of the curve. When one tries to make the curve smoother, the curve may diverge from some measured points. When trying to improve the interpolation, regions with tension can appear.

Figure 2.(c) displays the final result with all the optimized curves representing a pressure surface. The improvement obtained can be understood when comparing Fig. 2.(c) with the curves shown in Figs. 2.(a) and 2.(b).

2.2.3 Surface Creation from Curves

Since all curves are determined, one can now create a surface that passes through these curves. Fig. 3.(a) shows the pressure and suction surfaces created by the interpolation of optimized curves obtained by simulated annealing. Once the surface is determined, it is possible to reconstruct the blade by assuming rotational symmetry around an axis (see Fig. 3.(b)). Information on the axis of symmetry is also acquired by the measurement system.

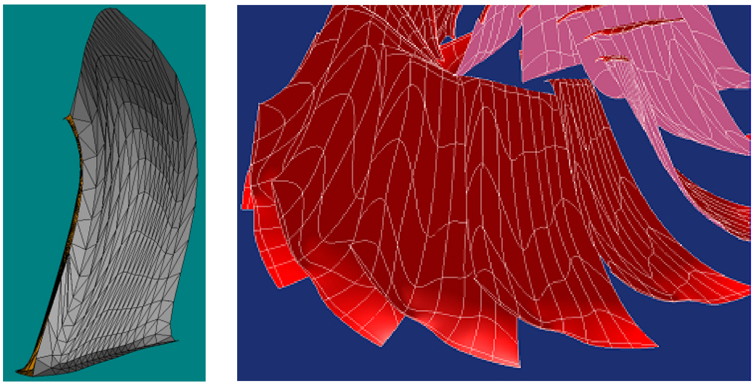

(a) (b)

Figure 3. (a) Surface created from the optimized curves. (b) Surfaces rotated along the axis of rotational symmetry.

2.3 COMPARISON WITH THE CALCULATED SURFACE

The surface shown in Fig. 3.(b) has a geometry very close to the originally designed surface. This blade is chosen with care, because its geometry is used as a standard. However, a blade with a suitable surface is not always available. In this case, the curves to be measured should be acquired over a region with acceptable geometry, and the points should be defined, as shown in Fig. 1.

Once the surfaces are obtained, the calculated surface can be used in the measuring system and the distance between a measured point and the calculated surface can be determined.

3. FEM FLUID FLOW SIMULATION

A hydropower generator component is chosen from a Francis type turbine from hydroelectric power plant of Jaguari. As a first step of the computational modeling, the CAD model is built. Manipulating the CAD model of the blade, the band and the crown of the turbine are added. The final CAD model of the turbine is shown in Fig. 4.(a) together with the CAD model of a section of the turbine (Fig. 4.(b)).

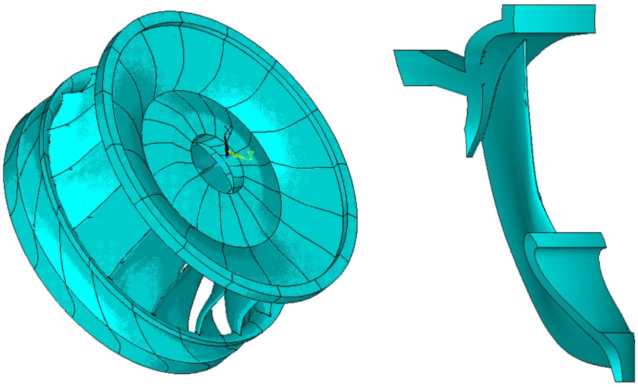

(a) (b)

Figure 4. (a) Final CAD model of the turbine; (b) CAD model of a section of the turbine of the HEU Jaguari (Hydroeletric Unit, dimensions: height $= 0.97m$ and diameter $= 1.85m$).

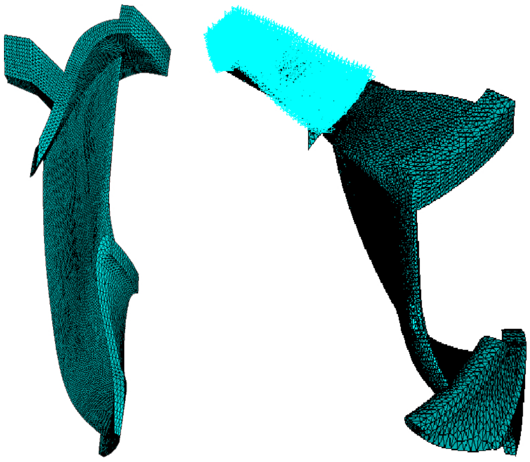
To perform the structural simulation it is necessary to know the pressure loads that act on the turbine blade. This load is obtained through a fluid flow simulation (Nilsson and Davidson, 2003) using the software CFX (ANSYS CFX Tutorials, 2007). However, the CAD volume of liquid that is defined between two adjacent blades of the turbine must be generated first. Then, this model is discretized into a finite element mesh using the ANSYS software (ANSYS Tutorials, 2007). For the fluid flow simulation, the critical state of operation of the turbine, or a maximum drop of $66.6m$ ($\sim 6.87atm$) is considered, which results in a turbine pressure of $7.87atm$. The angular velocity is considered equal to $31.4rad/s$ ($= 300rpm$ or $5Hz$) and the acceleration of gravity equal to $(9.81m/s^2)$.

The calculated power and flow rate are equal to $\sim 11.8MW$ ($\sim 15,850hp$) and $42.3m^3/s$, respectively. These results are consistent with data from the level curves of HEU Jaguari. In all structural simulations presented in this article, the same boundary conditions are considered in the fluid flow simulation to lift the pressure loads in the blade.

4. CAE MODEL CONSTRUCTION

This phase includes the finite elements mesh construction and specification of boundary conditions and mechanical loads. Although the turbine has 16 blades, a CAD model of one turbine section with an angle of 22.5^o is used, as shown in Fig. 4.(b). To model the complete turbine using only one section, it is necessary to use the command "cyclic" in the ANSYS (ANSYS Tutorials, 2007), which considers a cyclic symmetry for the model. However, to use the command "cyclic" it is necessary to generate an identical mesh on opposite surfaces of the cyclical section model, i.e., the same number of nodes in the same positions of the equivalent cyclical surfaces. Generated a mesh that meets the cyclical condition, the ANSYS automatically recognizes the other sectors and expands the model to the complete model, as shown in Fig. 4.(a).

For discretization of the model, it is used the SOLID95 ANSYS element. The mesh obtained, shown in Fig. 5.(a), has $210,936$ elements and $157,885$ nodes. Observe that the constructed mesh is uniform throughout the model to ensure a more accurate numerical analysis, especially for the calculation of mechanical stresses. The material considered is the steel and the specified mechanical properties are the Young's modulus, Poisson's coefficient, and density equal to $207GPa$, 0.3, and $7800kg/m^3$, respectively.



(a) (b)

Figure 5. (a) Mesh generated for the cyclical section; (b) Mechanical boundary conditions.

After the generation of cyclic mesh, the next step is the application of mechanical displacement and loads boundary conditions. The displacement components Ux, Uy, and Uz for the regions corresponding to the crown are set up, as shown in Fig. 5.(b). The mechanical load is obtained from the CFX as a pressure field applied only to the blade nodes, as shown in Fig. 6. SURF154 (ANSYS Tutorials (2007)) should be used to transmit the load of pressure to the structure and care should be taken to specify a density close to zero for to this element. The maximum intensity of pressure is $1.65MPa$, and occurs at the blade entry edge.

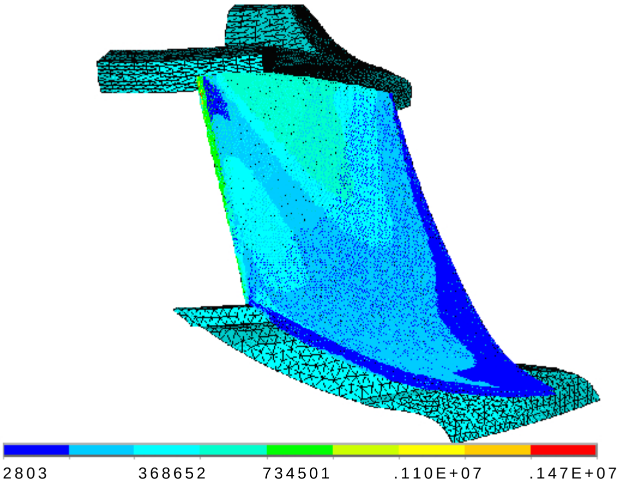


Figure 6. Pressure load applied to the blade (units are in Pa).

The other mechanical loads result from inertia due to angular velocity and from its own weight. The next step is to perform the static structural analysis.

5. FEM STRUCTURAL SIMULATION

Numerical results are presented for mechanical stresses and displacements considering the static analysis.

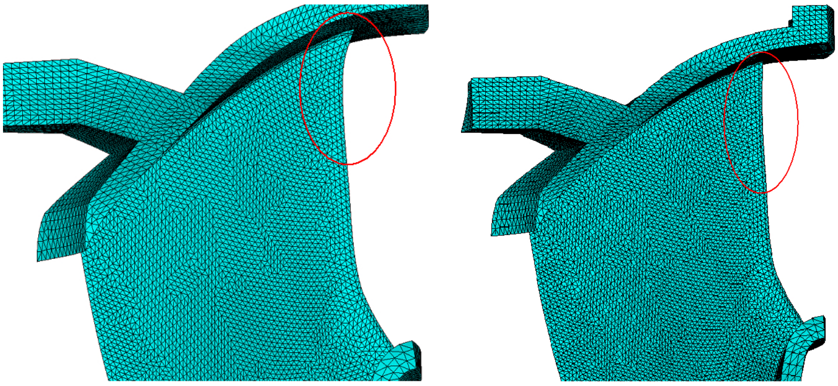
5.1 Static Analysis

The simulations presented below intend to examine the entry edge radius of curvature (between blade and crown) influence and the presence of holes generated in the blade due to cavitation. This way, the distribution of stresses and displacements in that region and its influence on the lifetime of the turbine will be determined. Only von Mises stress results are presented below. The lifetime analysis is presented on the fatigue section.

5.1.1 Analysis of the Entry Edge Radius of Curvature between Blade and Crown

Figure 6 shows the geometric differences between the two models (with and without the radius of curvature). In both models, the same conditions of loading and boundary are considered. However, for each model presented, a new field of pressure must be generated using the CFX. The model with rounding radius has a pressure field larger than that of the original model, due to the flow change at the entry edge. The resulting displacement modulus is shown in Figs. 8.(a) and 8.(b) for the models of Figs. 7.(a) and 7.(b), respectively.

Analyzing the displacement results, it appears that the rounding radius caused a small increase in the intensity of displacement. However, one can notice that the rounding radius caused changes in the turbine bending, as shown in Fig. 8. Thus, the rounding radius increases the intensity of displacement in the crown entry area, exactly the region affected by the rounding. It would be interesting to conduct the same study for the rounding radius at the output.



(a) (b)

Figure 7. Comparison between FEM models: (a) with the entry edge radius between blade and crown, (b) without rounding.

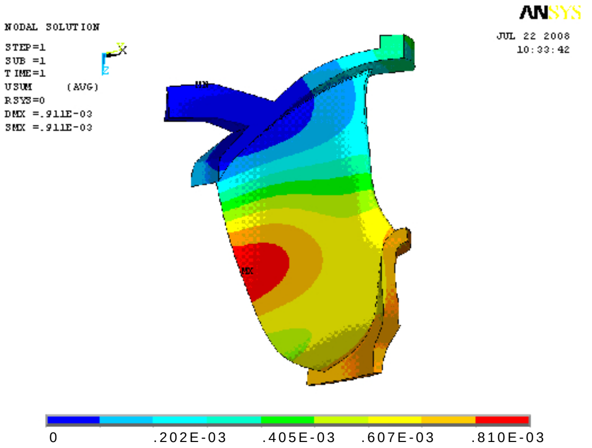
Figure 9 shows the von Mises mechanical stresses. The stress concentrates in the interface regions (blade–crown and blade–band) and the output edge, as shown in Figs. 8.(a) and 8.(b) for the models with and without rounding radius, respectively. Table 1 shows a comparison of minimum displacement and maximum von Mises stress from simulations with both models.

Table 1. Comparison of displacement and maximum von Mises stress with respect to models of Figs. 7.(a) and 7.(b)

	Maximum Displacement (mm)	Maximum von Mises stress (MPa)
Without rounding radius	0.62	146
With rounding radius I	0.91	177

5.1.2 Analysis of the Cavitation Effect Influence in Mechanical Stresses

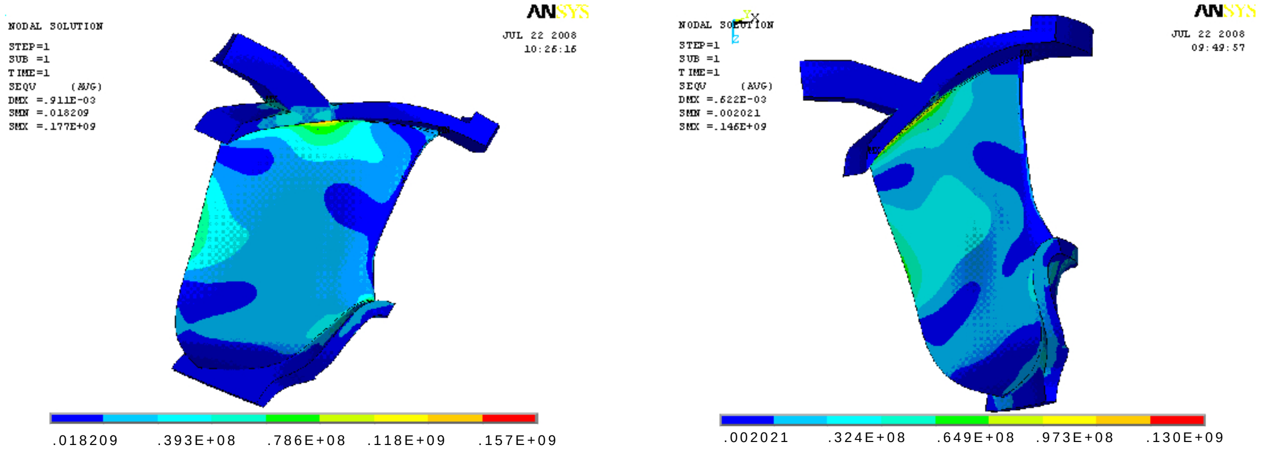
The influence of the cavitation effect in the stress concentration of blade is discussed. In this modeling, elements in the cavitation subjected region are removed to simulate the absence of material, as shown in Fig. 10.(a).



(a) (b)

Figure 8. Displacement modulus. (a) Model with the entry edge radius between blade and crown, (b) without rounding (units in m).

Boundary conditions (load and constraints of mechanical displacement) are kept equal to the previous simulations, and the characteristics of the original model remains (without changes in the radius curvature). The resulting intensity of mechanical displacement and von Mises stress are shown in Figs 11.(a) and 11.(b), respectively.



(a) (b)

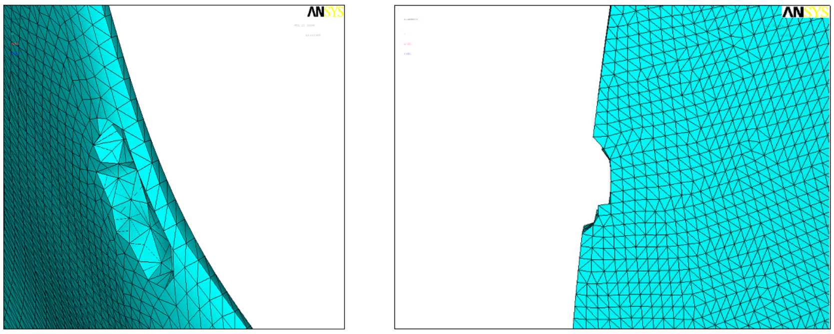
Figure 9. von Mises Mechanical stresses. (a) Model with the entry edge radius between blade and crown, (b) without rounding (units in Pa).

As shown in Fig. 11.(b) there is an increase in mechanical stresses in the cavity. However, as the region analyzed is not located at the interface between blade and crown or blade and band, the stress increase is not strong. Although, the cavitation region is critical in real cases, and may have higher removal of material, in this numerical modeling, the amount of material removed is not enough to alter significantly the mechanical stresses. Thus, a second model is built where a larger amount of material is removed, as shown in Fig. 10.(b).

In Fig. 12.(a) the displacements for this second model are shown. These results show that the turbine is rigid enough to withstand loss of material, even in critical regions. The maximum von Mises mechanical stress is increased by $10MPa$, as shown in Fig. 12.(b), creating artificial cavitation in a region of high stresses. However, this model causes changes in the flow, whose influence can be precisely obtained by examining the new flow, which has not been conducted due to the project focus. Table 2 shows a comparison of displacement and maximum von Mises stress with the simulation results of the previous models.

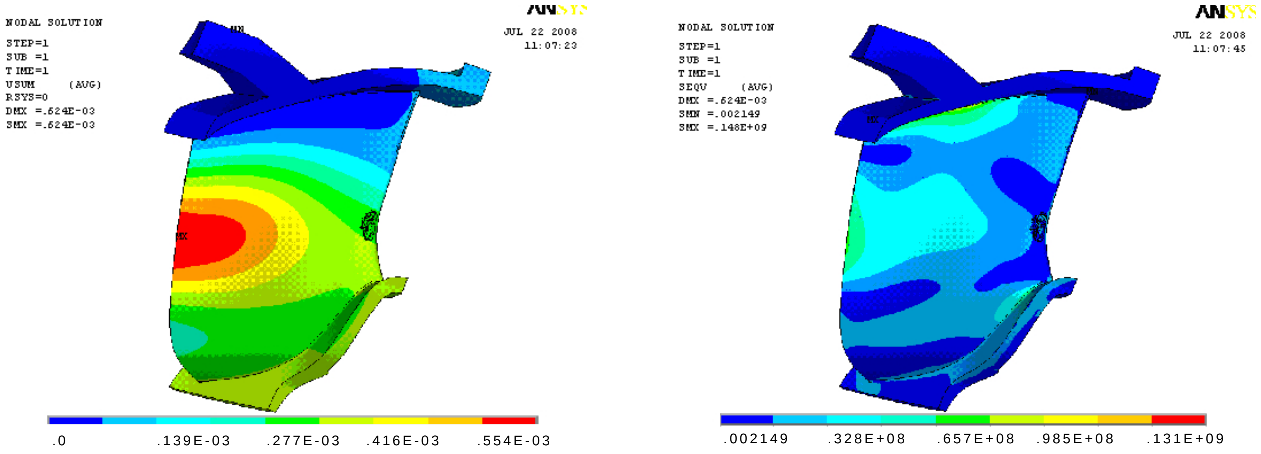
5.1.3 Fatigue Analysis

To estimate the fatigue lifetime, the graphic of Fig. 13 and the von Mises mechanical stress distribution obtained in the previous section (Shigley, 1984) are used. Comparing the influence of the entry curvature edge radius, with the original



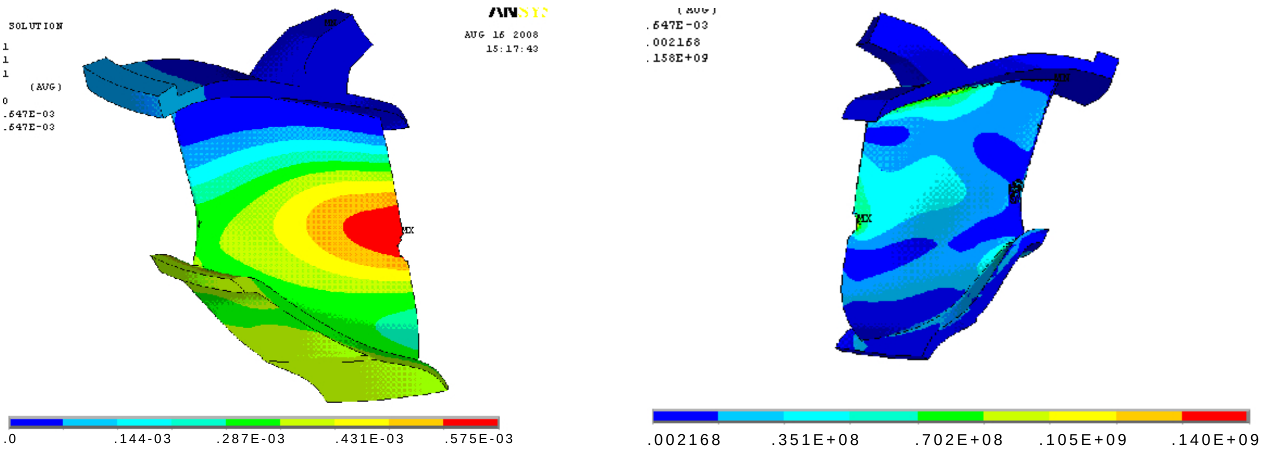
(a) (b)

Figure 10. (a) Blade cavitation near the entry edge of the suction surface, (b) Detail of the second model finite element mesh of the cavitated blade near the output edge.



(a) (b)

Figure 11. (a) Displacements modulus in the model with cavitated region (units in m), (b) Mechanical von Mises stress on the cavitated model I (units in Pa).



(a) (b)

Figure 12. (a) Displacement modulus in the second model of blade cavitation (units in m), (b) mechanical von Mises stresses in the second model of blade cavitation (units in Pa).

model (without radius) has a larger lifetime than the case with rounding radius. The values of lifetime are 6 × 105 and 2 × 105 cycles calculated for tensions of 146 and 177 MPa, respectively. For the models in which it the influence of the

Table 2. Comparison of displacement and maximum von Mises stress with respect to models of Figs. 10.(a) and 10.(b)

	Maximum Displacement (mm)	von Mises maximum stress (MPa)
Cavitation Defect I	0.62	148
Cavitation Defect II	0.64	158

cavitation region has been examined, the difference in the number of cycles is very small when compared to the original model, especially in the blade cavity first model. However, it is important to emphasize that in this fatigue analysis, the most critical stress state is considered to occur in 100% of the cycles, which is not the real case. In a more realistic model, it should be considered the average of the mechanical stresses in a given period, resulting in less stress than the presented analysis considered.

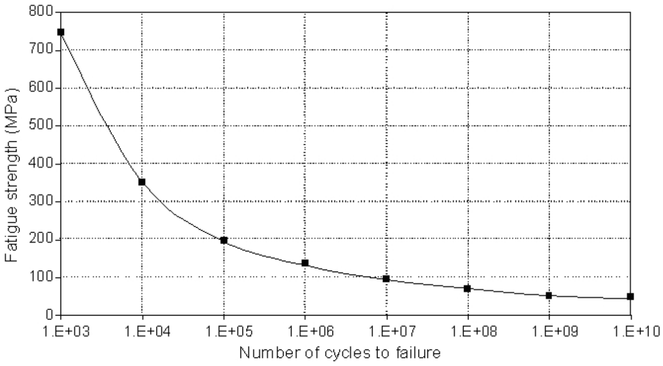


Figure 13. Fatigue characteristics of low carbon steel alloy in water.

6. APPLICATION OF OPTIMIZATION TECHNIQUES IN WEAR ANALYSIS

The topological optimization method (TOM) (Bendsøe and Sigmund, 2003) has been applied to examine the points in which the material can be removed without damaging the turbine structural rigidity, and thus, its performance. It is a first approach to evaluate the limit of wear.

To apply the TOM to the turbine, the TOSCA software (TOSCA, 2007) is used together with ANSYS software. In TOSCA, the objective and the constraint functions represent the stiffness maximization and the amount of material to be removed, respectively. The ANSYS is used as a solver. In the used model, it is considered as load the the turbine weight, pressure, and speed of rotation, with the values described above. The same boundary conditions as in the past models are used, but the cyclic symmetry condition.

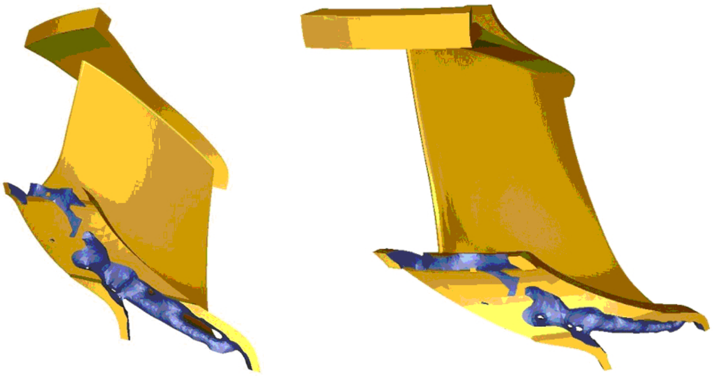
The optimization considers that removing material does not decrease the turbine structural rigidity, and the amount of material to be removed is controlled by constraining the volume. Only the band and the internal crown are considered editable regions, as long as the final amount of material of the turbine is larger than or equal to 80% of the original material. The result is shown in Fig. 14. Note that the method removes material from the lower internal region of the band, because for the used model, this region contributes least to the structural rigidity. As the region of the blade can be optimized, one can observe that material is removed from the blade, near the input lower edge.

7. CONCLUSIONS

This study presents a methodology for determining the suction and pressure surfaces initial geometry that approximates the originally designed geometry. The algorithm automatically does a balance between two opposing forces: interpolation (distance) and smoothing (length). Subsequently, the determined surface can be used in the repairing process to evaluate the distance between a point on the real surface and the determined surface.

A CAD model is built from the cloud of points measured from the turbine of the Jaguari hydroelectric plant, allowing the creation of a CAE model to calculate the pressure distribution through the CFX analysis and to perform numerical modeling in ANSYS for displacements and stress distribution.

Sensitivity studies are implemented considering an rounding entry edge radius between the blade and the crown. The influence of cavitation effect in the stress concentration of the turbine blade is also analyzed. The presence of the rounding edge entry radius increases the displacement and stress due to the change of pressure distribution. The presence of cavitation defects causes an increase of displacement and mechanical stress. For these studies, it is assumed an infinite



(a) (b)

Figure 14. Post–processed result of the optimized turbine considering all loads and material volume constraint equal to 80%.

fatigue life considering one daily start up and stop.

The use of optimization techniques to evaluate the wear limit resulted in the removal of material only in band, because the pressure applied to the blade does not allow the removal of its material and the crown is an important element for the turbine stiffness because it connects the turbine to the shaft. However the band mass is important because it guarantees a minimum inertia for the turbine. Thus, it is suggested to replace the volume restriction by a inertia moment restriction. One can also consider as a region for removing material, the region of weld between blade and crown and blade and band.

The implemented computational methodology allows optimizing the maintenance of components, i.e., it assesses and predicts the structural behavior of mechanical components subjected to various loading conditions, to diagnose faults and lifetime extension, regardless experimental tests.

8. ACKNOWLEDGEMENTS

Marcos de Sales Guerra Tsuzuki, Emilio Carlos Nelli Silva, Julio Cezar Adamowski and Flávio Buiochi are partially supported by CNPq. This project is supported by ANEEL.

9. REFERENCES

ANSYS, 2007, Release 11.0. ANSYS CFX tutorials. ANSYS, Inc.

ANSYS, 2007, Release 11.0. ANSYS tutorials. ANSYS, Inc.

Bartels, R.H., Beatty, J.C., Barsky, B.A., 1986, "An Introduction to Splines for use in Computer Graphics and Geometric Modeling", Ed. Morgan Kaufmann.

Bendsøe, M.P. and Sigmund, O., 2003, "Topology optimization: theory, methods and application", Springer, Berlin.

Dubas M. and Schuch, M., 1987, "Static and Dynamic Calculation of a Francis Turbine Runner of with some Remarks on Accuracy", Computers and Structures, Vol. 27, pp. 645-655.

Elber, G., 2001, "Rendering with Parallel Stripes", IEEE Computer Graphics and Applications, Vol. 21, pp. 44–52.

Kirkpatrick, S., Gellat, C.D., Vecchi, M.P., 1983, "Optimization by Simulated Annelaing", Science, Vol. 220, pp. 671-680.

Liang, Q.W., Rodriguez, 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, Vol. 36, pp.1106-1118.

Nilsson, H. and Davidson, L., 2003, "Validations of CFD against detailed velocity and pressure measurements in water turbine runner flow", International Journal for Numerical Methods in Fluids, Vol. 41, pp.863-879.

Richter, C.-H., 2003, "Structural Design of Modern Steam Turbine Blades using Adina", Computers and Structures, Vol. 81, pp. 919-927.

Rodriguez, C.G., Egusquiza, E., Escaler, X., Liang, Q.W., Avellan, F., 2006, "Experimental Investigation of Added Mass Effects on a Francis Turbine Runner in Still Water", Journal of Fluids and Structures, Vol. 22, pp. 699-712.

Shigley, J. E., 1984, "Elementos de Máquinas", Livros Técnicos e Científicos Editora S.A., terceira edição.

TOSCA, 2007, General Documentation, version 6.1, rev. 1.

Wickström, A., 1997, "Structural Analysis of Francis Turbine Runners Using Adina", Computers and Structures, Vol. 64, pp. 1087-1095.

10. Responsibility notice

The author(s) is (are) the only responsible for the printed material included in this paper