Proceedings of COBEM 2009
20th International Congress of Mechanical Engineering
November 15-20, 2009, Gramado, RS, Brazil

NUMERICAL SIMULATION OF THE OPTIMIZED HYDRAULIC TURBINE BLADES

Anna Paula de S. P. Rodrigues, anjucav@unb.br
Antonio C. P. Brasil Junior, brasiljr@unb.br
University of Brasilia
Department of Mechanical Engeneering
Laboratory of Energy and Environment
70910-900. Brasília-DF

Abstract. *The objective of the present paper is to determine the optimal shape for the hydraulic turbines blade, for which it has the best performance. To estimate the performance of a given blade design of a hydrokinetic turbine (HKT), the ANSYS-CFX 11 comercial code in batch mode was used. The optimal geometry is determined using Genetic Algorithms (GA). The objective function maximized here is the torque. The blade shapes resulting by this optimization process were compared with the results of a previous optimization process. The new results had shown an increase the power output of the HKT, confirming the effectiveness of the use of GA and ANSYS-CFX 11 as an optimization tool.*

Keywords*: Hidrokinetic turbine, optimization, genetics algorithms, simulation, ANSYS-CFX*

1. INTRODUCTION

Considerable efforts are devoted to improve the efficiency of turbomachinery and the design of the optimal blades is a very important part to improving this efficiency, since the optimal blades will ensure that the turbine will always work on its optimal performance.

Classical methodologies of optimal blades can be described as an iterative processes based on previous and initial designs (Lopez, 2006). In this type of project, the objective is to determine the optimal shape for the blade, using computational fluid dynamics (CFD) methods as a computational tool. This tool is used to calculate the blade performance, CFD methods solve the flow around the blade, by solving the flow equations for prescribed boundary conditions on the borders of the domain.

In the blade design phases, the optimal geometry is outcome of the design problem, making it necessary to analyze by trial-error a large number of designs in order to find the best one. Each blade analysis requires a CFD computation, which can make this process computationally expensive and thus time-consuming. It does not ensure that the best shape has been found (Harinck, 2004).

The imperative needs of “better” design, over shorter time periods, promote the efforts to explore new design methodologies and to improve the design cycle. For example, use of artificial intelligence (AI) techniques.

Few articles about blade shape optimization are available in the literature. In the work of Shingai *et al.*,(2006), an axial turbine runner blade is optimized by using simulated annealing (SA) techniques. The optimized runner is obtained and experimental verification shows an increase in performance.

Flores *et al.* (2006) presents numerical optimization of turbomachinery blades shapes, using artificial neural network. The aim of this work is to evaluate the importance and efficiency of neural network to model and predict the efficiency values of the turbine. The developed model can be used for the prediction of the efficiency in short simulation time.

Recently, particle swarm optimization (PSO) technique has attracted considerable attention among various modern heuristic optimization techniques. Some results can be observed in Mouser and Dunn (2005); Panda and Padhy (2007).

This paper presents a methodology for the design optimization of the axial runner blade using CFD analysis tools and genetics algorithms. In this work the optimization is applied in the hydrokinetic turbine (HKT) runner blade. The genetics algorithm was chosen as an optimization methodology because it can handle both discrete and continuous variables, nonlinear objective and constraint functions without requiring gradient information. Besides, it does not require a database as a neural network and it can presents similar results and a faster computational time when compared to PSO.

The domain used in this methodology of optimization is composed of the: section of the rotor and the channel between the blades. The geometry of the cone after the rotor blade was discarded, keeping the plane the meridian channel in the output. This was done because the simplification of the mesh generation, limiting the number of nodes and the region of interest is the runner blades.

2. GEOMETRY OF AXIAL HIDORKINETIC TURBINES

Hydrokinetic turbines, Fig. 1, are a kind of turbine designed to generate electricity using only the kinetic energy of water flow in rivers. These turbines convert the kinetic energy of water directly, without interrupting its natural course, similar to the wind turbine. Greater detail of hydrokinetic turbines can be found at Brasil-Junior *et al.*, (2006) and Van-Els *et al*.,(2003).

For the generation of the domain of calculation, channel between the blades of HKT shows in Fig. 2, and mesh generation, the software ANSYS TurboGrid was used. The TurboGrid is a computational tool that generates structured meshes of regular geometries of turbomachines blades. The use of predefined topologies templates makes it a friendly tool for the user, optimizing the process of generating the mesh.

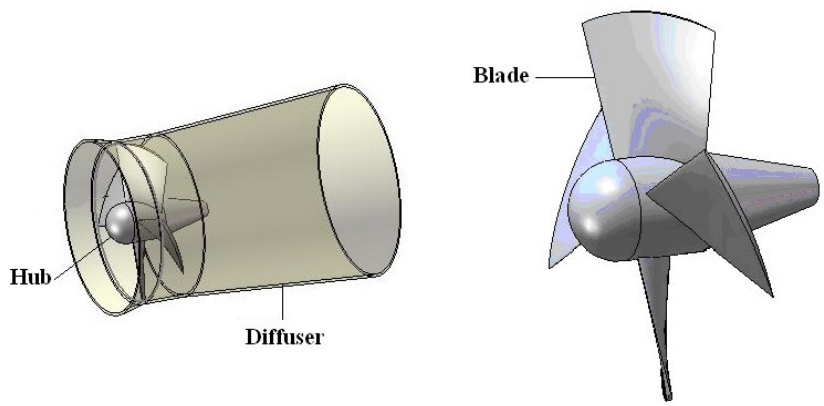

Figure 1.Hydrokinetic turbine.

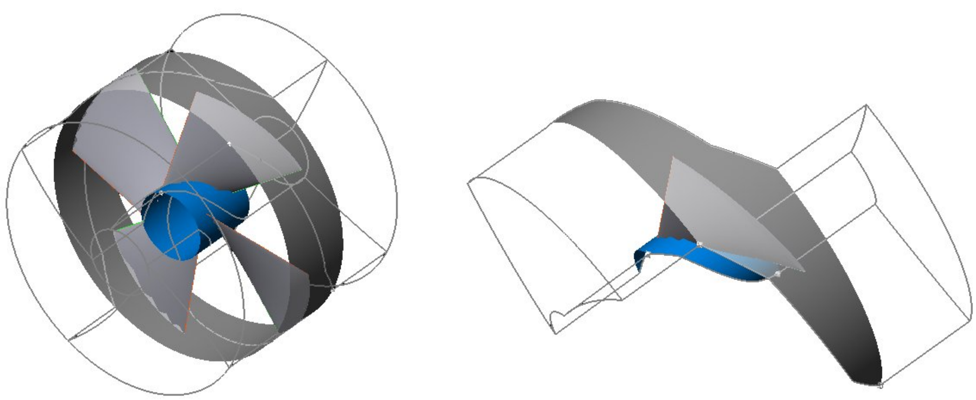

Figure 2.Channel between blades of hydrokinetic turbine.

The construction of the channel between blades was performed through the import of the rotor geometry of the HKT to TurboGrid. This is the reading of three basic files. These files define the path of the flow and geometry of the blades and should contain the coordinates of the cube, the shroud and the blade profiles in the radial stations established. These files are defined as:

- **File of the blades geometry**: It contains the points that define the radial profiles in each season, being a closed curve in the Cartesian coordinate system or cylindrical. A minimum of two profiles is required, one must be exactly on the surface of the cube and another just on the surface of the shroud. In ASCII format.

- **File of the hub geometry**: It contains the points that define the cube of the machine, in the Cartesian coordinate system or cylindrical. The points must is directed towards the amount of the flow downstream, and exceed the trailing edge and leading edge, i.e. to define the meridian section defined for the domain of computation.

- **File of the shroud geometry**: It contains the points that define the shroud of the machine, in the Cartesian coordinate system or cylindrical. The points are directed towards the amount of the flow downstream, and exceed the trailing edge and leading edge, i.e. to define the meridian section defined for the domain of calculation.

After the representation of the channel between blades of HKT, this was submitted to the ANSYS-CFX 11 in order to obtain a numerical analysis. This allows qualitatively assess the geometries generated by obtaining the power output of hydraulic turbines, in addition to the visualization of flow between the blades.

3. CFD SIMULATIONS MODULE

The next step after the geometry generation in the optimization scheme is to determine if a new geometry proposal can better meet the expected requirements than other designs. The evaluation of the performances is obtained by running Computational Fluid Dynamics simulations.

The main interest in CFD comes from the flow structure insight they can provide. Although the results obtained are valuable and realistic, they are not fully representative of reality and final validations with experimental data are commonly performed. They accelerate the decision making cycle to obtain a new design reducing the elapsed time required by traditional design processes (López, 2006).

For the resolution of the flow was used the commercial package ANSYS-CFX 11, as shown in Fig. 3. What enabled the analysis of the problem since the formulation of its geometry and mesh, to visualize the flow later.

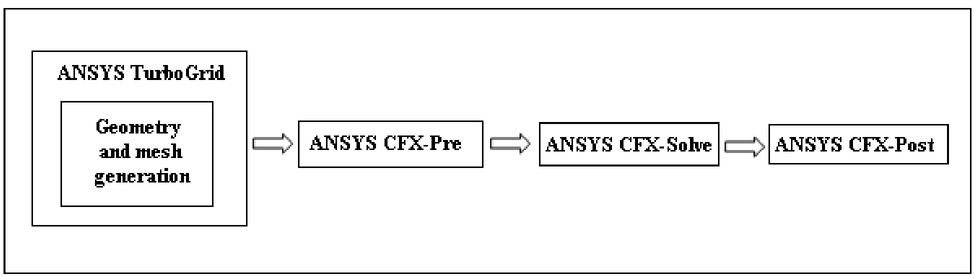

Figure 3.Module of numerical simulation in ANSYS-CFX 11.

In ANSYS TurboGrid were performed the following steps:

1. Geometry definition, construction of the channel between blades, described in section 2;
2. Select of topologies to adjust the specifics of a particular blade design. The topology is a structure of blocks that acts as a framework for positioning mesh elements. Topology blocks represent sections of the mesh that contain a regular pattern of hexahedral elements. The method of topology used in this paper was H/J/C/L-Grid, this type allows a separate choice of topology type for the upstream and downstream ends of the passage mesh.
3. Mesh generation using the current state of the Topology Set and Mesh Data objects. This mesh is the input to pre-processing module.

The pre-processing is responsible for determining the boundary conditions of the problem (inlet, outlet and wall). Already in the solver, are solved the Navier-Stokes equations, discretized under the finite volume method, allowing the process of problem convergence. Finally, post-processing is possible to obtain views of the flow, values of forces and factors, in addition to the fields of property. The flow formulation in hydraulic turbines, in addition to the turbulence models used in numerical simulation in this work can be found at (Rodrigues, 2007).

The entire process shown in Fig. 3 was performed in batch mode, or execution of a series of programs (jobs) without human interaction, through the command line interface. This command line has been implemented in Matlab 7.1, facilitating the subsequent coupling of the numerical simulation module with the optimization module.

4. GENETIC ALGORITHM OPTIMIZATION

The goal of this optimization process is improve the performance of hydrokinetic turbine, consequently, a higher power output. It makes use of optimization method-genetic algorithms (GA).

Genetic algorithms are a numerical technique that simulates Darwin's evolutionary theory to find the optimum. GA combines selection, crossover, mutation, and elitism operators with the goal of finding the best solution to a problem (Manikas and Cain, 1996). According to this theory, an individual (geometry) with favorable genetic characteristics (values of the design variables) will most likely produce more preferment offspring. By selecting them as parent, the next generation will have better individuals than the previous one. The best one of the last generation is the solution of the GA optimization process.

A correct setting of the GA parameters is therefore very important to reach the optimum with minimum effort. In the present work, the variables for the GA algorithm are real code, where an individual is characterized by a vector containing [value of leading edge angle (β_e), trailing edge angle (β_s) and chord (L_c)] of each blade section.

Thus the objective function of this algorithm is the maximum power output, achieved through a combination of individuals described above. A genetic algorithm (using Matlab 7.1 on Windows XP platform) has been implemented and is described below:

4. A random initial population is generated by setting the number of individuals, upper and lower limit of chromosome, generation’s number and likelihood of elitism and mutation. Where each subject or chromosome is composed of the fields [β_e, β_s, L_c];
5. Computation the fitness of each individual population, which are submitted to the selection process, determining which subject will move to the crossover. The method of classification used was the roulette method.
6. The next step is the crossover. In this step genetic material is exchanged between the subjects and altered (mutation). This process is repeated to create the *n* individuals of new generation.
7. In the end of this process, the best subject is selected.

The whole process of optimization is shown in Fig. 4. This figure may arise from the coupling module of numerical simulation in ANSYS-CFX 11, with the optimization module, genetic algorithm code.

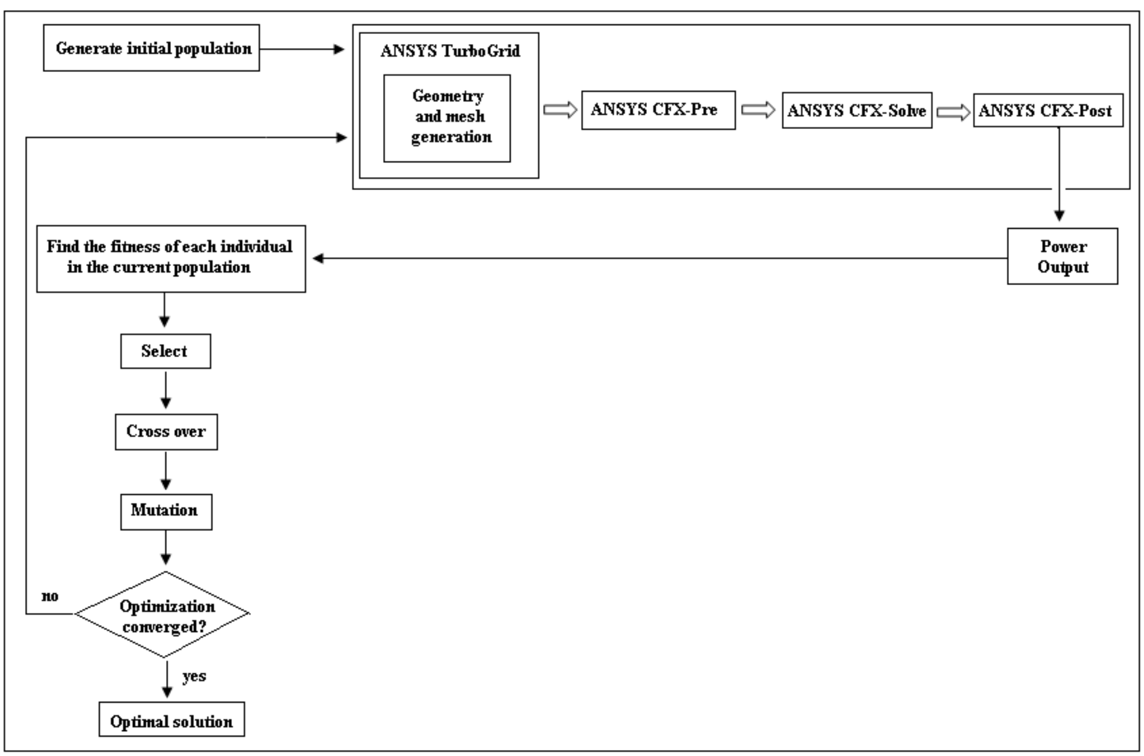

Figure 4.Optimization module.

Using this algorithm, an optimized blade is obtained. This algorithm presents output as the combination of leading edge and trailing edge angle and chord of each radial position, of the best individual. These data allow the calculation of the coordinates of the blade profiles, and then its transformation to cylindrical coordinates. Getting as output values $X(X_{ci}, X_{cs})$, $Y(Y_{ci}, Y_{cs})$ and $Z(Z_{ci}, Z_{cs})$ to under and over the airfoil in cylindrical coordinates.

Thus, after the reconstruction of the blade geometry, it is possible to generate the curves file (.curve). This file contains the geometry of the channel between the optimized blades rotor in ANSYS TurboGrid. The main purpose of the construction of this geometry in TurboGrid is further optimized numerical simulation of the blade in ANSYS-CFX 11, enabling an analysis of this new geometry.

5. RESULTS AND DISCUSSIONS

The genetic algorithm described in section 4, was performed with 30 individuals forming the population, a mutation probably of 0.3 percent, an elite probability of 0.3 percent and 100 generations as the halt condition for convergence. On the ANSYS-CFX module, Fig. 2, it was used the rotation speed of 60 rpm and 90 rpm, beyond the following boundary conditions. Resulting in computational domain, Fig. 5, of the channel between blades.

- Inlet: freestream velocity equal to 2 m/s;
- Outlet: reference pressure;
- Shroud: counter rotating wall;
- Rotating periodicity between runner blade;

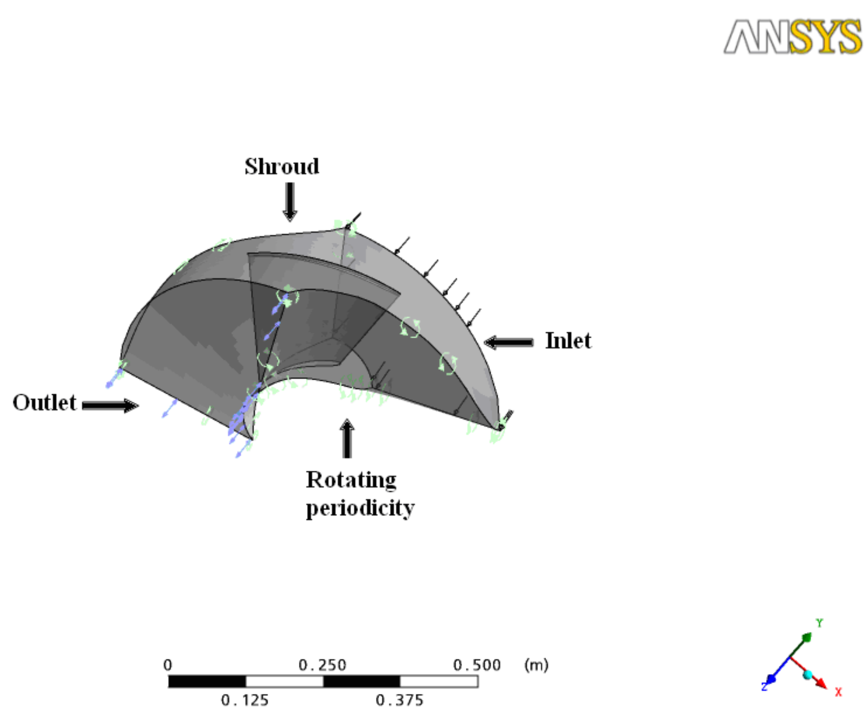

Figure 5.Computational domains of the channel between blades.

In the optimization process were generated two new optimized blades geometries (60 rpm and 90 rpm). These new geometries were simulated (using ANSYS-CFX 11) on a steady state with a free flow velocity of 2 m/s , and different rotation speeds. This simulation allows the computation of the torque on a blade, and after, it is multiplied by the total blades number, in these case four blades. Thus, the ANSYS-CFX 11 post was used. In the next step, the curve of turbine power output was generated, which is compared to the curve of the old optimization, Fig. 6 and Fig. 7.

The old optimization has been achieved through the use of a simplified mathematical model (implemented in Matlab 7.1) which describes the behavior of hydrokinetic turbine. It was used to calculate the fitness of each individual in the process of optimization. Moreover, in this work the fitness is calculated using the ANSYS-CFX 11. The description of how the old optimization has been done can be found in (Rodrigues et al, 2008 and Rodrigues, 2007).

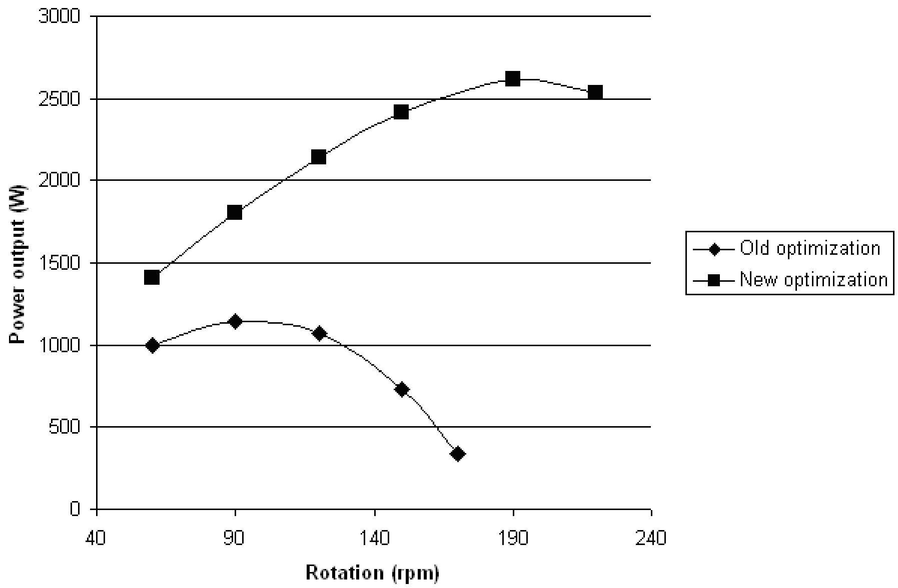

Figure 6.Power generated for the 60 rpm optimization.

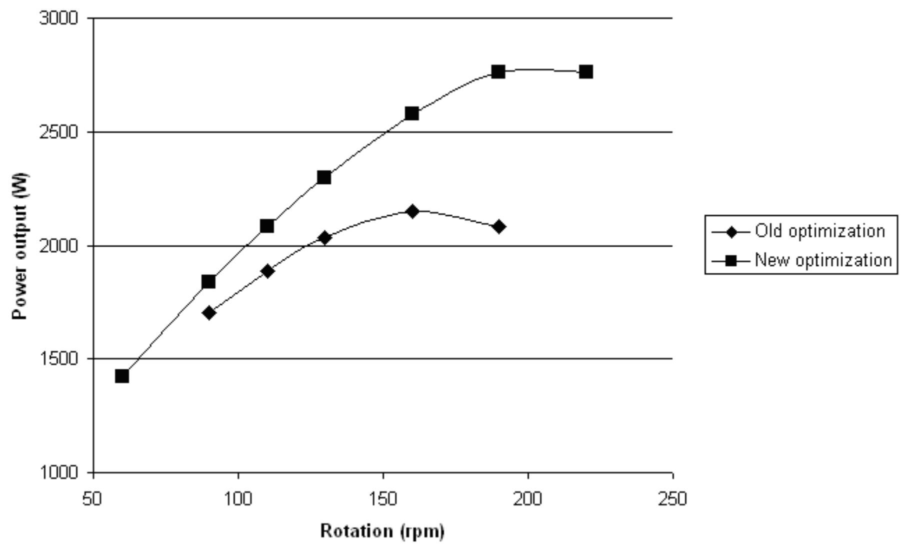

Figure 7.Power generated for the 90 rpm optimization.

In these charts, an improvement in the HKT performance, mainly for optimization at 60 rpm can be noted. It is due to the fact that the CFX methodology is more accurate than the simplified mathematical model, which was previously used.

The comparison between the new (blue) and old blade geometry (green), new and old optimization are shown in Fig. 8. One can note the effect caused due to geometric changes, especially in the optimization to 60 rpm. The same effect can be noted in the graph of Fig. 6.

These new geometries were evaluated through numerical simulations in ANSYS-CFX 11. The stream lines, Fig. 9 (optimized blade for 60 rpm) and 10 (optimized blade for 90 rpm), show the trajectory followed by the flow along the channel between blades. Note that these lines follow the channel direction without deviations, this feature demonstrates the proper positioning of the blade.

From the pressure field acting on one rotor blade, both for 60 rpm optimization (Fig. 11) as a 90 rpm (Fig. 12), we can see a decrease of the same occurring of the upstream to downstream. This energy decrease experienced by the pressure is converted into kinetic energy to flow.

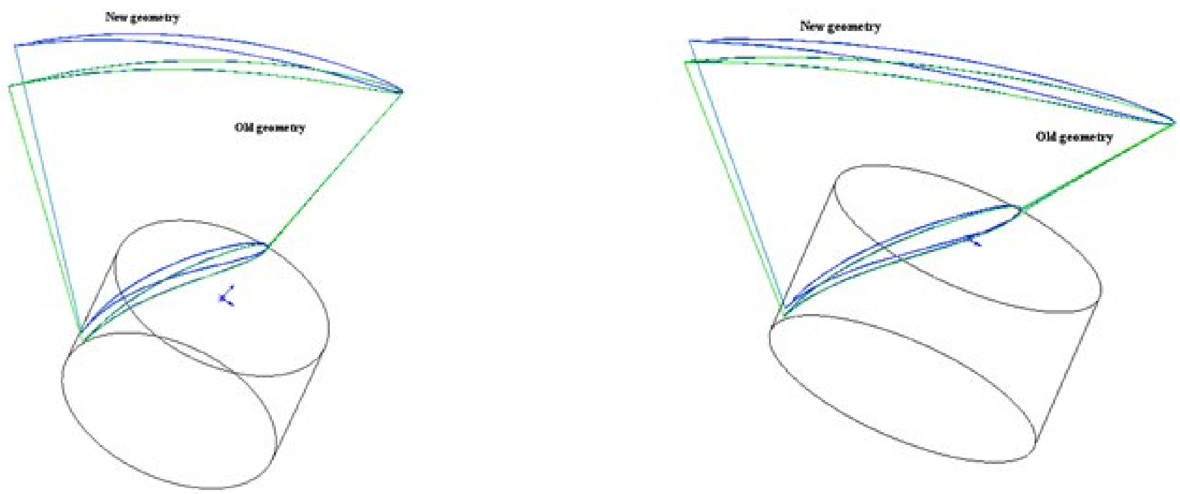

Figure 8.Comparison between the new (blue) and old blade optimization geometry (green) for the 60 rpm (left) and 90 rpm (right).

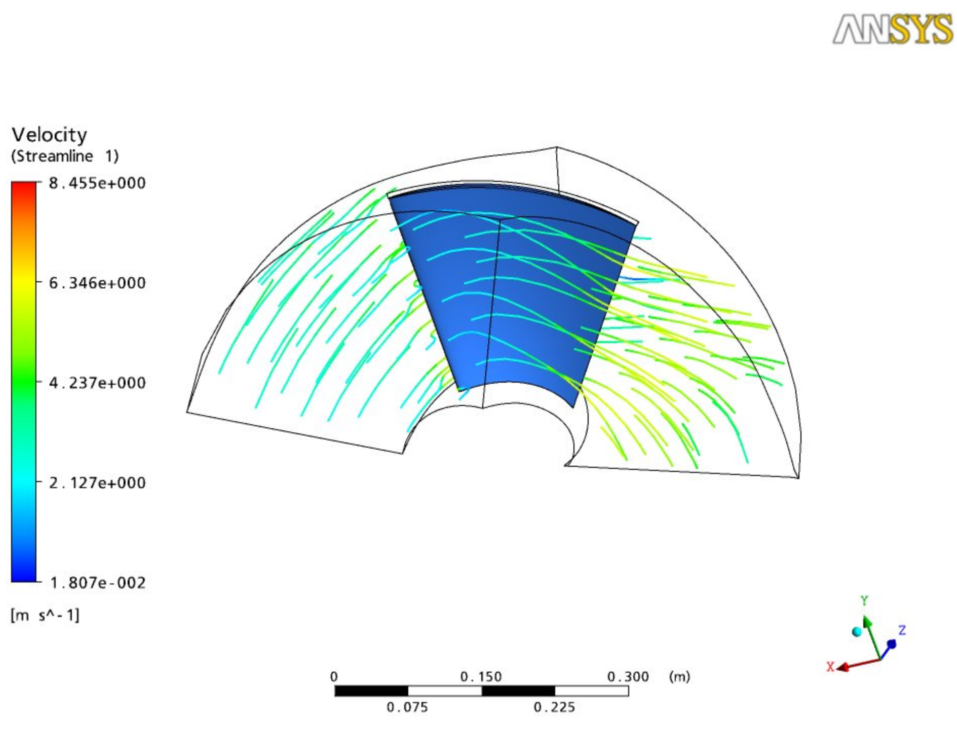

Figure 9.Stream lines for the 60 rpm blade optimization.

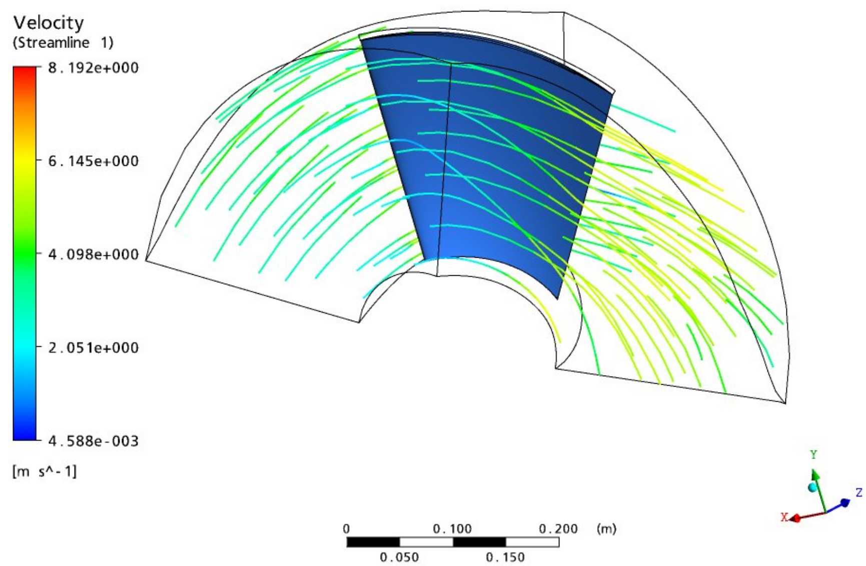

Figure 10.Stream lines for the 90 rpm blade optimization.

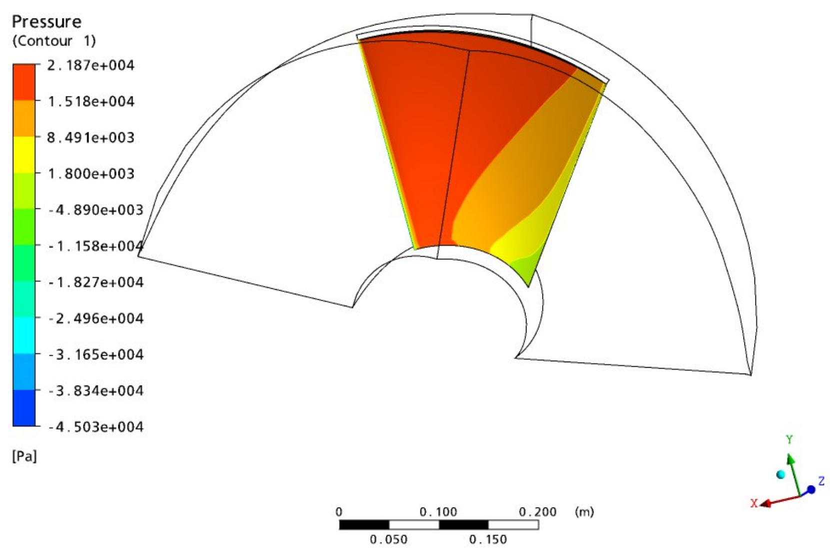

Figure 11.Pressure field for the 60 rpm blade optimization.

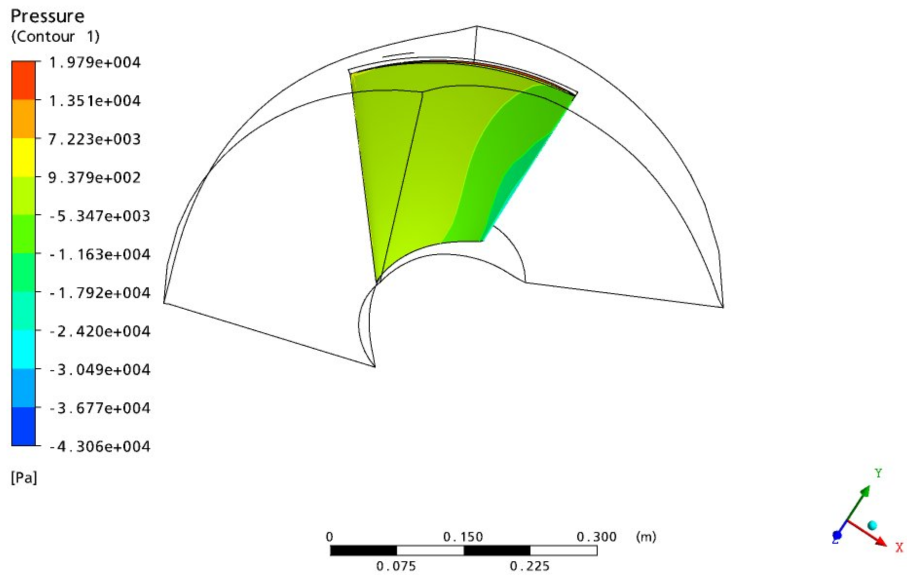

Figure 12.Pressure field for the 90 rpm blade optimization.

6. CONCLUSION

This paper has described the development of an optimization methodology to improve the performance of an axial turbine, hydrokinetic turbine. The numerical simulation module, ANSYS-CFX 11, coupled to an optimization module, genetics algorithms were used. The results obtained were compared to the results of an old methodology, which used a simplified mathematical model instead of the commercial code ANSYS-CFX 11. The new results showed a significant improvement, due to accuracy of the ANSYS-CFX 11 calculation.

Although this new methodology present a computational time higher than the previous one, the improvement in the power output proved the feasibility of this methodology in the optimization of hydrokinetic turbine, or any other axial turbine.

7. ACKNOWLEDGEMENTS

This work was supported by P&D program of ELETRONORTE S/A and CAPES. The authors wish to express his appreciation to Luciano Noleto (UnB) for his suggestions and discussions.

8. REFERENCES

Brasil-Junior, A. C. P., Salomon, L. B. R., Van-Els, R. and Ferreira, W. O., 2006. "A new conception of hydrokinetic turbine for isolated communities in amazon", in 'Proc. of CONEM 2006'.

Flores J. G., Hernandez J. A., Urquiza G.,2006, "Numerical Optimization of the Hydraulic Turbine Runner Blades Applying Neural Networks". UAEM. Cuernavaca, Morelos, Mexico.

Harinck J., 2004, "Genetic Algorithm for Radial Trubomachinery Design Optimization". Delft University of Technology.

López, L. F., 2006, "Surface Parameterization and Optimum Design Methodology for Hydraulic Turbines", Thése, École Polytechnique Fédérale de Lausanne.

Manikas, T. W. and Cain, J. T., 1996, "Genetic Algorithms vs. Simulated Annealing: A Comparison of Approaches for Solving the Circuit Partitioning Problem". University of Pittsburgh. Pittisburgh, PA.

Mouser, C. R. and Dunn, S. A., 2005, "Comparing Genetic Algorithm and Particle Swarm Optimisation for an Inverse Problem Exercise". ANZIAM J. 46. ppC89-C101.

Panda, S. and Padhy, N.P., 2007, "Comparison of Particle Swarm Optimization and Genetic Algorithm for TCSC-based Controller Design". International Journal of Computer Science and Engineering 1:1.

Rodrigues, A.P.S.P., 2007, "Parametrização e Simulação Numérica da Turbina Hidrocinética-Otimização via Algoritmos Genéticos". Dissertação de Mestrado.Departamento de Engenharia Mecânica.Universidade de Brasilia.

Rodrigues, A.P.S.P., Brasil Junior, A.C.P., Salomon, L.R.B., 2008, "Simulação e Otimização de Turbinas Hidrocinéticas". V Congresso Nacional de Engenharia Mecânica. Salvador, Bahia, Brasil.

Shingai K., Katayama K., Shimmei K.,2006, "Optimization of the Axial Turbine Runner Blade Using a Simulated Annealing Algorithm".23 IAHR Symposium. Yokohama.

Van-Els, R., Campos, C. O., Henriques, A. M. D. and Balduino, L. F., 2003. "Hydrokinetic propeller type turbine for the electrification of isolated householders or communityand social end-users", in ’Proc. of 17th Congress of Mech. Eng.’.

9. RESPONSIBILITY NOTICE

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