24th ABCM International Congress of Mechanical Engineering
December 3-8, 2017, Curitiba, PR, Brazil

COBEM-2017-1237
OPTIMIZATION STUDY ON AN ULTRA-LOW HEAD BULB HYDRO TURBINE

Antonio Carlos Barkett Botan
Ramiro Gustavo Ramirez Camacho
Universidade Federal de Itajubá, Instituto de Engenharia Mecânica, Av. BPS, 1303, Bairro Pinheirinho, Itajubá – MG
acbotan@yahoo.com.br
ramirez@unifei.edu.br
Geraldo Lucio Tiago Filho
Universidade Federal de Itajubá, Instituto de Recursos Naturais, Av. BPS, 1303, Bairro Pinheirinho, Itajubá – MG
tiago@unifei.edu.br

Abstract. *The increase of efficiency of a turbomachine requires a behavior analysis of the flow on its hydromechanical components, in a global way. For a proposal of modification in the profile of the components on a conceptual machine, aiming its optimization, it is necessary to survey into the reason of the possible shortcomings and, finally, to present arguments for their optimization. The use of Computational Fluid Dynamics (CFD) tools are widely used in these researches. Consolidated axial turbines, in general, presents very significant efficiencies, however, machines that operate under special conditions, such as turbines for ultra-low heads condition (0.5 meters to 5.0 meters), are still under development, opening space for research on performance improvements. This work presents an analysis based on the results obtained by computational study through CFD, in order to evaluate an ultra-low head turbine model, proposing the optimization of its performance. The analysis discussed in this paper suggested the exclusion of the original draft tube system, in order to verify the behavior of the fluid flow. This modification resulted in a slight improvement of the efficiency.*

Keywords: *Bulb turbine, ultra-low head, CFD analysis*

1. INTRODUCTION

The use of turbines operating in head ranges up to 5.0 meters is currently under development. Machines that operate up to this range of heads favors the generation of electric power without the need for larger civil works, and present low environmental impacts considering that it also has fish friendly characteristics. The technology based in axial turbines for very low head uses condition have been widely studied and demonstrated as a great technological option for decentralized generation of low environment impact (Alexander et al, 2009)(Fraser et al, 2007).

In the current work, it is presented a model of a bulb-type axial turbine developed in collaboration between teams of researchers from Canada and Brazil, countries that present sites with very similar low heads hydro potential. The turbine model is composed of a fixed six blades guide-vanes system in the upstream position of the turbine, a runner composed of three fixed blades of hydrodynamic profile and fish friendly characteristics, and at its downstream a set of double draft tube with an internal cone and four fixed blades guide-vanes system for the normalization of the flow and to guarantee its structural integrity.

With the aid of numerical analysis tools (Computational Fluid Dynamics - CFD) it is possible to quantify the behavior and efficiency of this turbine. On the other hand, the numerical solution with optimization algorithms allows to improve the geometry of the components of the turbine for higher hydraulic efficiencies, allowing a higher performance coefficient, without constructive expenses of the components for tests.

2. METHODOLOGY

The applied methodology is based on the performance evaluation of the flow on the input set (guide vanes), runner and outlet (draft tube) domains. The generated mesh on the preliminary model is classified as structured hexahedral for the inlet and outlet domains (stationary) and non-structured tetrahedral to the runner domain (rotating) (Fig. 1).

The primary analysis is based on the geometry and results presented by Martin et al (2016), based on a report of the computational results for the prototype. The unit quantities are transposed by principles of similarity on turbomachinery, since in Martin et al (2016) the analysis was done on the prototype, with a throat diameter of 750 mm, and in this work the analysis is based on the reduced scale model with throat diameter of 360 mm.

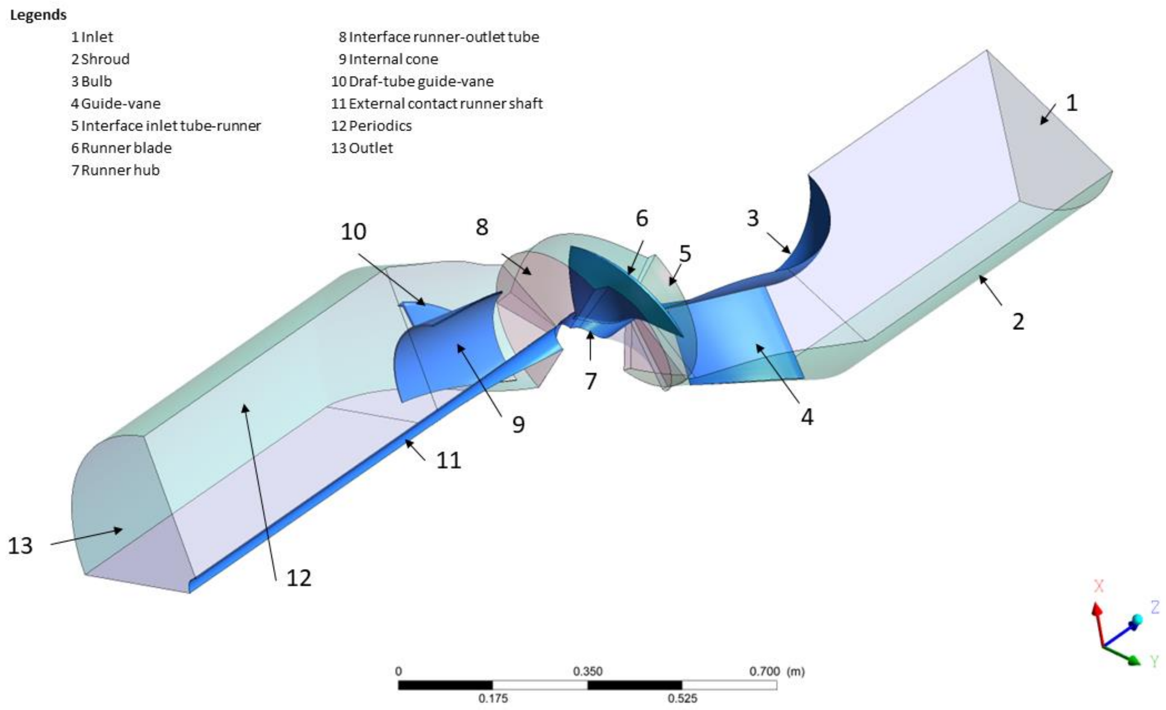


Figure 1. Turbine set regions

2.1 Computational fluid dynamics

CFD analysis was realized in four steps: Geometry, Mesh, Solver and Result analysis. To create the tridimensional geometry was used software as SolidWorks® and ICEM-CFD® from Ansys® package. Meshes were created using ICEM-CFD®, and the solver used for calculation of Navier-Stokes equations was CFX® (Ansys®). The analysis of results, variable calculation, and graphical analysis for flow behavior on turbomachinery were done using CFD-Post® (Ansys®).

2.1.1. Turbine model

The turbine model studied have proportion dimensions of 0.48 from prototype scale. For the turbine set it was developed in three domains as:

- Inlet domain: composed by upstream tube from turbine, distributor grid composed for six fixed blades guide-vanes and bulb;
- Runner: composed by three fixed blades;
- Outlet domain: composed by draft tube of two passages with internal cone and flow targeting system composed by four fixed blades guide-vanes, and turbine downstream tube.

Each domain was divided due to blades and guide vanes gap to create periodicity condition, in order to reduce the number of elements in mesh construction, consequently reducing computational cost.

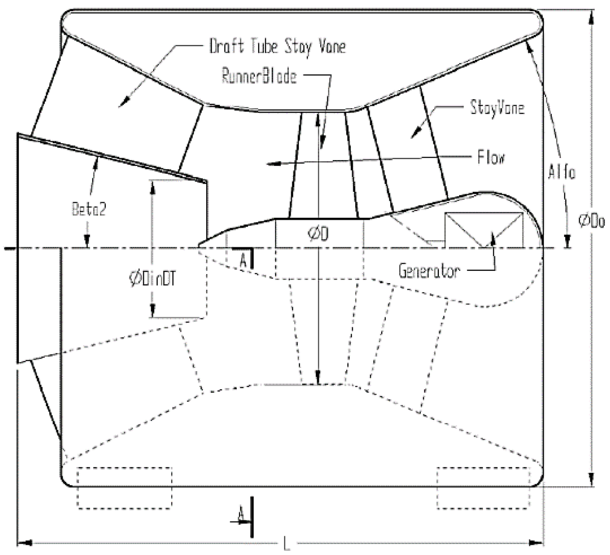


Figure 2. General arrangement of the turbine [3]

2.1.2. Mesh

The meshes created on computational model were developed as hexahedral structured for Inlet and Outlet domains (stationary) and non-structured tetrahedral for the Runner domain (rotating), and the number of elements and nodes are shown in Tab. 1. All domains were created with periodicity, which reduces the computational cost.

Table 1. Mesh information.

Domain	Nodes	Elements
Inlet domain: Guide-vanes	116,4550	1,121,630
Runner	231,114	1,249,263
Outlet domain: Draft tube (double)	484,780	458,522
All Domains	1,880,444	2,829,415

2.1.3. Solver

Flow conditions were simulated using as boundary conditions the rotation speed showed at Tab. 2, in order to vary total pressure on inlet zone (inlet face of Inlet domain) and static pressure on outlet zone (outlet face on Outlet domain). The values are calculated based on energy equation described, for constant head H=0.5 meters:

$$p_t = p_e + p_d = \rho g H + \rho \frac{v^2}{2} \tag{1}$$

Table 2. Boundary Conditions for analysis in CFD model (Dth = 0,360 m; Hnet = 0,5 m).

n [rpm]	195	266	339	413	486	564	641	725	818
Q [m³/s]	0,151	0,165	0,185	0,207	0,231	0,258	0,286	0,316	0,350

The interface connections between Inlet domain and Outlet domain with Runner domain were considered as Frozen Rotor.

The turbulence model applied was the k-omega SST - *Shear Stress Transport* (Enomoto et al, 2016)(Guénete et al, 2012)(Versteeg and Malalasekera, 1995). The turbulence model SST is a hybrid model, using the k-epsilon model in the region far from wall and k-omega model in regions of wall proximity (Versteeg and Malalasekera, 1995). Also, k-omega model uses wall functions, allowing be adopted y+ value with order between 200 and 400 (Ramirez et al, 2014).

In this way, SST model is able to trail the flow behavior concerning as big as small turbulence scales. The k and ω transport equations are presented in Eq. (2) and Eq. (3) (Menter, 1994).

Turbulence kinetic energy

$$\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_i} = P_k - \beta * k\omega + \frac{\partial}{\partial x_j}\left[(\nu + \sigma_k \nu_T)\frac{\partial k}{\partial x_j}\right] \tag{2}$$

Specific dissipation rate

$$\frac{\partial \omega}{\partial t} + U_j \frac{\partial \omega}{\partial x_j} = \alpha S^2 - \beta \omega^2 + \frac{\partial}{\partial x_j}\left[(\nu + \sigma_\omega \nu_T)\frac{\partial \omega}{\partial x_j}\right] + 2(1 - F_1)\sigma_{\omega 2}\frac{1}{\omega}\frac{\partial k}{\partial x_i}\frac{\partial \omega}{\partial x_i} \tag{3}$$

Where the kinematic eddy viscosity is given by:

$$\nu_T = \frac{a_1 k}{\max(a_1 \omega, SF_2)} \tag{4}$$

The other closure coefficients and auxiliary relations are given in Eq. (5), (6), (7) and (8):

$$F_2 = \tanh\left[\left[max\left(\frac{2\sqrt{k}}{\beta * \omega y}, \frac{500\nu}{y^2 \omega}\right)\right]^2\right] \tag{5}$$

$$P_k = \min\left(\tau_{ij}\frac{\partial y}{\partial x}, 10\beta * k\omega\right) \tag{6}$$

$$F_1 = \tanh\left\{\left\{min\left[max\left(\frac{\sqrt{k}}{\beta * \omega y}, \frac{500\nu}{y^2 \omega}\right), \frac{4\sigma_{\omega 2} k}{CD_{k\omega} y^2}\right]\right\}^4\right\} \tag{7}$$

$$CD_{k\omega} = \max\left(2\rho\sigma_{\omega 2}\frac{1}{\omega}\frac{\partial k}{\partial x_i}\frac{\partial \omega}{\partial x_i}, 10^{-10}\right) \tag{8}$$

Constant values are:
α2 = 0,44; βω2 = 0,0828; σk2 = 1,0; σω2 = 1,1682

Where mixture function F_1 is equal to the value to the unit on the wall, tending its value to zero on the external region of the boundary layer.

3. RESULTS AND DISCUSSION

3.1. First model – reduced scaled from prototype

As the model was tested computationally in nine different conditions presented in Tab. 2 for a constant head of 0.5 meters, Fig. 3 shows the velocity streamline since the inlet region in Inlet Domain, crossing for rotate domain of the Runner and guided to the double draft tube system, in Outlet Domain. In this case, the figure shows the resulting velocity streamline in imposed conditions on model for the lowest rotation 195 rpm (Fig. 3a), the maximum efficiency rotation 413 rpm (Fig. 3b) and the highest rotation 818 rpm (Fig. 3c).

Notice that on Fig. 3a, 3b and 3c the shock of the flow after passage by rotor on draft tube guide-vanes. This become evident for lower operating rotations (Fig. 3a), achieving a slight improvement for higher rotation (Fig. 3c). These shocks cause some loss on machine performance, once that draft tube have function to recover part of the not used energy the during work realized by the runner, converting outlet velocity flow from rotor in pressure, to reduce the outlet losses (Pfleiderer and Petermann, 1979)(Souza, 2011). However, the highest rotation analyzed in this paper showed the lowest performance.

At the maximum efficiency point, n = 413 rpm, shaft power obtained was 849 Watt for an efficiency of 0.76. Figure 4 presents the efficiency curve and shaft power and hydraulic power curves for the unitary turbine.

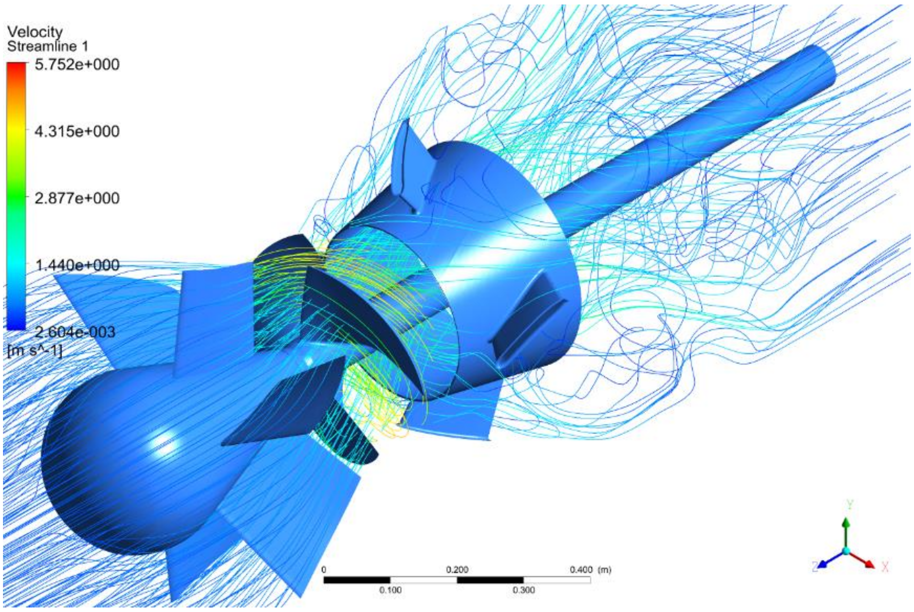


Figure 3a. Velocity streamline for n = 195 rpm, $Q_{theoretical}$ = 0.151 m³/s

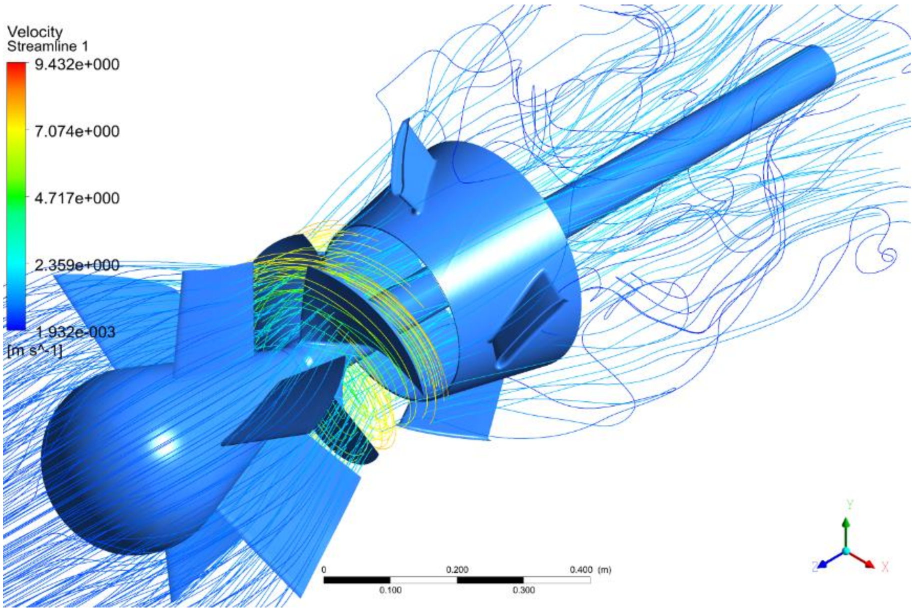


Figure 3b. Velocity streamline for n = 413 rpm, $Q_{theoretical}$ = 0.207 m³/s

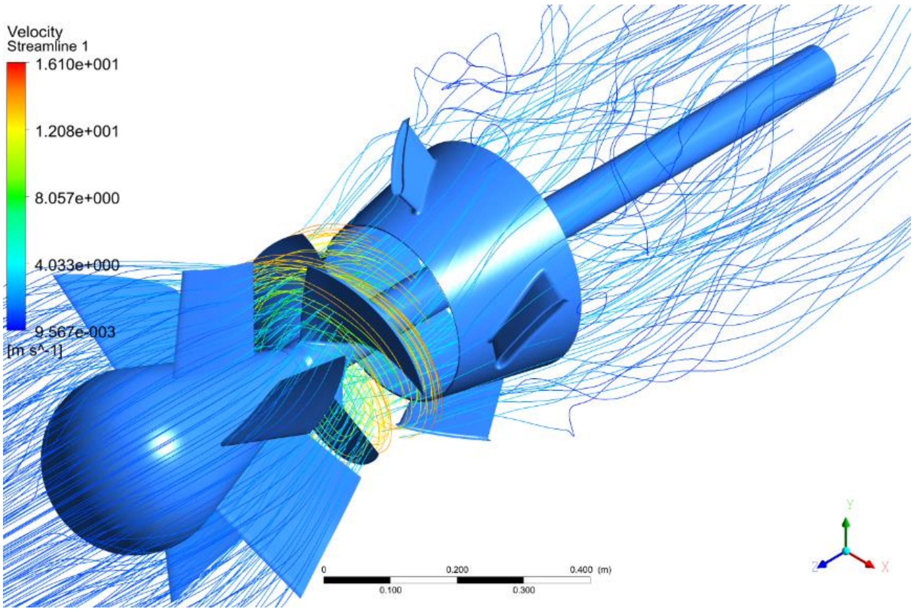


Figure 3c. Velocity streamline for n = 818 rpm, $Q_{theoretical}$ = 0.350 m³/s

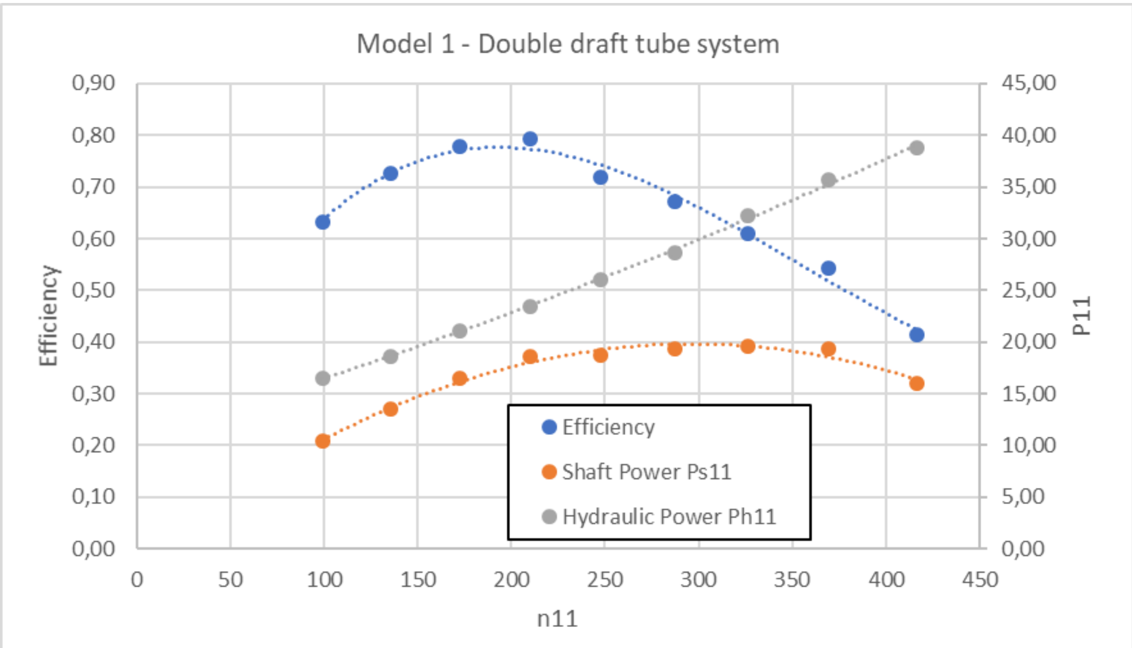


Figure 4. Efficiency curve of turbine model with double draft tube system

3.2. Second model – without double draft tube system

Due to the occurrence of flow shocks on the wall of the guide-vanes in the draft tube, a new analysis of the flow after the runner in a free tube was suggested. For that, another geometry of the Outlet Domain was created without the inner cone and without guide-vanes. It was created a new hexahedral structured mesh type, with 845,152 elements and 882,500 nodes.

The boundary conditions applied to the Solver, as well as the turbulence model, were the same as previously used (Tab. 2). It is necessary to repeat the same procedure realized in the previous geometry in order to evaluate the model and the behavior of the flow under the same conditions. Figures 5a, 5b and 5c show the behavior of the velocity streamlines in the three domains set for the lowest rotation n = 195 rpm (Fig. 5a), the maximum efficiency rotation n = 413 rpm (Fig. 5b) and the highest rotation n = 818 rpm (Fig. 5c).

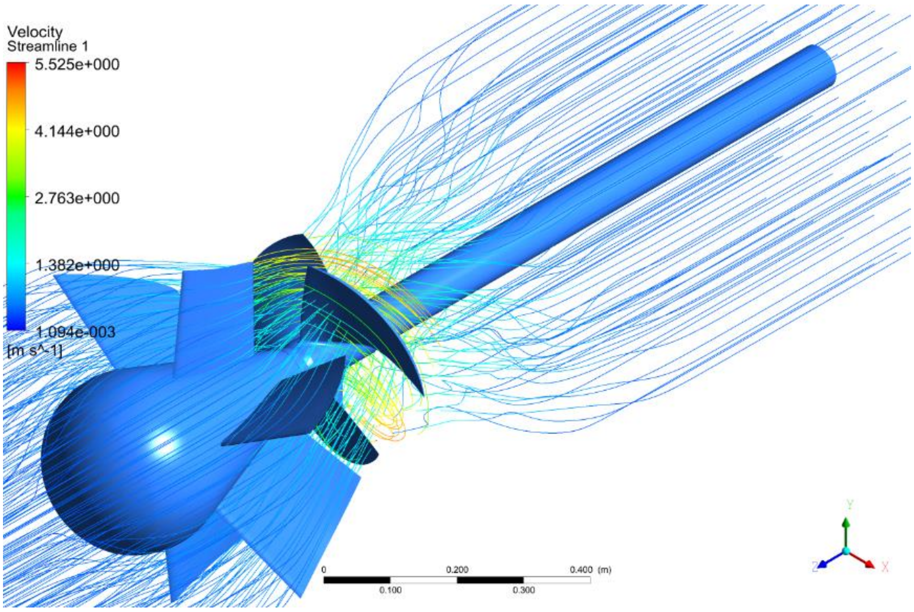


Figure 5a. Velocity streamline for n = 195 rpm, $Q_{theoretical}$ = 0.151 m³/s

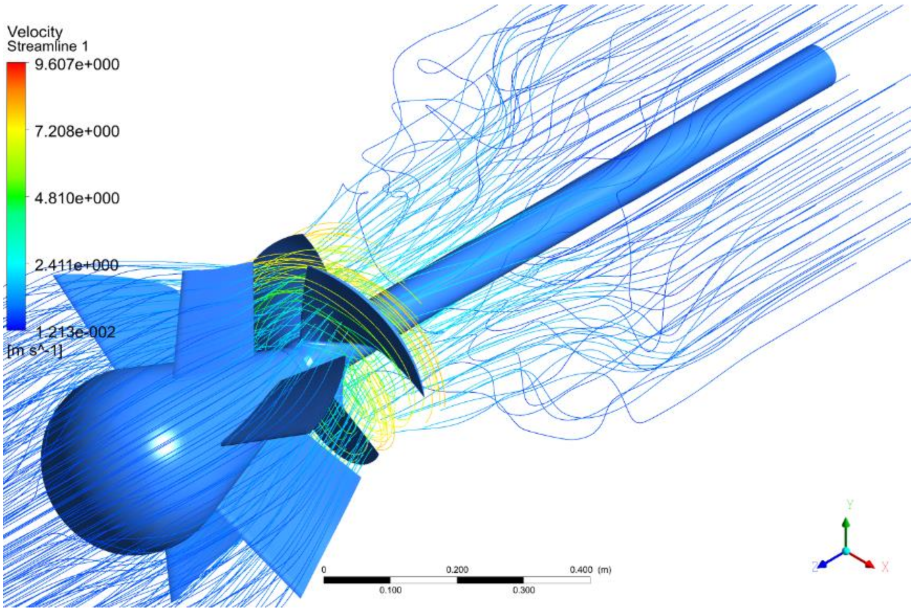


Figure 5b. Velocity streamline for n = 413 rpm, $Q_{theoretical}$ = 0.207 m³/s

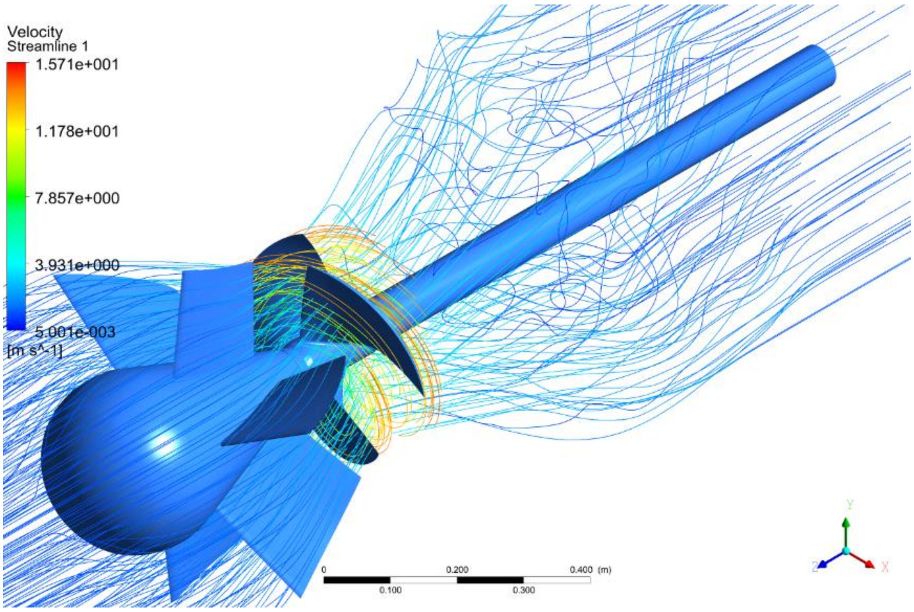


Figure 5c. Velocity streamline for n = 818 rpm, $Q_{theoretical}$ = 0.350 m³/s

It is noticed in Fig. 4 that the turbulence of the flow after the runner increases according the velocity is raised. But this turbulence is between the conic section and the linear outlet section. After that, the flow is stabilized. However, in Fig. 6, it is noticed that the efficiency is minimum under maximum velocity boundary conditions. Figure 6 presents the efficiency curve and shaft power and hydraulic power curves for the unitary turbine.

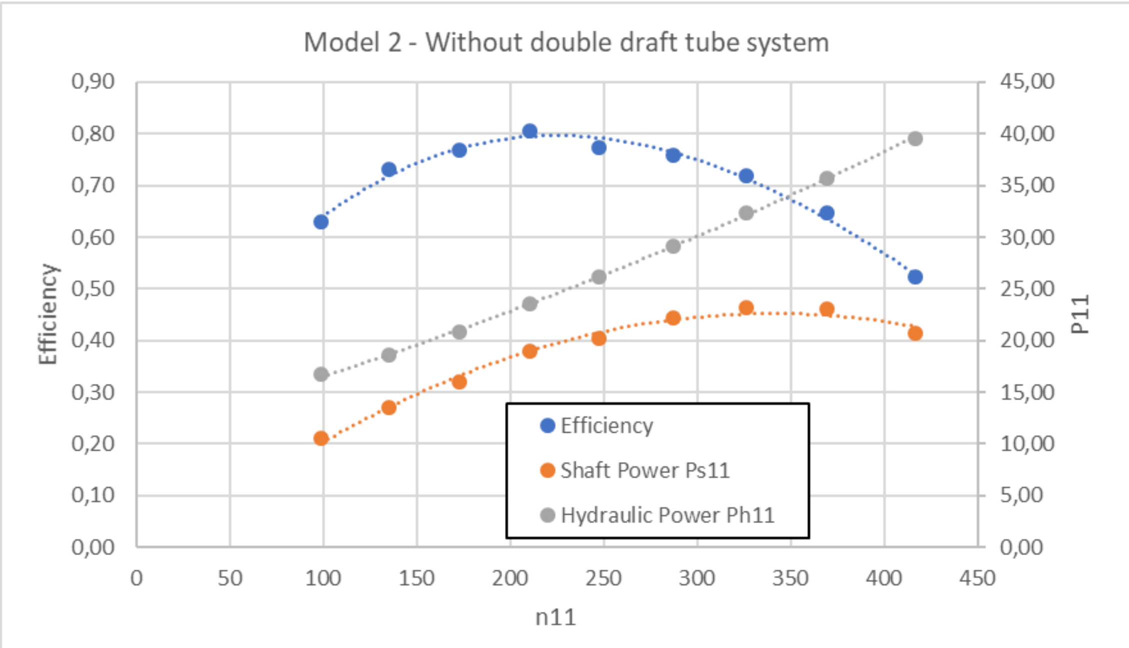


Figure 6. Efficiency curve of turbine model with single outlet draft tube system

3.3. Comparative analysis

Comparing analysis done in Model 1 and Model 2, it is noticed that there was a small gain in efficiency for the second model, without double draft tube system. The analytical unitary machine obtained results of efficiency of 0.76 for the Model 1 under n11 = 193 rpm and Q11 = 2.326 m³/s, and the model 2 had efficiency of 0.80 under n11 = 224 rpm and Q11 = 2.539 m³/s, as seen in Fig. 7a and Fig. 7b, resulting in an increase of the performance in 4% more. An explanation for the variation between the results is given by the greater rotations, and with the increase of the vectors of the absolute velocity of flow do not find obstacles in the way, thus avoiding more turbulence and restrictions of area. Figures 8a and 8b show the streamline velocity for both models.

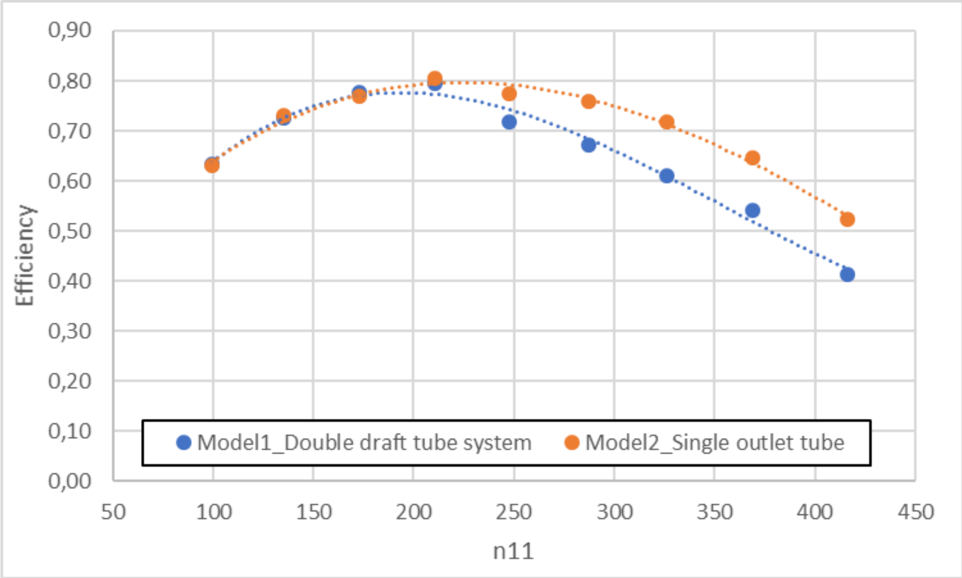


Figure 7a. Comparative chart between efficiency of Model 1 and Model 2 in function of n11

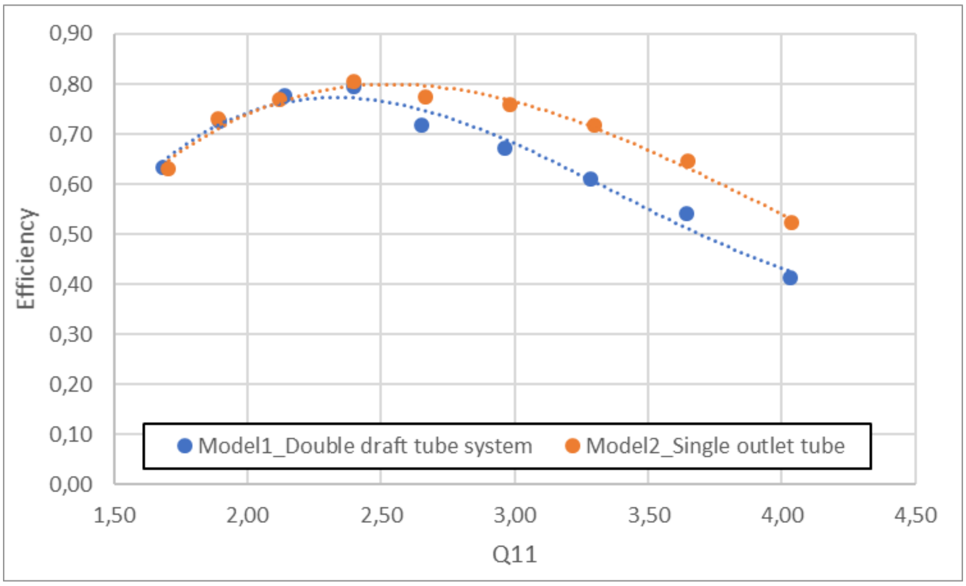
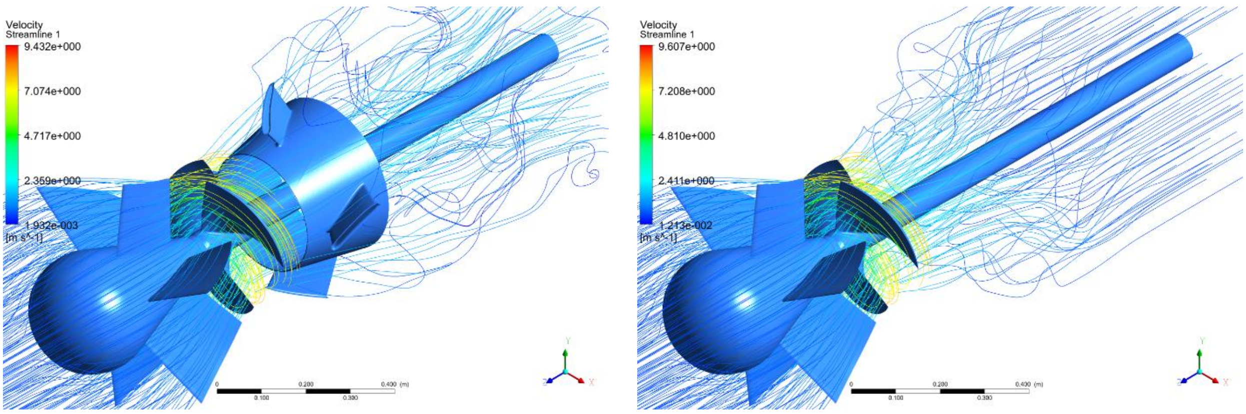


Figure 7b. Comparative chart between efficiency of Model 1 and Model 2 in function of Q11



(a) (b)

Figure 8. Streamline velocity for (a) Model 1, and (b) Model 2

4. CONCLUSIONS

The primary results achieved in this paper suggested the analyzes of a modified geometry, removing the group of double draft tube from outlet tube. Such a suggestion is resulting from the analyzed shocks of the fluid flow on the pressure surface of the guide-vanes in the Outlet Domain. This modification resulted in a raise of 4% of the efficiency of the model. As the draft tube set proposed for this machine is a complication factor because its interference on the flow, it is suggested for further studies the optimization analyzes regarding the geometric construction of the assembly or the elimination of the same, modifying the geometry of the draft tube on the Outlet Domain in order to recover the energy neglected by the Runner.

5. ACKNOWLEDGEMENTS

This work was supported by Fundação de Amparo de à Pesquisa Minas Gerais (FAPEMIG). Authors also are grateful to the partners from ISTP-FAPEMIG agreement that is referent to R&D project that resulted on the turbine project discussed in this paper.

6. REFERENCES

Alexander, K. V.; Giddens E. P.; Fuller A. M. “Axial-flow turbines for low head microhydro systems”. *Renewable Energy* Vol. 34, p. 35–47, 2009.

Ansys Inc. CFX Solver – Theory guide, 2016.

Enomoto, Y.; Nakamura, T.; Ohtake, N.; Kubo, K.; and Zhai, Y. “Numerical simulation of turbulence flow in a Bulb turbine”. *Earth and Environmental Science* Vol. 49, 2016.

Fraser, R.; Deschênes, C.; O'neil, C. and Leclerc, M. “VLH: Development of a new turbine for Very Low Head sites”. *Waterpower XV*, paper No 157, 1-9, HCI Publications, 2007 - www.hcipub.com

Guénette, V; Houde, S.; Ciocan, G. D.; Dumas, G.; Huang, J.; and Deschênes, C. “Numerical prediction of a bulb turbine performance hill chart through RANS simulations”. *Earth and Environmental Science* Vol. 15, 2012.

Martin, J.; Chang, L.; Swiderski, J.; Tung, T. T.; Tiago Filho, G. L.; Rezek, A. J.; Barbosa, W. and Botan, A. C. B. “Development of Innovative Ultra Low-head Mini-hydro Turbine-generator System-Laboratory Test Results Report and Field Demonstration Plan”. In: *Hydrovision International, 2016*, Minneapolis. Hydrovision International, 2016.

Menter, F. R. "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications". *AIAA Journal*, Vol. 32, no 8. pp. 1598-1605, 1994.

Pfleiderer, C.; and Petermann, H. “Máquinas de Fluxo, 5ª edição”. Livros Técnicos e Científicos Editora S.A., 455 p., 1979.

Ramirez Camacho, R. G., Oliveira, A. C., Oliveira, W., Silva, E. R. “Methodology for aerodynamic design and optimization of axial rotor blade based on the effects of sweep and dihedral”. *IGHEM 2014*, Universidade Federal de Itajubá, Itajubá, 2014.

Souza, Z. “Projeto de máquinas de fluxo: tomo I, base teórica e experimental”. 1 Ed. Rio de Janeiro: Interciência/Acta, 2011.

Versteeg, H. K., and Malalasekera, W. “An Introduction to Computational Fluid Dynamics: The Finite Volume Method, 1st ed”. England, Ed. Longman Scientific & Technical, 257 p., 1995.

7. RESPONSIBILITY NOTICE