NUMERICAL STUDY OF DRAFT TUBE OF A BULB HIDRAULIC TURBINE

José Gustavo Coelho, josegustavo@unb.br
Antonio C. P. Brasil Junior, brasiljr@unb.br
Universidade de Brasilia
Departamento de Engenharia Mecânica
Laboratório de Energia e Ambiente
Campus Darcy Ribeiro, Brasilia – DF, Brasil

Abstract. In this work a numerical study of draft tube of a bulb hydraulic turbine is presented, where a new geometry is proposed. This new proposal of draft tube has the unaffected ratio area, a great reduction in his length and approximately the same efficiency of the draft tube conventionally used. The numerical simulations were obtained in commercial software of calculation of flow (CFX-10), using the turbulence model SST, that allows a description of the field fluid dynnamic near to the wall. The simulation strategy has an intention of identifying the stall of the boundary layer precisely limits near to the wall and recirculations in the central part, once those are the great causes of the decrease of efficiency of a draft tube. Finally, it is obtained qualitative and quantitative results about the flow in draft tubes.

Keywords: *bulb, SST, draft tube, turbulence.*

1. INTRODUCTION

The international demand of energy is constantly increasing and in Brazil it would not be different. Through the National Energy Balance of 2005, it is had that the increase of the offer of internal energy from 1970 to 2004 was of 219%. With this increasing need, investments in the energy area turn more necessary. A great example of this is the future installation of Belo Monte's Hydroelectric Complex (CHE-BM), in the Madeira River, western Amazonia, which it will be the most important enterprise of electric power generation in Brazil in the beginning of the century. This project has an innovative conception that involves the use of two hydraulic power plants, one with Bulb turbines and other on which will be Francis turbines.

In the current days, where the environmental impacts need to be reduced, the use of machines Bulb type becomes a fact concrete, this because those work to thread of water with low reservoir, carting in a smaller flooded area, when compared with other machines of larger fall as Francis or Kaplan.

During the accomplishment of the turbine project a form used now to reduce these costs is through the numerical simulations, because this can help in the initial dimensioning of the problem, as in the validation of the project, once it doesn't exist more the need to do different reduced models, what would cart in a great spent amount.

In the numerical analysis of hydraulic turbines, some important works deserve eminence: Labrecque et al. (1996), makes a comparison between the numerical simulation of a complete hydraulic turbine and the simulation of the same through the interaction among their parts; Tamm et al. (2002), that makes the analysis of the applications and limits of a model of CFD (Computational Fluid Dynamic) specific for turbomachines.

The technical and scientific literature about turbines hydroelectric Bulb specifically is quite scarce. Although great part of projects of this machine type bases on methodologies known for axial machines, some specificity should be observed. A good description on Bulb turbines can be found in the work of (Henry, 1992).

As that machine is of the reaction type (or of total flow), the draft tube becomes indispensable. The flow in the draft tube presents a variety of phenomena just as rotation of the flow, collapse of vortex, change of circular geometry for rectangular in the traverse sections, etc. These phenomena usually interact in a no-lineal way, creating a great challenge for the modeling and the simulation of the flow. To minimize the losses of the flow it is necessary tools and models that take into account those phenomena and that help the old methods of remodeling of the draft tube.

With that concern various specialized laboratories in hydraulic turbines have been investing in the analysis of the draft tube. So, some works deserve prominence like Coelho et al. (2006a), that makes an expressive analysis of the influence of the swirl in the flow in diffusers; Coelho et al. (2006b) that accomplishes a transient study of the flow in conical diffusers; Grotjans (2001) that does the simulation of the draft tube using the software CFX; Puente et al. (2001), that proposes an automatic optimization of the draft tube; Bergiströn (2000), where were made verification and validation of the numerical simulations for flows in draft tubes; Japikse (2000), in his work, makes a correlation with the geometry, the swirl and the aerodynamic blockade in the inlet to determine the efficiency of a ring diffuser; and Avellan (2000) analyzes the influences of the boundary conditions in the flow in the draft tubes.

In face of the function and of the importance of the draft tube for a Bulb hydraulic turbine, this work intends for the modeling, simulation and characterization of the compound, unstable and three-dimensional flow of the draft tube. For this, the strategy used to model the turbulence will be RANS (Reynolds Averaged Navier-Stokes), using the turbulence model SST (Shear Stress Transport).

For the construction of the geometries, the commercial software SOLIDWORKS® is used. Two different geometries, the conventional draft tube and a new geometry are made proposed in this work. This work has the intention of decreasing in approximately 50% the length of the draft tube. It is adopted to analyze these two geometries for the comparison of results.

The analysis of that complex flow takes place through the software CFX-10 of ANSYS, where Cp, recovery pressure coefficient, is used for to determine the efficiency of the draft tube. Beyond of that, it is also analyzed the flow of the fluid, through the fields of speeds, stream lines, pressure variation in the draft tube, etc

2. GOVERNING EQUATIONS AND TURBULENCE MODELING

The governing equations of the analyzed flow are the equations of the continuity and the conservation of the movement, that they can be expressed in her medium form, respectively, as

$$\frac{\partial}{\partial x_j}\left(\overline{u_j}\right)=0; \tag{1}$$

$$\frac{\partial}{\partial t}\rho\left(\overline{u_j}\right)+\frac{\partial}{\partial x_k}\rho\left(\overline{u_j u_k}\right)=-\frac{\partial\overline{p}}{\partial x_i}+\frac{\partial\overline{\tau_{ij}}}{\partial x_i}+\frac{\partial}{\partial x_k}\rho\left(\nu_\tau\left(\frac{\partial\overline{u_j}}{\partial u_k}+\frac{\partial\overline{u_k}}{\partial u_j}\right)\right), \tag{2}$$

where u_i are the components of speed, ρ it is the specific mass, p is the pressure, τ_{ij} it is the tensor of viscous tensions and ν_τ is the turbulent viscosity, that it will be modeled inside of a closing context in first order using the model SST.

The model SST was created originally by Menter *et al* (1994). It doesn't act in itself a new turbulence model, but the composition among the others models, κ-ω and κ-ε. His operation is very simple, the transport equations for κ-ω are used in regions near the wall, and the transport equations for κ-ε are used in central part. The additional transport equations of this model are given for:

$$\rho\frac{\partial k}{\partial t}+\rho\overline{u_j}\frac{\partial k}{\partial x_j}=\frac{\partial}{\partial x_j}\left(\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right)+\widetilde{P}_k-\beta^*\rho\kappa\omega, \tag{3}$$

where:

$$P_k=\mu_t\frac{\partial u_i}{\partial x_j}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\Rightarrow P_k=\min(P_k,10\cdot\beta^*\rho\kappa\omega); \tag{4}$$

$$\rho\frac{\partial\omega}{\partial t}+\rho\overline{u_j}\frac{\partial\omega}{\partial x_j}=\frac{\partial}{\partial x_j}\left((\mu+\sigma_\omega\mu_t)\frac{\partial\omega}{\partial x_j}\right)+\alpha\rho S^2-\beta\rho\omega^2+2(1-F_1)\rho\sigma_{w2}\frac{1}{\omega}\frac{\partial\kappa}{\partial x_i}\frac{\partial\omega}{\partial x_i}, \tag{5}$$

where F1 is defined as:

$$F_1=\tanh\left\{\left\{\min\left[\max\left(\frac{\sqrt{\kappa}}{B^*\omega y},\frac{500\nu}{y^2\omega}\right),\frac{4\rho\sigma_{\omega2}\kappa}{CD_{\kappa\omega}y^2}\right]\right\}^4\right\}, \tag{6}$$

with

$$CD_{\kappa\omega}=\max\left(2\rho\sigma_{\omega2}\frac{1}{\omega}\frac{\partial\kappa}{\partial x_i}\frac{\partial\omega}{\partial x_i},10^{-10}\right), \tag{7}$$

and y it is the distance of the surface of no slipping.

The other constants are all originating from the models κ – ε e κ – ω with some adjustments and they are certain as: β^*= 0,09, α_1 =5/9, β_1 = 3/40, σ_{k1}= 0,85, $\sigma_{\omega1}$ = 0,5, α_2 =0,44, β_2 = 0,0828, σ_{k2} = 1 e $\sigma_{\omega2}$ = 0,856. (Menter, 2003).

It is noticed that the turbulence viscosity is calculated in this model as

$$\mu_t = \rho \frac{a_1 k}{\max(a_1\omega, \left(S_{ij}S_{ij}\right)^{\frac{1}{2}} F_2)}; \tag{8}$$

where $\left(S_{ij}S_{ij}\right)^{\frac{1}{2}}$ is a measured invariant of the tensor deformation rate and F2 is one of the combination functions and it is certain for:

$$F_2 = \tanh\left\{\left[\max\left(\frac{2\sqrt{\kappa}}{B*\omega y}, \frac{500\nu}{y^2\omega}\right)^2\right]\right\}. \tag{9}$$

The formulations of the mixture functions F1 and F2 are based in the distance until the wall and in the variables. The mixture functions have as characteristic the delimitation of areas the where each model will performance. Through the value found for the functions, the model will change the formulation in the transport equations, where the first mixture function (F2) is responsible for the change of models in the formulation of the turbulent viscosity and the other mixture function, F1 (Eq. 6) is responsible for the determination of the constants of the model, and for the change of models in the transport equation of ω.

3. NUMERICAL METHODOLOGY

In this work, the used method is of finite volumes (CFX), where the approximate equations are obtained through the balance of conservation of the evolutionary property (mass, quantity of movement, etc.) in the elementary volume. For the obtaining of the approximate equations, he breaks of the differential equation in her preservative form, integrating on the finite volume.

The discretization of the domain (Fig. 1) in volume of finite control takes place through a mesh, Fig. 2, where in that the knots are surrounded by the surfaces that understand the volume. Those knots are the responsible for the storage of all of the properties of the fluids and the solutions of the variables. In this study, the mesh has tetrahedral elements.

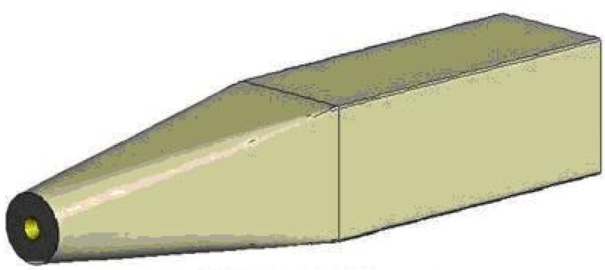


Figure 1a: Conventional draft tube.

Figure 1b: Optimized draft tube.

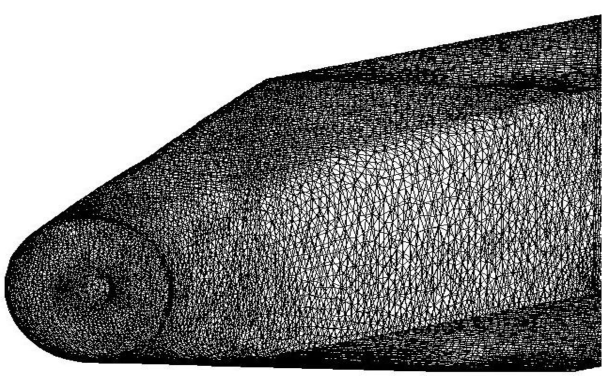


Figure 2a: Mesh in the Conventional draft tube

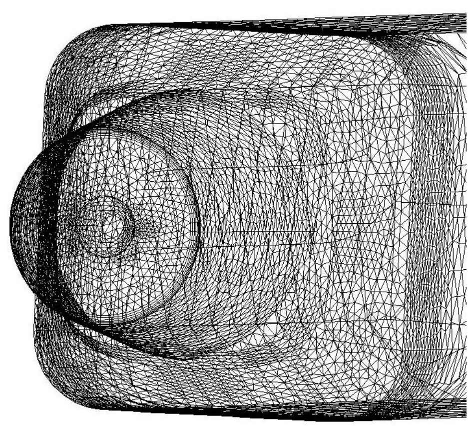


Figure 2b: Mesh in the Optimized draft tube.

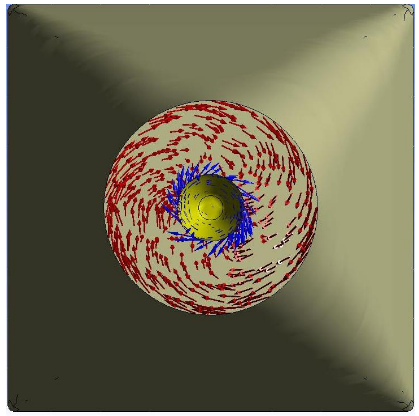


Figure 3: Boundary conditions in the inlet of the draft tube, where in red there are the exported data of the numerical simulation of the runner, and in blue the rotation of the machine. The intensity of the vectors is not being considered in that illustration.

The discretization of the not structured mesh described above was obtained starting from the use of a solid 3D. The creation of this solid occurs in the software SOLIDWORKS® and the implementation processing takes place in CFX-10. The meshes used in this work are composed of 1842492 elements (365922 nodes) for conventional draft tube (Fig. 2a) and 1568417 elements (302105 nodes) for the draft tube modified (Fig. 2b).

The computational domain used in this work is the draft tube and a prolongation of its final part. It takes place this prolongation so that eventual influences of the outlet conditions are not noticed, what could produce numerical instabilities.

For the boundary conditions of inlet (Fig. 3), results of the accomplished numeric simulations of part of the channel of water, distributor and runner of the Bulb machine are used. Those simulations were accomplished separately and they won't be discussed in this work. Other boundary condition used was the rotation of the machine, imposed in the cone, located in the inlet of the draft tube, show in blue in Fig. 3.

The boundary condition of outlet was imposed through the pressure, where the pressure of outlet is the static pressure in the reservoir. For the wall, the boundary condition is used of no slipping.

4. RESULTS AND DISCUSSIONS

For the realization of this work, a hydraulic machine is used that is approximately in operation for 20 years (the Centrale de Murray Lock and Dam, located in Arkansas, USA). The preference for this installation feels because in the initial period of this study, this was the machine with larger readiness of data. In the table below, some nominal data of this installation are mentioned.

Table 1: Characteristics of the Bulb turbine used in this work, located in Arkansas, USA.

H (m)	5,029	*n* (rad/s)	4,71
Q (m3/s)	430,00	Blades of Distributor	16
Pot (MW)	19,40	Blades of Runner	3

where H is the height, Q the mass flow, *Pot* the potency, and *n* the rotation of the machine.

In this work, to determine the efficiency of the draft tube through the coefficient of recovered pressure, Cp, set out in Eq. 10. This coefficient can be defined as the ratio among the pressure recovered by the available dynamic pressure.

$$Cp = \frac{p - p_1}{\frac{1}{2}\rho U^2} \tag{10}$$

where, p is the medium pressure in positions along the diffuser, p_1 is the static pressure in the inlet of the diffuser, p is the density of the water and U is the medium speed in the inlet of the diffuser.

The simulation of the draft tube is realized in two stages. The first is the conventional draft tube and the other is the new proposed geometry. They take place those two phases different with the intention of comparing the obtained results.

4.1. Conventional draft tube

In this stage of the work it is realized the simulation of the real draft tube, (Fig. 4). Analyzing the flow, Fig. 5, and the stream lines (Fig. 6) it is noticed that the flow doesn't present neither recirculation in the central part, nor stall in the wall.

This draft tube presents an excellent Cp (0,98). This was already supposed since a standard total angulation is used, with approximately 11º.

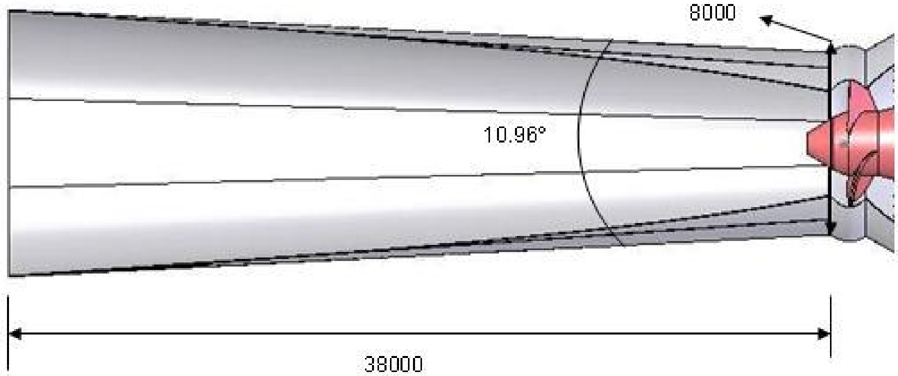


Figure 4: Dimensions of the draft tube, in mm.

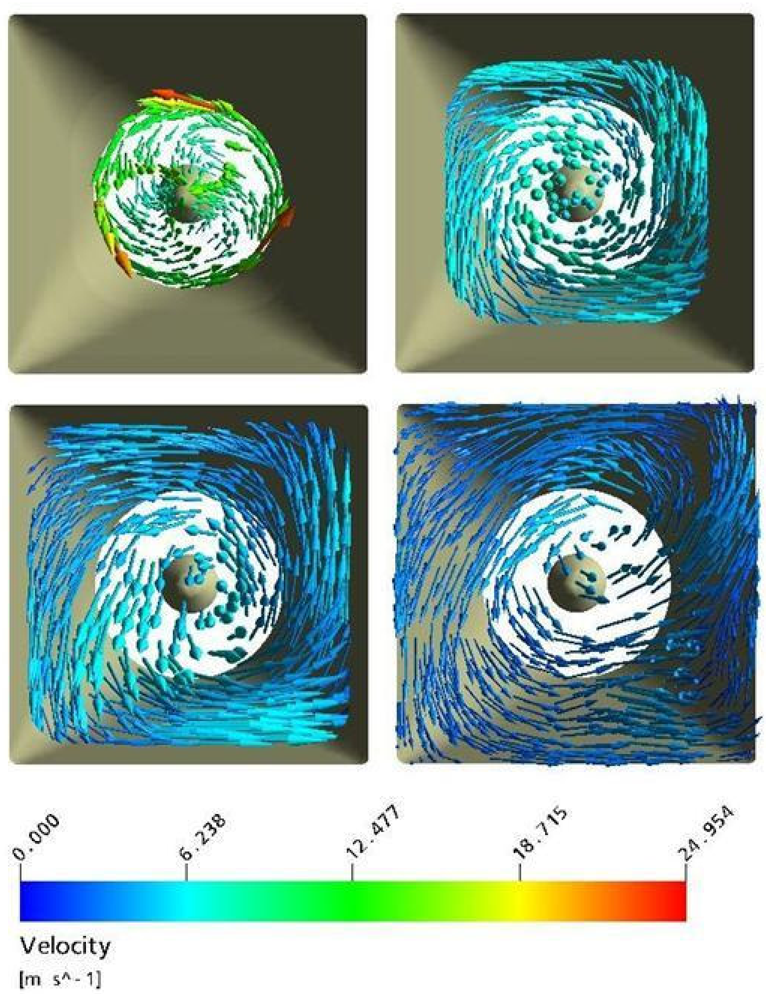


Figure 5: Evolution of the flow in the draft tube in the inlet, to 13m, to 26 and 38m.

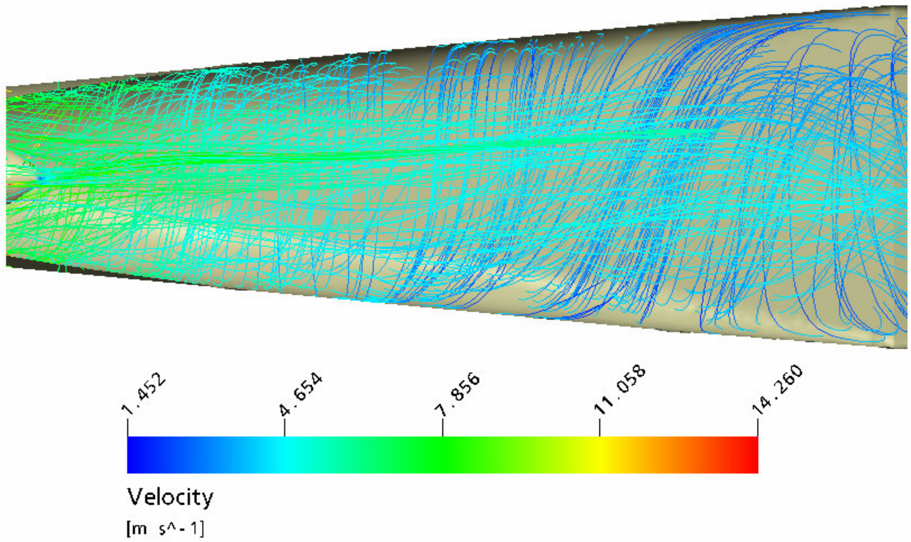


Figure 6: Stream Line.

4.2. Optimized draft tube

This is the central part of this work, where the optimization has the intention of reducing the length of the draft tube and like this, the cost of the construction of the turbine as a whole. The achieved decrease approximates of 50%, in other words, the everything of conventional suction possesses length of 38m while the new geometry possesses 20m.

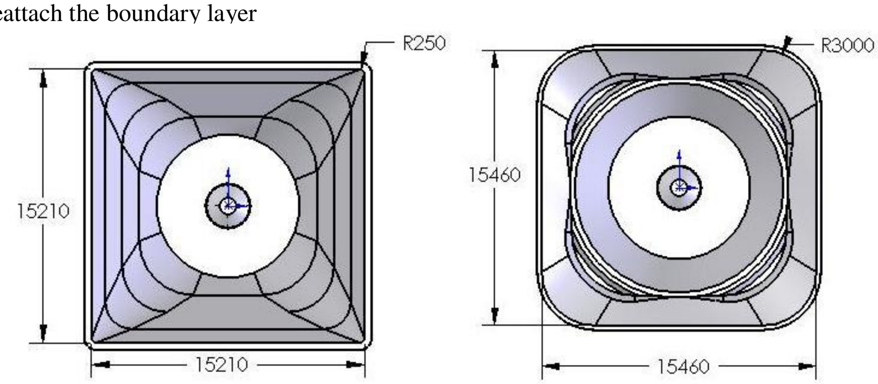
So that the comparison of results becomes satisfactory, the area ratio (A_1/A_2) is maintained constant. The area A_1 (inlet of the draft tube) doesn't present any alteration, while the geometric format of the outlet (A_2) is modified, as visualization presented in Fig. 7. This alteration takes place to eliminate the existent conical geometry in the original form, passing for a more circular format.

To maintain the same area ratio, the total angulation presents a considerable increment in its size, resulting in the stall of the boundary layer and/or recirculation in the central part of the diffuser (Fig. 8), depending on the intensity of the used swirl. Those phenomena aren't good, once they reduce the efficiency of the draft tube drastically.

Initially it intended to use a secondary inlet of fluid to solve both the problem of the stall of the boundary layer and the one of the recirculation in the central part, but that only secondary inlet was shown inefficient, because depending on the used format, the stall or the recirculation was eliminated, but never the two simultaneously.

Thus, two secondary inlets were inserted, Fig 9. The first located approximately to 15,3m (in the red color) of the inlet and with, mass flow same to 2% of the machine, with the intention of avoiding recirculation in the central part of the flow. The other inlet is placed to 16,8m (in blue) of the area of the inlet and it uses the flow of 8% of the total mass flow of the machine. So, this inlet has the function of avoiding the stall of the boundary layer.

The preference to insert those secondary inlets in the flow was made based in two simple principles. If that fluid injection goes too much approximate of the area of the inlet it is noticed the recirculation occurrence in the center of the diffuser. If the distance between the secondary inlet and the final area of the diffuser is decreased, the fluid won't have enough space to reattach the boundary layer



a) b)

Figure 7: Comparison among the conventional outlet the draft tube (a) and the outlet used in the optimization of the same (b). The values are in mm.

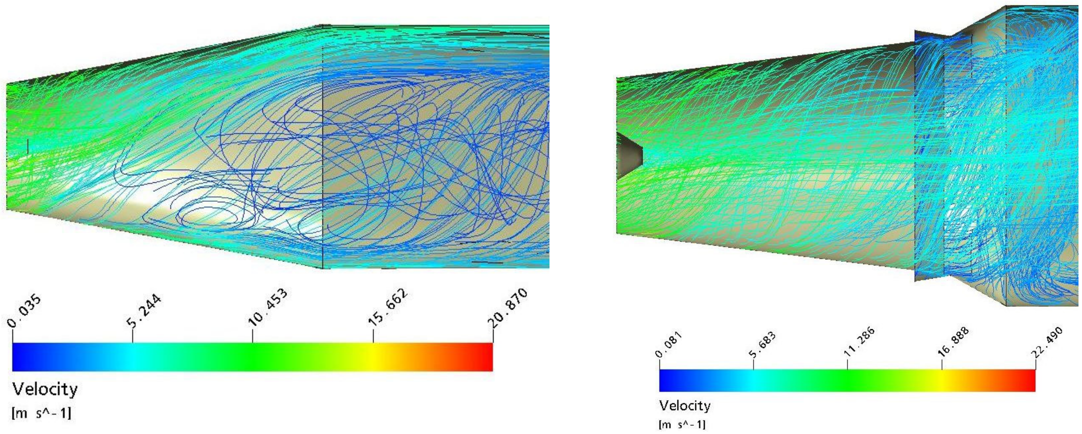


Figure 8: Stream line in the draft tube without secondary entrances and length equal of the optimized and stream line in the optimized draft tube.

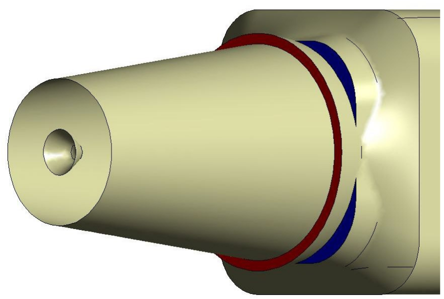


Figure 9: Illustration of the secondary inlets, where the first is shown in red color and the second in blue color.

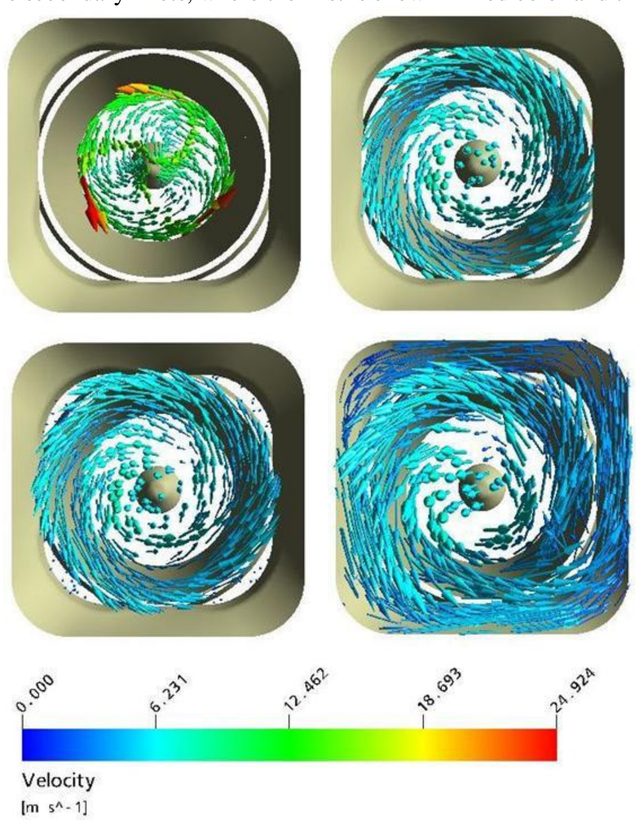


Figure 10: Evolution of the flow of the optimized draft tube, in the inlet, to 15m, to 17m and 20m.

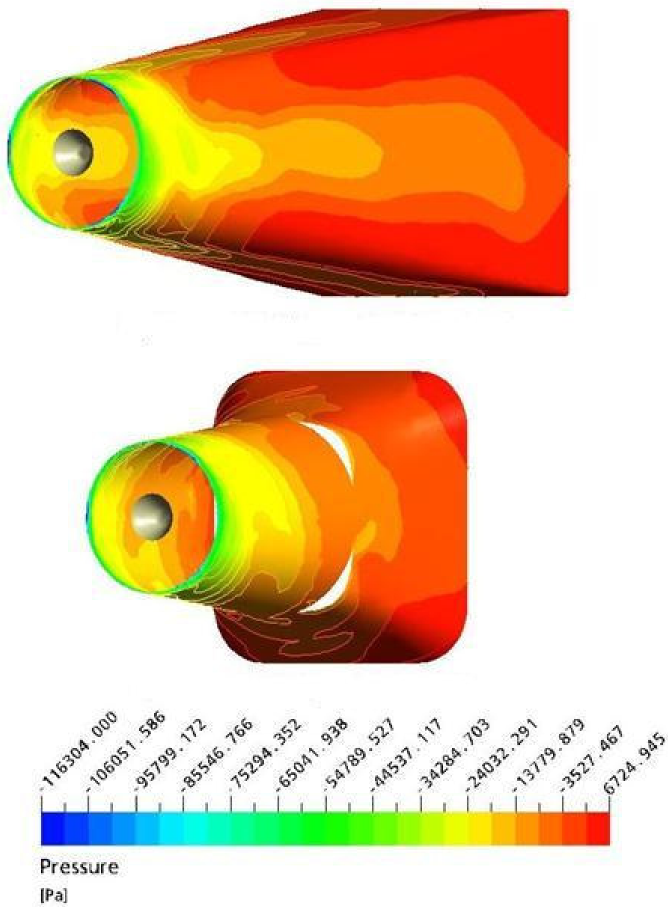


Figure 11: Pressure variation in the walls of the conventional draft tube and in the optimized one.

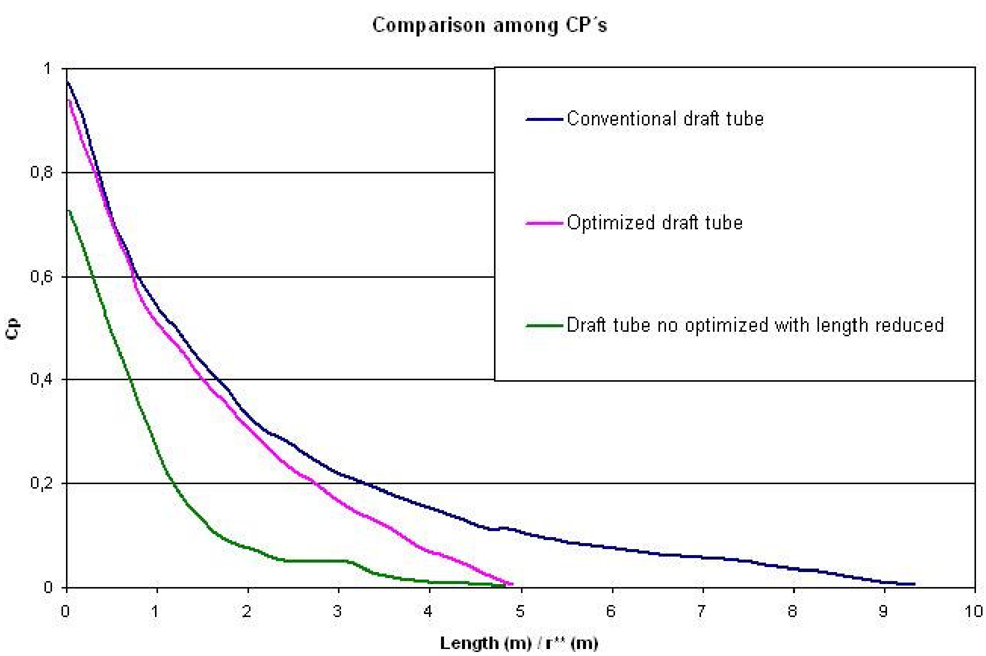


Figure 12: Comparison among Cp's.

Table 2: Comparison among Cp´s.

Geometry	Cp
Conventional draft tube	0,98
Optimized draft tube	0,95
Draft tube no optimized with length reduced (Fig. 8).	0,74

It has the intention of using the smallest possible amount of water. Like this, different flows are tested for the secondary inlets, 3, 5, 8 and 10% of the total flow of the machine. The one that obtained better result was the one of 10%. This is because the other flows were shown to have an inadequacy of amount of movement to do with that the boundary layer returned to glue to the wall and/or to reduce the existent recirculation in the central part of the draft tube.

The geometric formats chosen to do those secondary inlets are based in the geometric relationship of the final part of the draft tube. The first (circumference) injects water in the whole area of the diffuser, while the second injects fluid in four different areas, located in the places where the transition of the circular part for the square shows more critical (in what refers to the stall of the boundary layer). An illustration of those inlets is shown in Fig. 9.

In Fig. 10 it is shown the evolution of the flow in 4 traverse sections of the draft tube, respectively. It is noticed that the fluid flows without recirculation, proving that the secondary inlets of amount of movement and the used flow are accomplishing their job, making that neither recirculation in the central part, nor the stall on the wall of the draft tube occur.

Through Fig. 11, it is noticed the pressure variation on the walls of the geometries analyzed in this work. As it was already expected, the pressure is going to increase as we are approaching the final part of the draft tube. Through these data, a quantitative analysis of the pressure recovered by the diffuser is obtained. This information is shown valuable, once the larger the efficiency of the draft tube, the better the efficiency of the machine as a whole.

The variation of Cp along the length of the draft tube can be analyzed through Fig. 12, both for the conventional and optimized geometries. In this illustration, r** is the ray of the inlet of the draft tube and it is noticed that, even decreasing in approximately 50% the length of the draft tube, approximate efficiencies are achieved.

Tab. 2 shows the coefficient of recovered pressure, Cp, for the conventional geometry, the optimized one and a geometry with dimensions equal to the optimized one, however, without the secondary inlets. Comparing the results, it is noticed that without the secondary injection, the draft tube presents terrible results, Cp of the order of 74% of efficiency, while the conventional and optimized geometries present very close results.

An important subject refers to the constructive part. Initially, the creation of plumbing was cogitated for those secondary inlets, but that would end up making unfeasible its construction, so much for difficulties merely constructive, but mainly loss of turbining water. Later, it is noticed that with a drowning of the draft tube of approximately 1 meter, the necessary depth for that the total pressure is enough for the suction of the water by the secondary inlets. That means to say that there is no need to waste water diverting it of the rotor, water is already used.

5. CONCLUSION

Concluding, the hypothesis of reducing the length of the draft tube maintaining its efficiency approximately constant is shown valid. For that, two secondary inlets are used, that joined, inject about 10% of the flow of the machine. This new inlet becomes viable once the need to build plumbing is not necessary and to divert water of the machine, fluid is already used turbined, in other words, water is reused that already went by the runner. That is possible due to the depth of the drowning of the draft tube that, with about 1 m, already produces the necessary total pressure so that the inlet secondary pulls water inside of the draft tube.

6. ACKNOWLEDGEMENTS

This study was financed by the program P&D of ELETRONORTE S/A.

7. REFERENCES

Armfis S. W., Cho N. and Fletcher A. J., 1990, "Prediction of Turbulence Quantities for Swirling flow in Conical Diffusers", American Institute of Aeronautics and Astronautics, Vol. 23, No 3, pp. 453-460.

Avelan F., Mauri S. and Kueny J. L., 2000, "Numerical Prediction of the Flow in a Turbine Draft Tube Influence of the Boundary Conditions", ASME 2000 Fluids Engineering Division Summer Meeting, Boston, Massachusetts, USA, 7 p.

Azad R. S. and Kassab, S.Z., 1989, “Turbulent flow in a conical diffuser: Overview and implications”, Americal Institute of Phisics, A 1 (3), pp. 564 – 573

Balanço Energético Nacional, 2005, Ministério de Minas e Energia.

Berstron J. and Gebart R., 1999, “Estimation of Numerical Accuracy for the Flow Field in a Draft Tube”, International Journal of Numerical Methods for Heat & Fluid Flow, Vol. 6, No. 4 pp. 472-486

Clausen, P. D., Kish, S. G. and Wood D. H., 1993, “Measurement of a Swirling Turbulent Boundary Layer Developing in a Conical Diffuser”, Experimental Thermal and Fluid Science, Vol. 6, pp. 39-48.

Coelho, J. G., 2006, Dissertação de Mestrado, “Estudo Numérico de Tubos de Sucção de Turbinas Hidráulicas Tipo Bulbo”, Universidade de Brasília, 110 p.

Coelho, J. G., Brasil, A. C. P. J., Simulação Numérica da Influência do Swirl em Difusores, 11th Brazilian Congress of Thermal Sciences and Engineering, ENCIT 2006, ABCM, Brasil.

Coelho, J. G., Noleto, L. G., e Brasil, A. C. P. J., Escoamentos Turbulentos em Difusores Cônicos – simulações Transientes, 5ª Escola de Primavera de Transição e Turbulência, EPTT 2006, ABCM, Brasil.

Dixon S. L., 1998, “Fluid Mechanics and Thermodynamics of Turbomachinery”, Butterworth-Heinemann, England.

Grotjans H., 2001, “Simulation of Draft Tube Flow with CFX”, Second ERCOFTAC Worshop on Draft Tube Flow, Vattenfall Utvercling AB, Älvkarleby, Sweden.

Henry P., 1992, “Turbomachines Hydrauliques”, Presses Polytechniques et Universitaires Romandes, França, 407p.

Iaccarino G., 2000, “Prediction of the Turbulent Flow in a Diffuser with Commercial CFD Codes”, Annual Research Briefs 2000, pp. 271-279.

Japikse D,2000, “Correlation of Annular Diffuser Performance with Geometry, Swirl, and Blockage”, 11Th Thermal ans Fluid Analysis Worshop, Cleveland, Ohio.

Labrecque Y., Sabourin M. and Deschênes C., 1996, “Numerical Simulation of a Complete Turbine and Interaction Between Components”, Modeliling, Testing & Monitoring for Hydro Poweplants, Lausanne, Switzerland.

Maliska, C. R., 2002, “Transferência de Calor e Mecânica dos Fluidos Computacionais”, Livros Técnicos e Científicos Editora S.A., Rio de Janeiro, 424 p.

Menter F. R, 1994, “Two-equation eddy-viscosity turbulence models for engineering applications”, AIAA Journal Vol. 32, pp 1598-1605.

Menter F. R., Kuntz, M., e Langtry, R., 2003, “Ten years of industrial experience with the SST turbulence model”, Turbulence, heat and Mass transfer 4,8 p.

Puente L. R., “Reggio M. and Guibault F. 2001, Automatic Shape Optimization of a Hydraulic Turbine Draft Tube”, Departament of Mechanical Engineering, 6 p.

Tamm A., Gugau M and Stoffel B., 2002, “Experimental and 3D Numerical Analysis of the Flow Field in Turbomachines Part I”, International Congress on Quality Assessment of Numerical Simulations in Engineering, University of Conception, Chile.

Wilcox, D. C., 1993, “Turbulence Modelling for CFD”, DWC Industries Inc., La Canada, 460p.

White F. M., 1994, “Fluid Mechanics”, McGraw-Hill, Inc., London, 736p.

Zulcy S, abd Bran R., 1987, “Máquinas de Fluxo: Turbinas, Bombas, Ventiladores”, Ao Livro Tecnico, Rio de Janeiro, 262p.

8. RESPONSIBILITY NOTICE

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