INTERNATIONAL JOURNAL OF
ENERGY AND ENVIRONMENT

Volume 4, Issue 4, 2013 pp.539-548
Journal homepage: www.IJEE.IEEFoundation.org

Numerical simulation of draft tube flow of a bulb turbine

Coelho J. G.[1], Brasil Junior A. C. P.[2]

[1] Federal University of Triângulo Mineiro, Institute of Technological and Exact Sciences, Avenida Doutor Randolfo Borges Junior, 1250 – Uberaba – MG, Brazil.
[2] University of Brasilia, Department of Mechanical Engineering, Campus Darcy Ribeiro, Brasilia – DF, Brazil.

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-14), using the turbulence model SST, that allows a description of the field fluid dynamic 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; Simulation.

1. Introduction
In the current days, where the environmental impacts need to be reduced, the use of machines Bulb type becomes a fact concrete, this because this machine works with little depth, requiring a smaller area flooded, when compared with other machines of larger fall as Francis or Kaplan.
During the accomplishment of the turbine project a form used to reduce the 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. [1], 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. [2], 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 Henry [3].
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 Armfis et at. [4] where it is determined the turbulence quantities for swirling flow in conical diffusers; Clausen et al. [5], that makes an expressive analysis of the influence of the swirl in the flow in diffusers; Bergiströn [6], where were made verification and validation of the numerical simulations for flows in draft tubes; Japikse [7], 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; Avellan et al. [8] analyzes the influences of the boundary conditions in the flow in the draft tubes; Grotjans [9] that does the simulation of the draft tube using the software CFX; Puente et al. [10], that proposes an automatic optimization of the draft tube and Coelho [11] what makes a numerical study of the draft tubes of the Bulb hydraulic turbines;
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-14, 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

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

$$\frac{\partial}{\partial t}\rho\left(\overline{u}_j\right)+\frac{\partial}{\partial x_k}\rho\left(\overline{u_j u_k}\right)=-\frac{\partial \overline{p}}{\partial x_i}+\frac{\partial \overline{\tau_{ij}}}{\partial x_j}+\frac{\partial}{\partial x_k}\rho\left(v_t\left(\frac{\partial \overline{u_j}}{\partial u_k}+\frac{\partial \overline{u_k}}{\partial u_j}\right)\right) \tag{2}$$

where u_i are the components of speed, ρ it is the specific mass, p is the pressure, and τ_{ij} it is the tensor of viscous tensions v_t 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. [12]. 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

$$\rho\frac{\partial k}{\partial t}+\rho\overline{u}_j\frac{\partial k}{\partial x_j}=\frac{\partial}{\partial x_j}\left(\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right)+P_k-\beta*\rho k\omega \tag{3}$$

where

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

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

where F_1 is defined as

$$F_1=\tanh\left\{\left\{\min\left[\max\left(\frac{\sqrt{k}}{\beta^*\omega y'}\frac{500v}{y^2\omega}\right)\frac{4\rho\sigma_{\omega^2}k}{CD_{k\omega}y^2}\right]\right\}^4\right\} \tag{6}$$

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

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

The other constants are all originating from the models $\kappa-\varepsilon$ and $\kappa-\omega$ with some adjustments and they are certain as: β^*= 0.09, α1 =5/9, β1 = 3/40, σk1= 0.85, σω1 = 0.5, α2 =0.44, β2 = 0.0828, σk2 = 1 and σ_{ω^2} =0.856. [13].
It is noticed that the turbulence viscosity is calculated in this model as

$$\mu_t=\rho\frac{\alpha_1 k}{\max\left(\alpha_1\omega,\left(S_{ij}S_{ij}\right)^{\frac{1}{2}}F_2\right)}, \tag{8}$$

where $\left(S_{ij}S_{ij}\right)^{\frac{1}{2}}$ is a measured invariant of the tensor deformation rate and F_2 is one of the combination functions and it is certain for

$$F_2=\tanh\left\{\left[\max\left(\frac{2\sqrt{k}}{\beta^*\omega y'}\frac{500v}{y^2\omega}\right)^2\right]\right\} \tag{9}$$

The formulations of the mixture functions F_1 and F_2 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 (F_2) is responsible for the change of models in the formulation of the turbulent viscosity and the other mixture function, F_1 (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 (Figure 1) in volume of finite control takes place through a mesh, 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.
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-14. The meshes used in this work are composed of 1842492 elements

(365922 nodes) for conventional draft tube (Figure 1a) and 1568417 elements (302105 nodes) for the draft tube modified (Figure 1b).
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 [8].
For the boundary conditions of inlet (Figure 2), 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 Figure 2.
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.

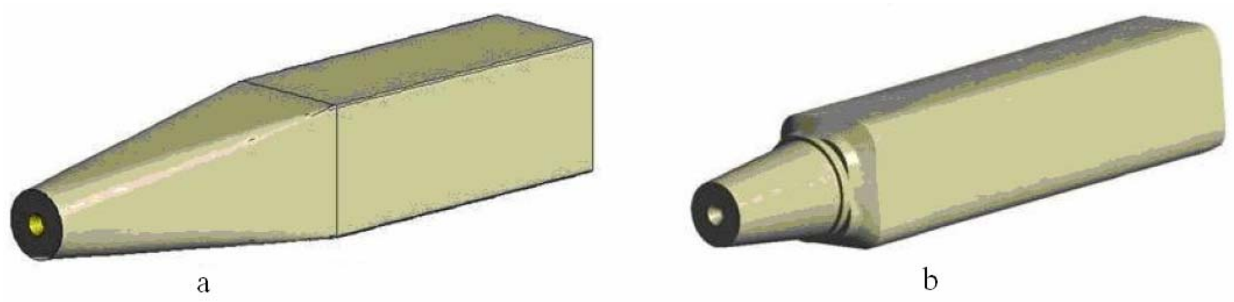

Figure 1. Conventional draft tube (a) and Optimized draft tube (b)

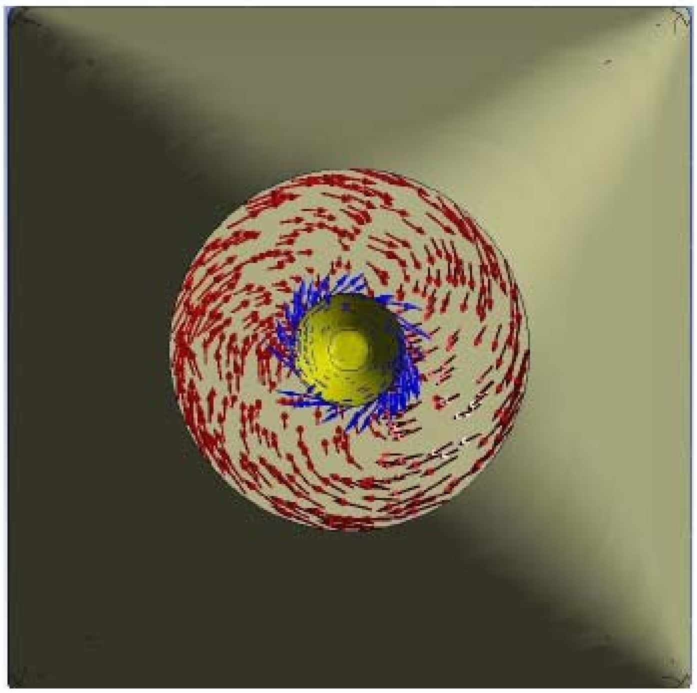

Figure 2. Boundary conditions in the inlet of the draft tube, where in red there are the exported data of the numerical simulation of the runner, and in blue the rotation of the machine. The intensity of the vectors is not being considered in that illustration

4. Results and discussions
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.

$$C_p = \frac{p - p_1}{0.5\rho U^2} \tag{10}$$

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

For validation of the results, it is compared with the data shown by Dixon et al. [14], where the recovered pressure is shown as a function of area ratio and the dimensionless length of the diffuser. This comparison is shown in Table 1 and Figure 3.

Table 1. Comparison between the experimental (Dixon et al, [14]) and numerically recovered pressure for the various diffusers

L (m) / R1 (m)	Numerical *Cp*	Experimental *Cp*
5.04	0.50	0.50
6.06	0.60	0.60
7.6	0.69	0.68
10.17	0.75	0.73
15.26	0.81	0.78

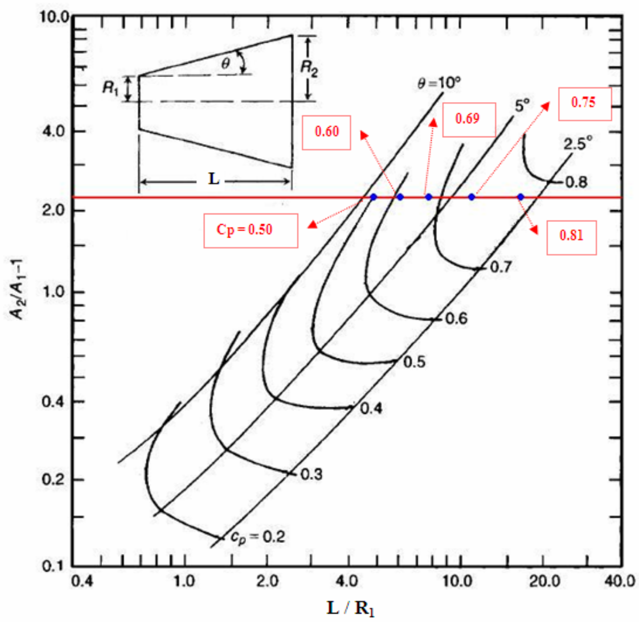

Figure 3. Comparison between Cp recovered numerically and experimental data [14]

For the realization of this work, a hydraulic machine is used that is approximately in operation for 20 years (the Murray Central Lock and Dam, located in Arkansas, USA) and in the Table 2, some nominal data of this installation are mentioned.

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

Table 2. Characteristics of the Bulb turbine used in this work, located in Arkansas, USA [3]

H (m)	5.029
Q (m^3/s)	430.00
Pot (MW)	19.40
n (rad/s)	4.71
Blades of Distributor	16
Blades of Runner	3

4.1 Conventional draft tube

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

This draft tube presents a good Cp (0.98). This was already supposed since a standard total angulation is used, with approximately 11º.

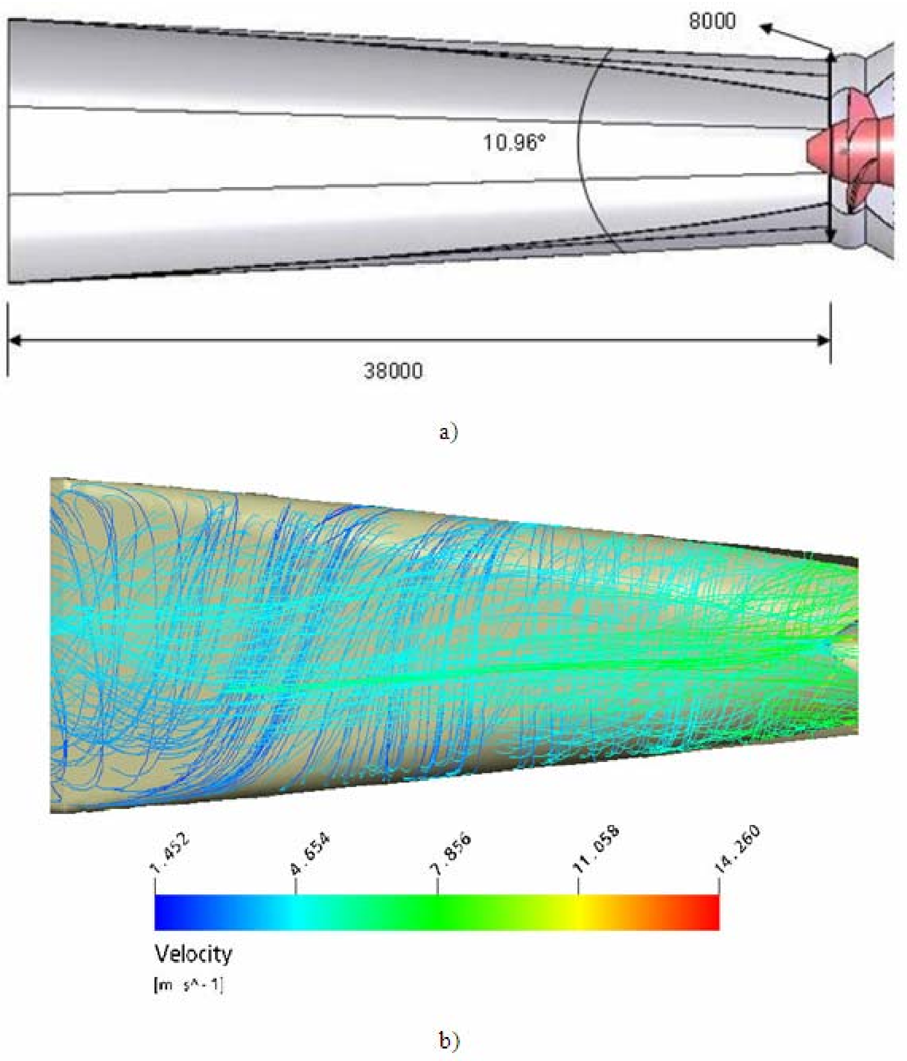

Figure 4. Dimensions of the draft tube, in mm (a) and the stream line (b)

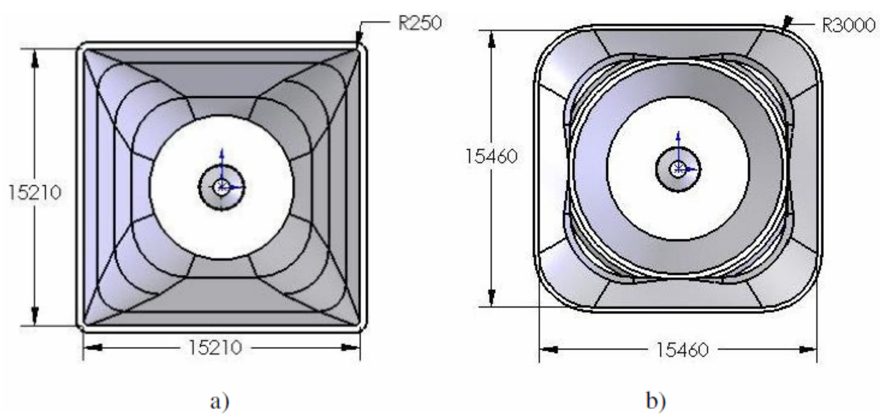

Figure 5. Comparison among the conventional outlet the draft tube (a) and the outlet used in the optimization of the same (b). The values are in mm

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 (A1/A2) is maintained constant. The area A1 (inlet of the draft tube) doesn't present any alteration, while the geometric format of the outlet (A2) is modified, as visualization presented in Figure 5. 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 with (Figure 6), 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, Figure 7. 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 massflow 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

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 Figure 7.

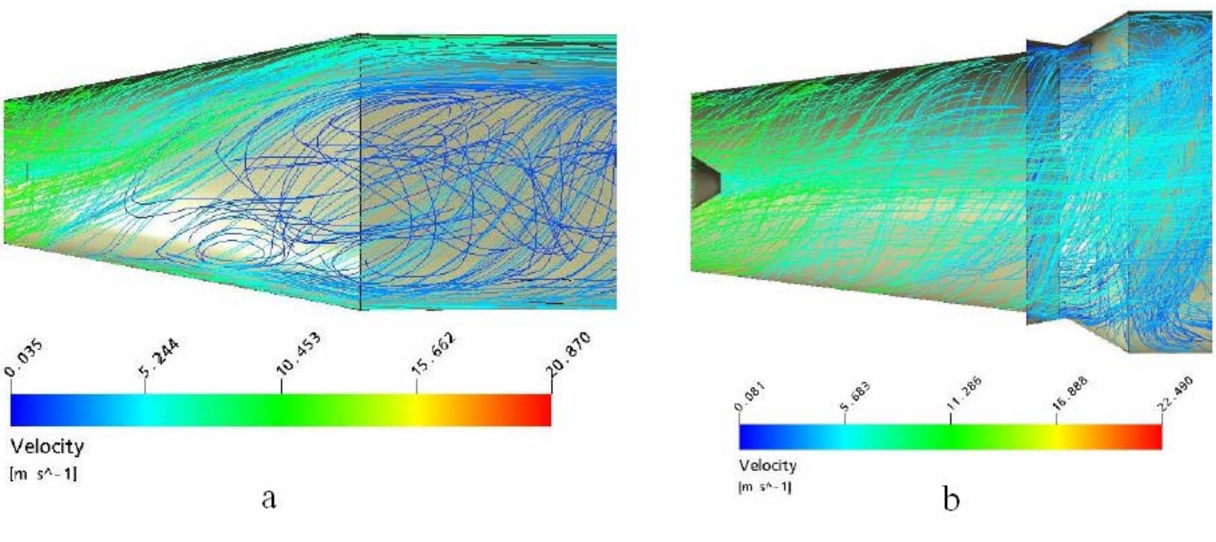

Figure 6. Stream line in the draft tube without secondary entrances and length equal of the optimized (a) and stream line in the optimized draft tube (b)

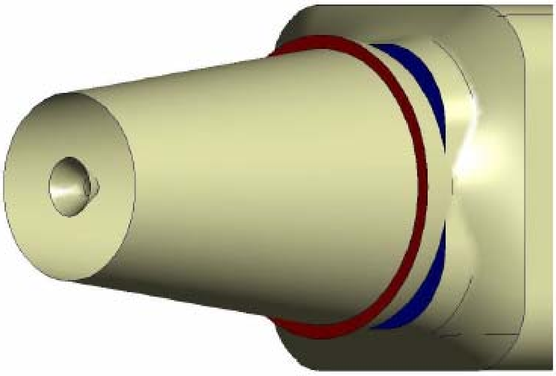

Figure 7. Illustration of the secondary inlets, where the first is shown in red color and the second in blue color

Figure 8 shows the evolution of the flow in 4 traverse sections of the draft tube. 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.

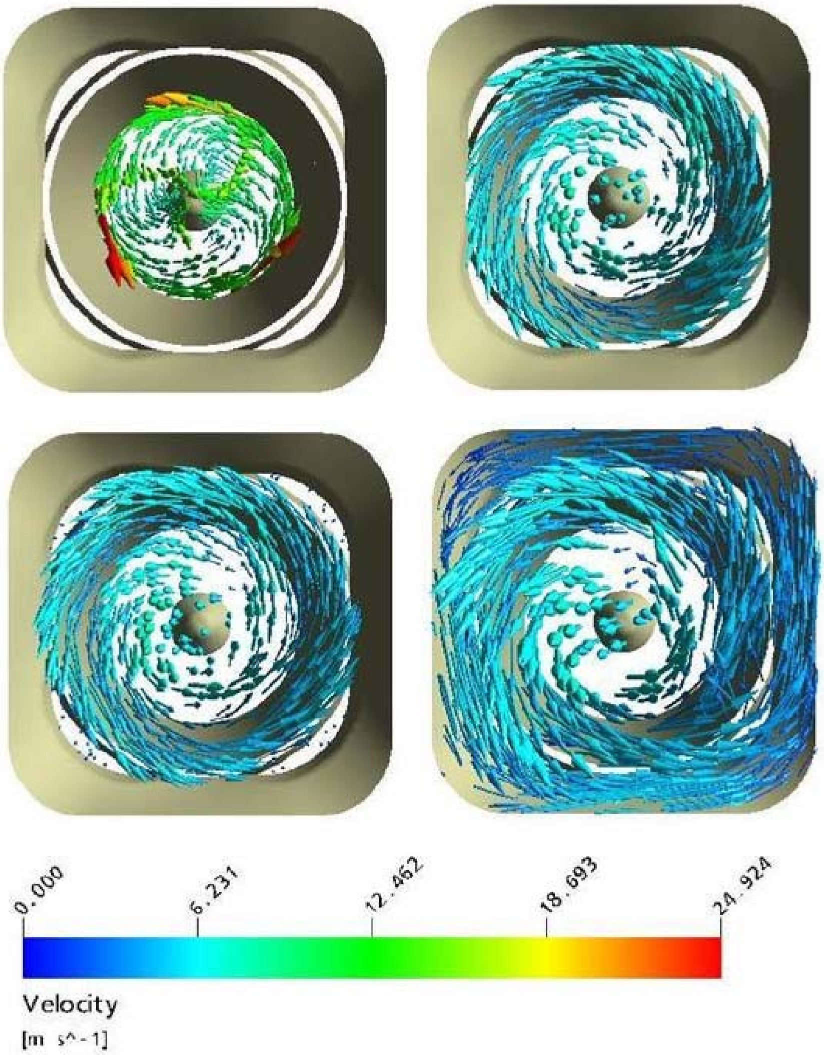

Figure 8. Evolution of the flow of the optimized draft tube, in the inlet, to 15m, to 17m and 20m

The variation of Cp along the length of the draft tube can be analyzed through Figure 9, 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.

Table 3 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.

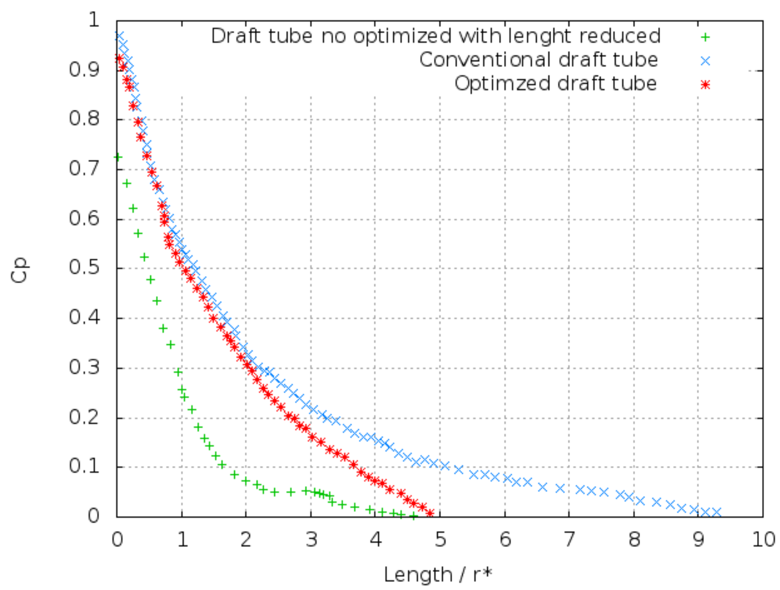

Figure 9. Comparison among Cp's, where r* is the ray of the inlet of the draft tube

Table 3. Comparison among Cp´s

Geometry	Cp
Conventional draft tube	0.98
Optimized draft tube	0.95
Draft tube no optimized with lenght reducedt (Figure 6)	0.74

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.

References

[1] 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

[2] 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.
[3] Henry P., 1992, Turbomachines Hydrauliques, Presses Polytechniques et Universitaires Romandes, França, 407p.
[4] 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
[5] 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.
[6] 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.
[7] Japikse D,2000, Correlation of Annular Diffuser Performance with Geometry, Swirl, and Blockage, 11Th Thermal ans Fluid Analysis Worshop, Cleveland, Ohio.
[8] 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.
[9] Grotjans H., 2001, Simulation of Draft Tube Flow with CFX, Second ERCOFTAC Worshop on Draft Tube Flow, Vattenfall Utvercling AB, Älvkarleby, Sweden.
[10] Puente L. R., Reggio M. and Guibault F. 2001, Automatic Shape Optimization of a Hydraulic Turbine Draft Tube, Departament of Mechanical Engineering, 6 p.
[11] Coelho, J. G., 2006, Dissertação de Mestrado, Estudo Numérico de Tubos de Sucção de Turbinas Hidráulicas Tipo Bulbo, Universidade de Brasília, 110 p.
[12] Menter F. R, 1994, Two-equation eddy-viscosity turbulence models for engineering applications, AIAA Journal Vol. 32, pp 1598-1605
[13] Menter F. R., Kuntz, M., and Langtry, R., 2003, Ten years of industrial experience with the SST turbulence model, Turbulence, heat and Mass transfer 4-8 p.
[14] Dixon S. L., 1998, Fluid Mechanics and Thermodynamics of Turbomachinery, Butterworth-Heinemann, England.

Coelho is adjunct professor at the Federal University of Triangulo Mineiro, Department of Mechanical Engineering and has experience in Numerical Simulation of fluids, ranging from the simulation with commercial software as the use of software development itself. These studies include the turbulent flow, flow machines and environmental flows. More information: http://lattes.cnpq.br/0250577148113150.
E-mail address: josegustavo@icte.uftm.edu.br

Brasil Junior is currently associate professor at the University of Brasilia, with seating in the Department of Mechanical Engineering. In the area of mechanical sciences the topics of interest are: Finite elements in fluids, turbulent flows, turbo-machinery and environmental flows. Alternatively the professor works in the area of sustainable development, like Amazon and Pantanal biomes. More information: http://lattes.cnpq.br/3642213701314265.
E-mail address: brasiljr@unb.br