NUMERICAL STUDY OF A BULB HYDRAULIC TURBINE

Cláudia Cristina Barbosa dos Santos, cau.cris@terra.com.br
José Gustavo Coelho, josegustavo@unb.br
Antonio C. P. Brasil Junior, brasiljr@unb.br
Universidade de Brasília
Campus Universitário Darci Ribeiro
Faculdade de Tecnologia
Departamento de Engenharia Mecânica
Laboratório de Energia e Ambiente
CEP: 70910-900 - Brasília - DF

Abstract. *In this paper a numerical study of the turbulent flow inside a bulb hydraulic turbine is presented. The hydraulic turbine is the representation of the future installation of Belo Monte's Hydro Power Plant (CHE-BM). The entire machine will be simulated, starting from the water inlet, crossing by the distributor, runner and blades, and the draft tube. All those parts were simulated separately, and assembled. The separated simulation intent to obtain the runner's diameter size. The assembled simulation intents the comparing the computational cost with the separated simulation. The geometry was generated using the SOLIDWORKS CAD commercial software. The simulations will be conducted by ANSYS CFX commercial software, with tetrahedrical grids. The SST turbulence model will be used due to its capability in previewing the near wall flow.*

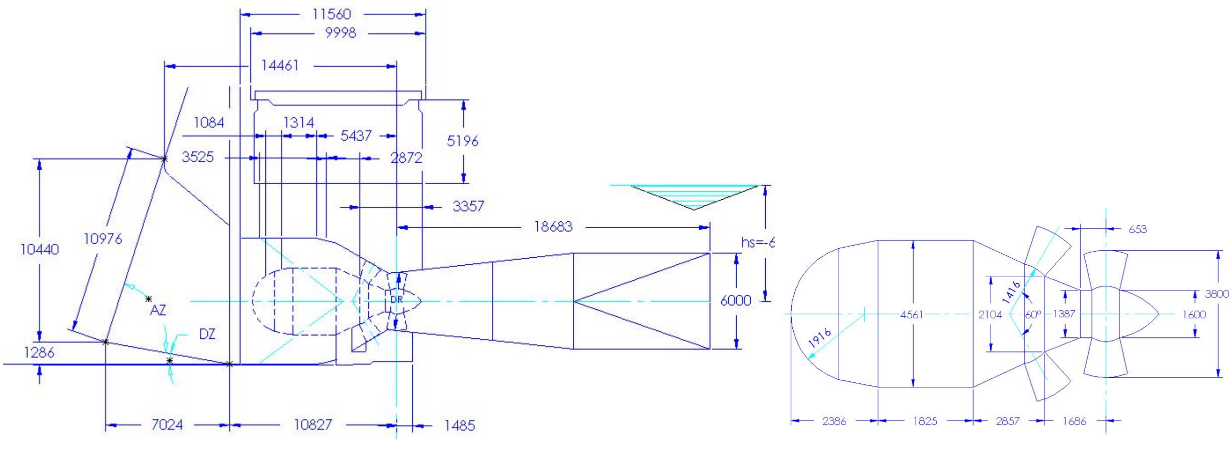
Keywords: *Bulb turbine, SST turbulence model, Separated simulation, Assembled simulation*

1. INTRODUCTION

The energy that provides the needs of the society move industries, transports, business e others economics sectors in the country. According to the National Energetic Balance of 2005, concerning the industry of electrical energy production, one can appoint to the fact that the electricity generation was raised from $11\ GW$ in 1970 at the seventies to $90,7\ GW$ in 2004. The Brazilian hydroelectric capability is $69\ GW$ at the same year, which represents around 26,6% of the total potential of Brazilian hydro power.

With the intention of attending this increase in the electric power demand, Brazilian electrical planning is implementing the Belo Monte's Hydro Power Plant (CHE-BM), where one of two power houses, located around the city of Altamira, will use a whole of seven bulb turbines, with production of $25,9\ MW$ for unit, totalizing $181,3\ MW$ when all turbines are working. This kind of hydraulic turbine does not need river blocking. Therefore, the size and environmental requests for dam construction are low. The immediate consequence is a less flooded area.

One of the initial phases of the project as this is the validation of the turbine dimensions as a function of the desired power output, that in this work is $25,9\ MW$. It intends reach this income using the preliminary geometry description of the project, supplied by the preliminary project S/A. The geometry description can be observed in Fig. 1.



(a) (b)

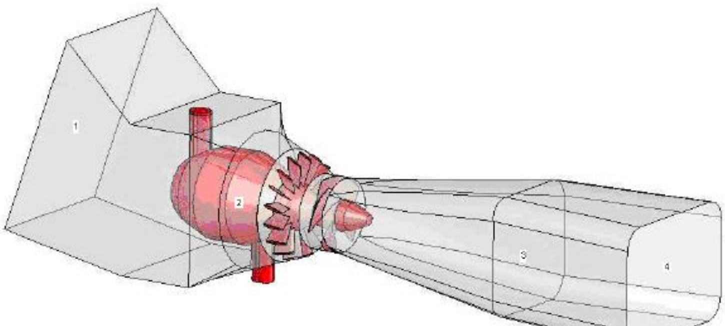
Figure 1. Geometry Description

One of the validation forms, among most current, is the numeric study of hydraulic turbines, as demonstrated in some important papers. Among these are: Labrecque et al. (1996) which, using software TASCflow, compares the numeric simulation in whole hydraulic turbine wiht the interaction obtained in the numeric simulation of the turbine parts, distributor, runner and draft tube. Massé et al. (1999), "define" the numeric simulation as being a tool that can be used for improve the performance of hydraulic turbines. In this paper, one performs an analysis of the turbine in operation and it proposes suggestions, which has function of improve the turbine performance. Tamm et al. (2002), it does an analysis of the uses and of the limits of a model in CFD (Computational Fluid Dynamic), specific models for turbo machines, where runner remains stopped and it does whole domain rotate, it is called frozen - rotor. In this paper, the author works with compressible and incompressible flow, in steady state.

Different possibilities of modeling e of simulation are much explored in the papers of Moura (2003) and Mauri (2002a), besides being introduced several papers about numerical techniques for the study of the flow, and as it is possible to revert in performance increase for machine.

For validation this project it uses the commercial software CAD SOLIDWORKS for the generation of geometry and the code ANSYS CFX-10 for the implementation and analysis of the hydrodynamic results. To model the phenomenon of the turbulence it uses the strategy RANS (Reynolds Averaged Navier-Stokes), URANS (Unsteady Reynolds Averaged Navier-Stokes) and the SST model (Shear Stress Transport). It resorts to this turbulence model by its great capacity of determining the boundary layer detachment, even in the presence of adverse gradient of pressure.

The strategy used for numeric simulation was, first, the simulation of the machine in her parts. After, was being the simulation with the whole machine, starting from the water inlet, crossing by the distributor, runner and blades, and the draft tube. This methodology was adopted with the plan of reproduce the boundaries conditions realistically and with the plan to compare both the simulations, avoiding thus the necessity to realize distinct simulations by parts. The Bulb Turbine used in this work is shown in Fig. 2.



1. Water inlet
2. Turbine
3. Draft tube
4. Prolongation

Figure 2. Bulb Turbine Components

2. TURBULENCE MODEL AND NUMERIC METODOLOGY

The equations that govern the analyzed flow are the continuity and the Navier-Stokes equation, which they can be express in their average form, respectively, as:

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

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

where u_i are the components of the speed, ρ is the specific mass, p is the pressure, σ_{ij} is the viscous stress tensor, and ν_t is the eddy viscosity, which will be modeled into the context of closure in first order using the model SST.

This model was proposed by Menter et al (1994) and it uses the equation of two others models, the $\kappa-\varepsilon$ and the $\kappa-\omega$. Its formulation is very simple, in extern flow region it uses the formulation of the $\kappa-\varepsilon$ model, and where this model is little efficient, in the region around wall, it uses the transport's equations of the $\kappa-\omega$ model. The transport's equations of the SST model are:

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

where ρ ,as said before, is the specific mass, μ is the viscosity and:

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

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

where F_1 is defined as:

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

with:

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

and y is the distance from surface of not sliding.

Others constants are originated of the $\kappa-\varepsilon$ and $\kappa-\omega$ models with some adjusments e are determined as: $\beta=0,09$, $\alpha_1=5/9$, $\beta_1=3/40$, $\sigma_{k1}=0,85$, $\sigma_{\omega 1}=0,5$, $\alpha_2=0,44$, $\beta_2=0,0828$, $\sigma_{k2}=1$ and $\sigma_{\omega 2}=0,856$. (Menter, 2003).

It is observed the eddy viscosity is calculated in this model as:

$$\mu_t=\rho\frac{a_1k}{\max(a_1\omega,(S_{ij}S_{ij})^{\frac{1}{2}}F_2)} \tag{8}$$

where $(S_{ij}S_{ij})^{\frac{1}{2}}$ is a invariant measure of the rate of strain tensor and F_2 is one of the combination's functions and it's determined by:

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

About the values found for the functions F_1 and F_2, the model will be move the formulation in the transport equations, where the mixer function F_1 (Eq. 6) is responsible by the determination of the model constants e by the change of the models in the transport equation ω, and F_2 (Eq. 9) is responsible by change of the models in the formulation of the eddy viscosity.

The software CFX - 10 adopts the methodology finite elements, where it approaches the equations trough of the conservation balance of the evolutionary property in the elementary volume. The stepping stone of the differential equation integrating its about a finite volume. This volume is discretized in a mesh, which is composed by an assembly of tetraedrical elements not structured. The mesh used in this work is composed by 3101862 elements (731687 nodes).

The domain used in this work (Fig. 2) has a prolongation in final part of the draft tube. This prolongation has as purpose of decrease the eventual influences caused by the boundaries conditions, which can produce numeric instability.

The boundaries conditions used in this work are basing in the geometry description supplied in the preliminary project, more specifically in the height for amount, which is $19,1\ m$, and in the distance between the machine axis and the free surface (Fig. 1), which is $6\ m$, what it supplies a nominal fall of $13,1\ m$. This way, it determines the total pressure in function of the fall (Eq. 10) and found the values that will be imposed in the inlet, the part of the water inlet, and in the outlet of the turbine, prolongation of the draft tube.

$$P=\rho gH \tag{10}$$

where g is the acceleration of the gravity and H is the fall. This way, knows that $P_{inlet}=187371\ Pa$, because $H_{upstream}=19,1\ m$, and $P_{oulet}=58860\ Pa$, once that $H_{downstream}=6\ m$.

3. RESULTS AND DISCUSSIONS

First, it simulated the water inlet, crossing by the distributor and runner. With the results that were obtained by simulation of the distributor and runner could be observed that this part of machine needs the more attention , because its results are complectly in disagree with original dimensions of preliminary project.

According to the preliminary project, the runner introduces a $3,8\ m$ as external diameter. But, with the qualitative and quantitative results obtained through of the visits to the Brazilian plans, which used Bulb turbine, allowed to observe that in this project the diameter is below the necessary. Among the plans visited are UHE Canoas 1 , UHE Igarapava and UHE Ourinhos, besides several PCH, as PCH Bortolan, PCH Primavera and PCH Culuene. Some informations about this hidroelectric and hidroelectric in study can be seen in Tab. 1.

Table 1. Visited Plants and CHE-BM

Plant	Turbine rated head	Diameter	Flow rate	Power (per machine)
UHE - Igarapava	17,1 m	6 m	275 m^3/s	43,6 MW
UHE - Canoas I	17 m	4,7 m	181 m^3/s	27,5 MW
UHE - Belo Monte	13,1 m	5,6 m	326,09 m^3/s	25,4 MW
UHE - Ourinhos	12,7 m	3,6 m	145m^3/s	14,8 MW
PCH - Primavera do Leste	10 m	1,5 m	13m^3/s	1,2 MW
PCH - Culuene	12,2 m	1 m	6 m^3/s	0,64 MW
PCH - Bortolan	12,7 m	1,15 m	7 m^3/s	0,715 MW

From the Fig. 3, can be observed a recirculation and shear lines detachment in the runner. This can indicate or that proposed mass flow is elevated, or the position that its blades are inadequate. This can certificated too, in the visits to brazilians hydroelectric, as can be seen in the Tab. 1 that exhibition some data about them.

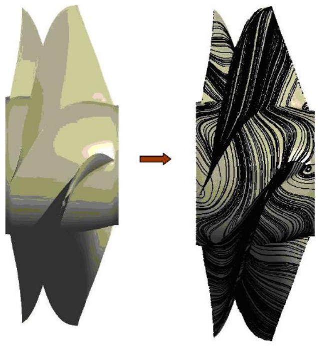
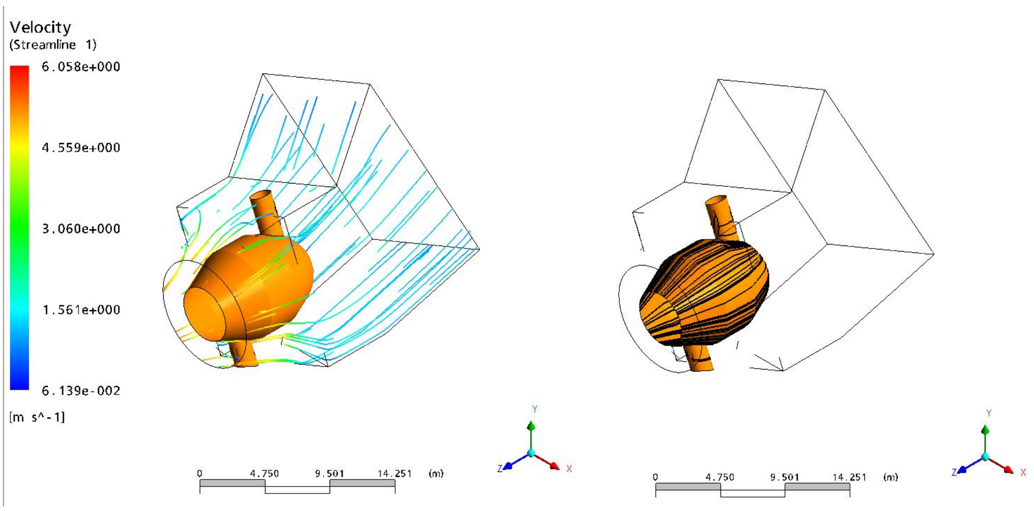


Figure 3. Geometry Detailed and Runner Shear Lines

From the data presents in Tab. 1 e of the required power value for the machine, this concludes the external diameter runner must be among the values $5,5\ m$ and $6,5\ m$. Thus, used, for others simulations, an external diameter of $5,6\ m$, which approached of the required power by the preliminary project, $25,9\ MW$. With the new simulation obtained the next results.

In Fig. 4 can be observed, through streamlines of the flow and the shear lines in bulb, that the flow is parallel to the axis of the bulb. And in Fig. 5 can visualize a pressure field presents in the interface, common domain to the bulb and to the distributor, which introduces a minimum variation. The Fig. 6 allows observing increase in velocity along the water inlet, which varies of near to zero, in inlet, to the about $6\ m/s$ in common interface with distributor.



(a) (b)

Figure 4. Streamlines and Shear Lines in Bulb

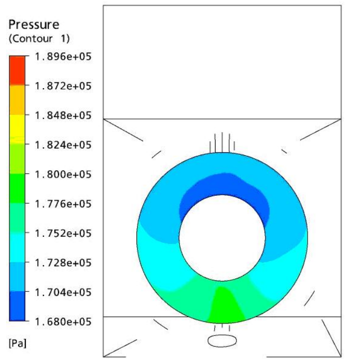
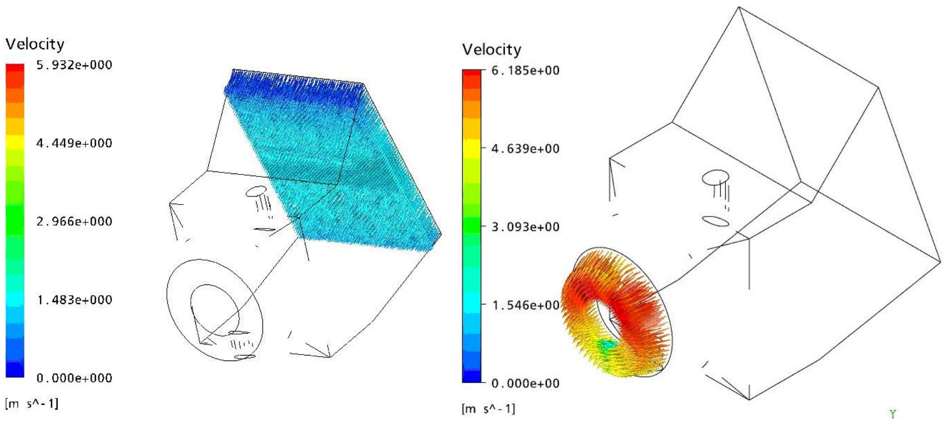


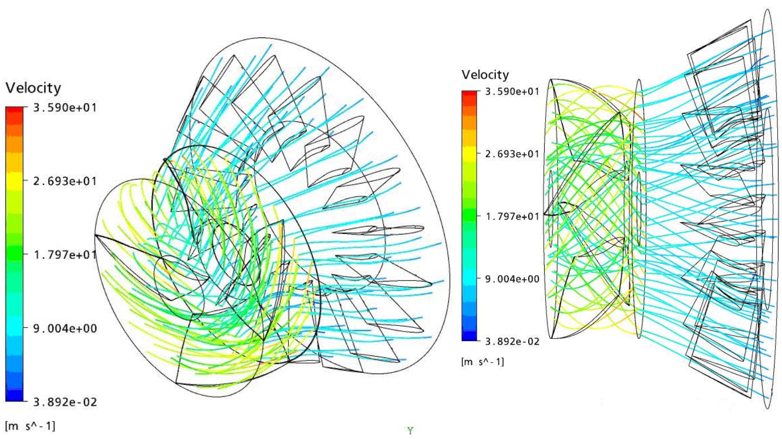
Figure 5. Pressure Field in Interface



(a) (b)

Figure 6. Velocity Vectors in Water Inlet

According Fig. 7 it's possible to visualize the streamlines, which ones follow the expected flow, because realizes that the flow has a increase in its direction in the distributor, crossing by runner with the rotation proposed by the distributor and in runner the fluid continues increasing its rotation, as expected. And in Fig. 8 it can observed that the shear lines presents in runner are attached in its blade surfaces, what it indicates the flow doesn't introduces recirculation in this. Ruprecht, in several papers, said that, in numeric simulation, when it's possible visualize a flow where the shear lines presents in runner are attached in its blade surfaces, it can say the runner geometry is adequated.



(a) (b)

Figure 7. Streamlines in Distributor and Runner

Figure 8. Shear Lines in Runner

Visualizing the Fig. 9, it's observed the pressure field introduces a minimum variation along the distributor and runner. It can verify yet, the pressure value has an increase in direction of the external diameter, as waited because the flow has larger speed how much nearest of the external diameter, so much in the distributor as in the runner. According Fig. 10, in the pressure field in blades of runner, in which it introduces a minimum variation and its larger value it observed in its leading edge.

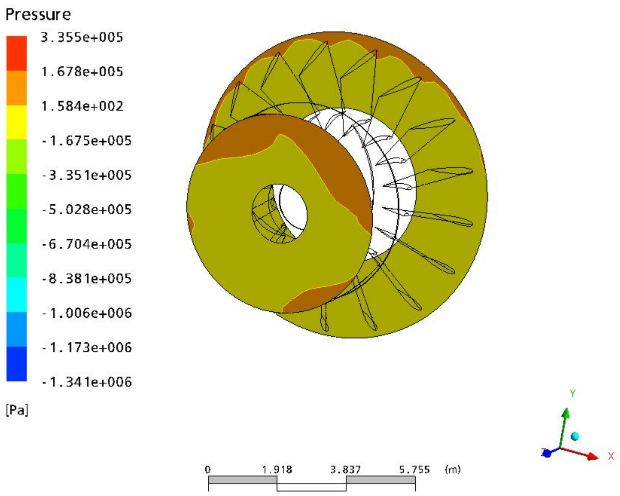


Figure 9. Pressure Field in Distributor and Runner

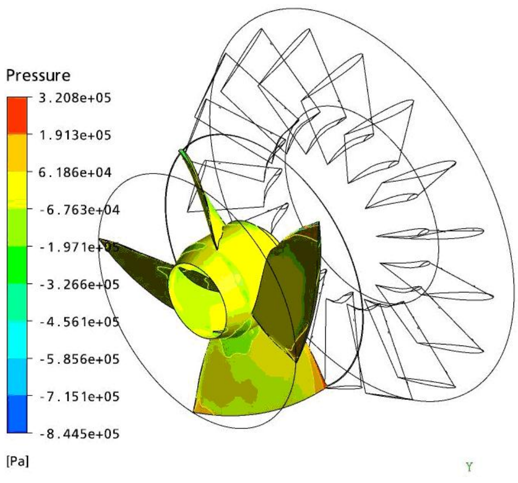


Figure 10. Pressure Field in Runner Blades

On the Fig. 11, it can observe the velocity has an increase along the distributor and runner. It's possible to observe the increase in distant regions of the wall, this occurs because in the wall has the boundary layer. That was expected, because the main function is to generate power, which is obtained from the speed increase. And, visualizing the velocity vectors presents in those figures, it's possible to observe that the velocity has an increase the its rotation when crossing by the distributor, suffering a larger increase yet, when crossing by the runner.

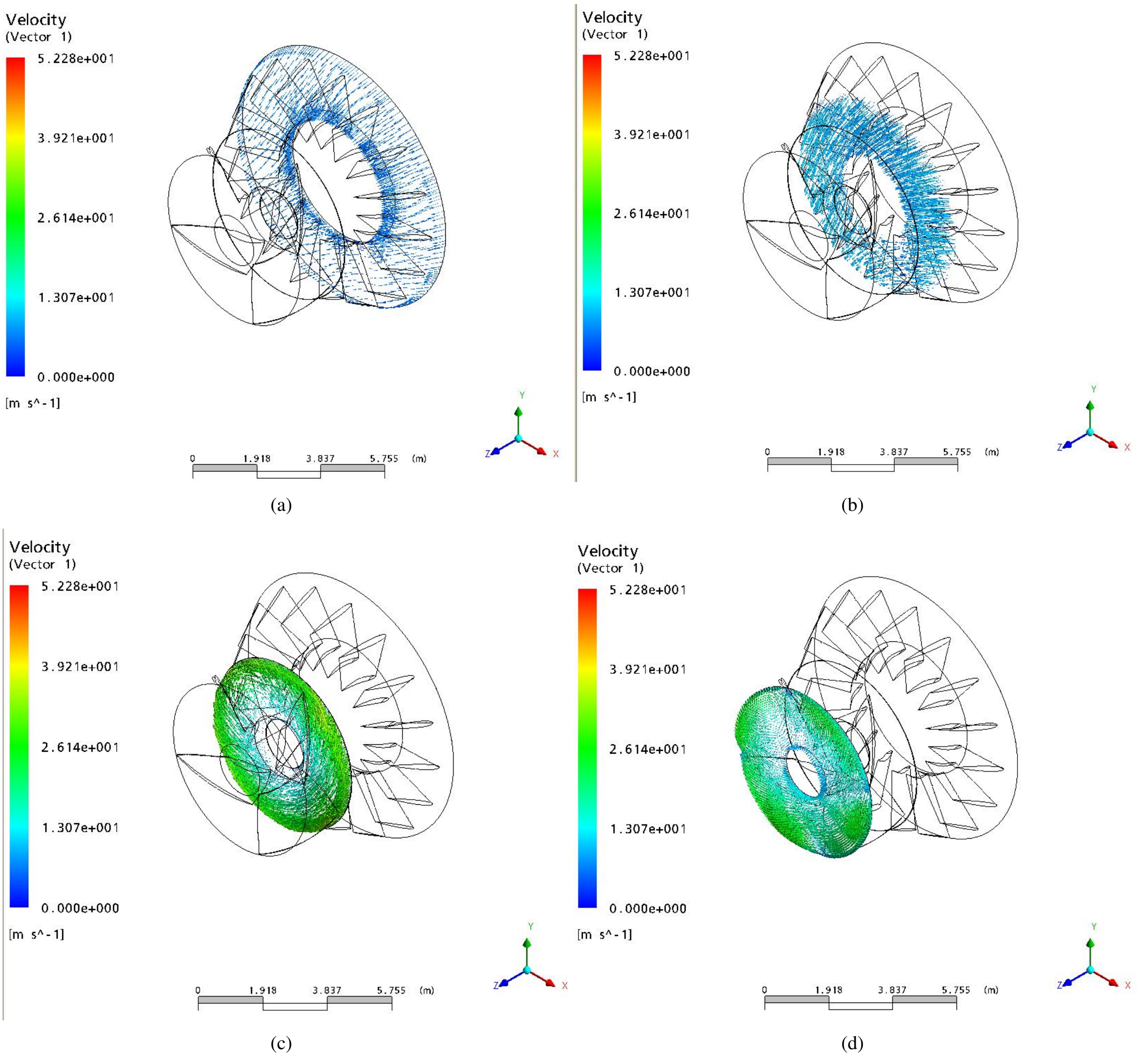


Figure 11. Velocity Vectors in Distributor and Runner

The efficiency of the partitioned simulation, calculated by data output, has a result of 61%. This efficiency was obtained from the power and mass flow, the which ones has as value $17,1\ MW$ and $220,26\ m^3/s$. The power is calculated with the torque and rotation velocity. The torque is obtained with the simulation, $1,6 \times 10^6\ N.m$, and the rotation velocity is defined, $10,47\ rad/s$.

To the assembled simulation, one can note that the streamlines acquire, after the rotor, longitudinal vorticity. This is a expected behavior due to the machine rotation Fig. 12. The helicoidal flow extends downstream to the draft tube, avoiding the appearance of stagnation points. The rotational pattern of the flow is reinforced by Fig. 14. The shear lines on the rotor Fig. 13 shows attached behavior from upstream to downstream. This is a indication of attached boundary layer and the non-existence of any viscous drag that affects the hydrodynamics of the machine, and for consequence, its efficiency.

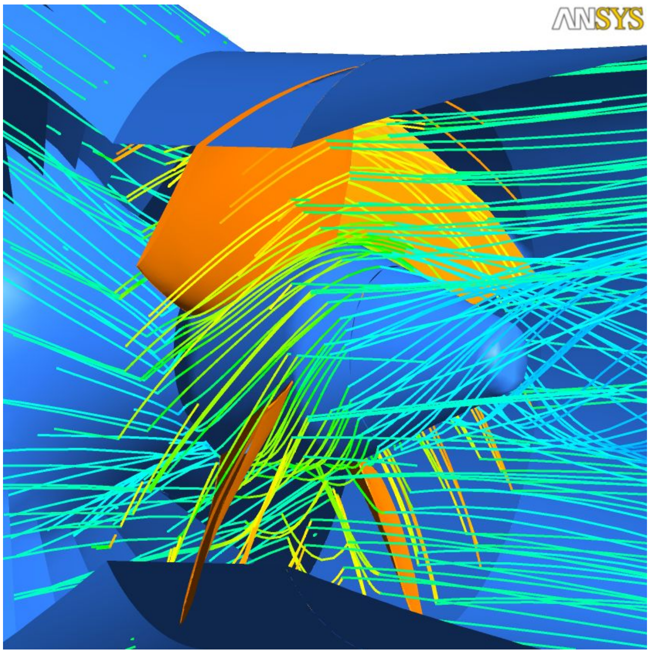


Figure 12. Streamlines Lines in Turbine

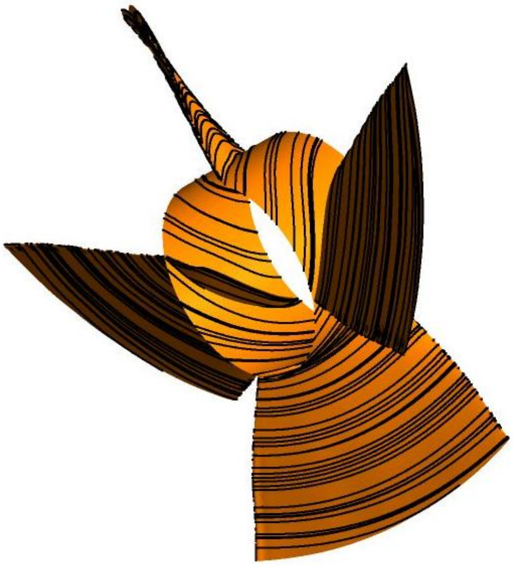


Figure 13. Shear Lines in Runner

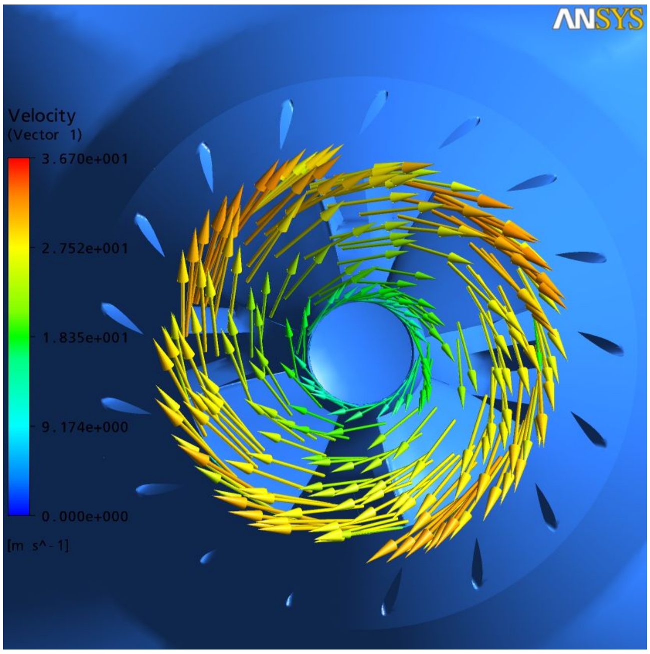


Figure 14. Velocity Vectors in Runner

The efficiency of the assembled simulation, calculated by data output, has as result 61%. This efficiency was obtained from the power and mass flow, the which ones has as value $25,4\ MW$ and $326,09\ m^3/s$. The power is calculated with the torque and rotation velocity. The torque is obtained with the simulation, $2,4 \times 10^6\ N.m$, and the rotation velocity is defined, $10,47\ rad/s$.

4. CONCLUSION

The comparation about two methodologies, parts and assembled simulation, it showed valid, since in the parts simulation, intended to analyze the preliminary project of the CHE-BM. Through this analysis, can observe the flow rate proposed in the preliminary project is low for the required power. From this point, adopts to other analysis, simulates the machine using only the pressure variation. In this case the mass flow isn't imposed and becomes the simulation result. Thus found required power, however, the calculated flow rate is greater then the flow rate estimated as optimum and the efficiency is lower of the expected. This efficiency constant by both cases, can be explained through the curve of the flow rate in function of the efficiency, Fig. 15, because it has the same parabolic characteristics. On the first case, a machine with size below expected was simulated, and on the second case, a machine with size above expected was simulated, alongside with its flow rate. Therefore, the results are placed at the two ends of the parabole. The next stage of this study is a middle analysis of the results, aiming to find better working conditions of the future installations of Belo Monte plant complex.

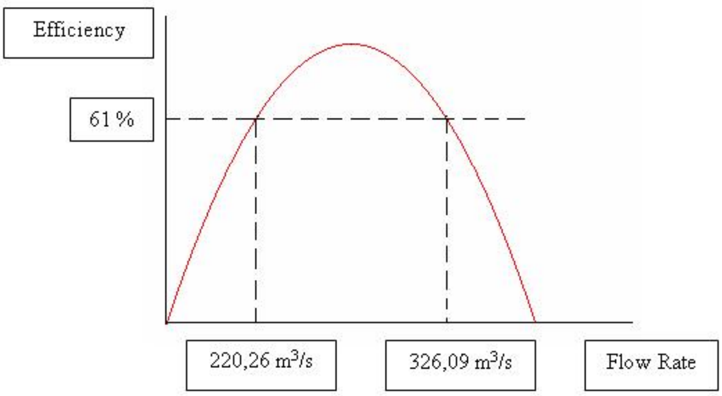


Figure 15. Flow Rate in Function of Efficiency

5. ACKNOWLEDGEMENTS

This work was supported by P&D program of ELETRONORTE S/A.

6. REFERENCES

Avelan, F., Maur, S., 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, 7p.

BRASIL, 2005, "Balanço Energético Nacional", Ministério das Minas e Energia, http://www.mme.gov.br/frontSide/site/view.do?viewPublicationId=10780&viewPublicationTypeId=9&queryUrl= http%3A%2F%2Fwww.mme.gov.br%2Fsite%2Fsearch.do%3FpreviousQuery%3Dbalan%25E7o%2Benergetico %2B2005%26pageNum%3D3

Berstron, J., 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

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 Hydaruliques, Presses Polytechniques et Universitaires Romandes", França, 407p.

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

Kovalev, N.N., 1965, "Hydroturbines Ű Design and Construction, Israel Program for Scientific Traslation", Jerusalém, 680p.

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

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., 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., Guibault F. 2001, "Automatic Shape Optimization of a Hydraulic Turbine Draft Tube", Departament of Mechanical Engineering, 6 p.

Tamm, A., Gugau, M., 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.

Zulcy, S., Bran, R.,1987, "Máquinas de Fluxo: Turbinas, Bombas, Ventiladores", Ao Livro Técnico, Rio de Janeiro, 262p.

Massé, B., Page, M., Magnan, R., Giroux, A.M., 1999, "Numerical Simulations: A Tool to Improve Performance of Hydraulic Turbines, WaterPower 99, Hydro Future: Tecnology, Markets and Policy", Las Vegas, Nevada, USA.

Tamm, A., 2002, "Experimental and 3d numerical analysis of the flow field in turbomachines part i", Int. Cong. On Quality Assessment of Numerical Simulations in Engineering

Moura, M. D., 2003, Simulação numérica de escoamentos turbulentos em tubos de sucção de turbinas hidráulicas, Master's theisis, UnB - ENM.

Mauri, S., Kueny, J. L., Avellan, F., Numerical prediction of the flow in a turbine draft tube influence of the boundary conditions.

7. Responsibility notice

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