AXIAL TURBINE FOR DOUBLE EFFECT TIDAL POWER PLANTS: A CFD ANALYSIS.

ANTONIO CARLOS BARKETT BOTAN[1,a], GERALDO LUCIO TIAGO FILHO [2,b], HELCIO FRANCISCO VILLA NOVA[3,c], THIAGO SOARES CORREA[4,d], OSVALDO R. SAAVEDRA[5,e]

[1, 2, 3, 4] Universidade Federal de Itajubá - Av. BPS, 1303 - ZIPCODE 37500-090 - Itajubá/MG, Brazil

[5]Universidade Federal do Maranhão - Av. dos Portugueses, S/N - ZIPCODE 65040-080 – São Luis/MA, Brazil

[a]acbotan1981@gmail.com, [b]tiago@unifei.edu.br, [c]helcio.villanova@gmail.com, [d]thiagoscorrea@yahoo.com.br, [e]o.saavedra@ieee.org

ABSTRACT

The paper shows a study on a turbine based on Wells turbine for OWC, to be used in tidal power plants. Its rectifying effect generated by its symmetric blades allows the same way of rotation movement whatever is the direction of the flow. With this concept applied in double effect tidal power plants, the harnessing of potential energy of the tides can be made during the passage of the water from ocean to reservoir and from the reservoir to the ocean. A first model of the rotor for this turbine was designed and tested in CFD by ANSYS CFX® in different conditions, varying the mass flow. The aim of this work is to achieve the best efficiency for the proposed model of rotor, and to propose future studies of optimization for the model.

Key-words: axial turbine, Wells turbine, tidal power, CFD analysis.

INTRODUCTION

Many devices have been developed for the harnessing of tidal power. Since tidal power plants using the potential energy of the tides in dams, like La Rance power plant (France), or hydrokinetic new model of turbines.

Tidal power plants that make use of dams operate in single effect or double effect. Plants which operate in single effect generate only during the passage of the water from the reservoir to the ocean, and plants which operate in double effect generate during both passage from reservoir to ocean and from ocean to the reservoir (JOG, 1989). The turbines used in this type of plant are generally type propeller turbines (Kaplan, Bulb and Straflo turbines) (MACINTYRE, 1983).

When the turbine operates in double effect, its design is more complex, due it has to attend in satisfactory conditions and many working regimes which it is submitted. The consequence is its efficiency is lower than if it is used as single effect (MACINTYRE, 1983). And due the diary operating time is lower than of turbines used in single effect, which require a bigger number of devices (MACINTYRE, 1983).

The type Wells turbine is an axial turbine with symmetric blades used in oscillating water column plants. Its symmetric blades allow the movement of rotation regardless the direction of the flow. Based on this conception, a model of axial turbine with symmetric blades was developed, to be used in double effect tidal power plants.

The model was developed to be tested in a test approach at LHPCH (Small Hydro Hydromechanic Laboratory) in Federal University of Itajubá – Brazil. A numerical model with the same dimensions was designed and tested on a CFD platform, by ANSYS CFX®. Tests were done with constant angular velocity (rpm) and varying the mass flow rate.

This paper describes the numerical CFD study of the proposed turbine rotor, presenting the results of the tests, and analyzing the behavior of the fluid and the efficiencies of the rotor. Since it is the first model of axial turbine with symmetric blades applied in tidal power conditions (low and very low head), suggestions to optimize the efficiency on the rotor on future studies will be presented at the end of this paper.

1 ROTOR DESCRIPTION

The rotor was based on Wells turbine rotor and other studies on this type of rotor. The Wells turbines are axial turbines consisted of a set of radially fixed blade with constant sections constructed by a *symmetric hydrodynamic* profile type NACA 00xx positioned at 90° to the rotor axis, in the rotation plane normal to the incident flow (RAGHUNATHAN, 1995). The tangential force of the rotor works only in one direction althoughairflow is oscillating, consequently, the turbine rotates in the same directionwithout rectifying valves to rectify the oscillating flow and produces power regardless ofwhich way the air is flowing (DHANASEKARAN & GOVARDHAN, 2005).

Although Wells turbine was developed to be used in airflow, its symmetric shape allows to be used in water flow. In (SOONS, 2006), the author assumes that the main difference is the density of the water which is approximately 800 timeslarger. Still in his paper, he notes that the pressure distribution over the blades must avoid the cavitation region.

For the model of rotor developed for this work some characteristics were considered. According to (RODRIGUEZ, 2009), many studies have already done on the properties of the Wells turbine about their low efficiency, low starting characteristics and high operating noise. Some works suggest to add guide vanes (DARABI AND PORIAVALI, 2007) and /or to vary the attack angle of the blades (variable-pitch turbines) (GARCIA, 2008).

Some parameters as presented in (SOONS, 2006) which were based in (RAGHUNATHAN, 1995) and "T. Basse's Wave Turbine" from T. Basse (1991), were considered for the designing of the current rotor, such as rotor solidity, hub-to-tipratio and number of the blades.

The rotor for this work was designed with blades NACA 0015, with attack angle, α, of 13 degrees combined with the angle of the guide vanes of the stator, on the inlet and outlet of the flow respectively. The rotor was developed by these conditions to respect the theoretic velocities triangles on the inlet and the outlet of the flow.Besides the current rotor was designed considering the use with guide vanes, this paper will describe the tests only on the rotor.

All the geometric details of the studied model of turbine are shown in Table 1.

Table1: Geometric details of the studied model of turbine.

Tip Diameter, D_t	0.112 m
Hub Diameter, D_h	0.188 m
Hub/Tip ratio h	0.60
Number of blades z	6
Solidity σ	0.60
Attack angle α	13 degrees
Profile	NACA 0015

The dimensions of the model of turbine were designed according the dimension of the tubes of the hydro system of the Hydro-Mechanical Laboratory for Small Hydro at the Federal University of Itajuba, Unifei. System is consisted of pipes with diameter 8" Schedule 40. A physical model was built in ABS plastic by 3-D printer to be tested in a test stand.

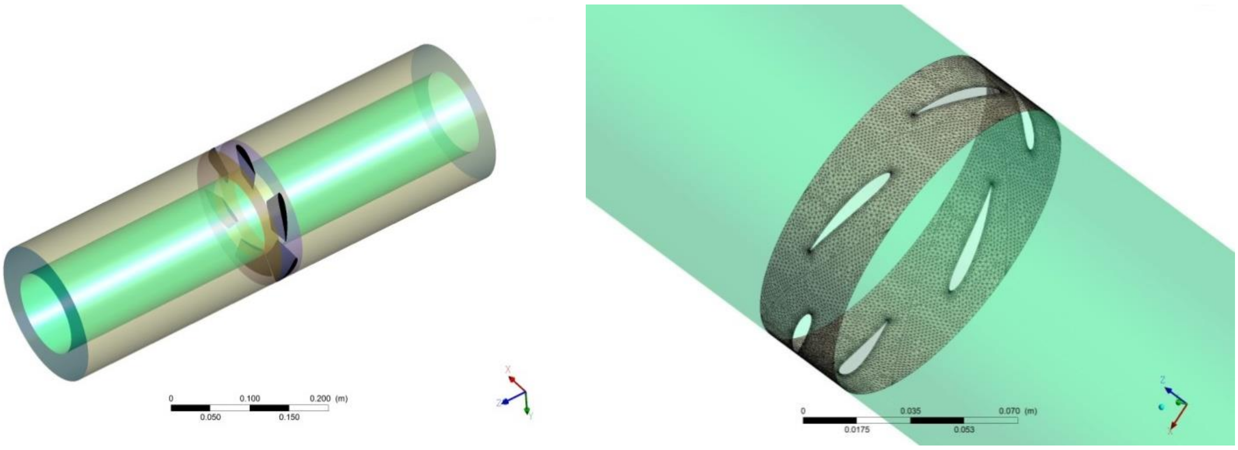
2 COMPUTATIONAL FLUID DYNAMICS TESTS

2.1 COMPUTATIONAL GRID

The objective of this step is to analyze the behavior of the water flow on the rotor blades. The aim is to determine the torque according to the boundary conditions, and consequently to obtain the shaft power and efficiency of the machine.

Three domains were created separately: Inlet, Rotor and Outlet. The assembled set composed by 6 blades was divided in one section to make the set less heavy in elements. Inlet and Outlet domains meshes were created with unstructured tetrahedral mesh. Unstructured meshes have the great advantage of being flexible to adapt to the limits of the domain, allowing an almost automatic construction of the representative mesh of the geometry. It is only needed to specify the number of nodes to the contours and a calculus algorithm (SOUZA, 2011).

Rotor domain is composed of structured tetrahedral mesh. Figure 1 shows the rotor domain meshes, and Table 2 presents the number of elements in each domain.



(a) (b)

Figure 1 – (a) Assembled set mesh, (b) rotor domain mesh.

Table 2. – Number of elements in each domain.

Domain	Elements
Inlet	168552
Rotor	588346
Outlet	168552

2.2 COMPUTATIONAL FLUID DYNAMICS TESTS

Tests were done considering constant angular velocity (rpm), varying the mass flow rate. Tested angular velocities were: 300 rpm, 400 rpm, 500 rpm, 600 rpm, 700 rpm and 800 rpm.

Being a turbulent model, the turbulence model chosen was k-epsilon model. K-epsilon turbulence model is the most popular and the most used in engineering problems solution. Is a robust, economic and produces a high variety of turbulent flows, which explains its popularity in simulations of industrial flows and heat transfer (SOUZA, 2011). K-epsilon model is a semi-empiric model based in transport model equations for turbulent kinetic energy (k) and its dissipation rate (epsilon).

Inflow and outflow boundary conditions chosen were Mass Flow Inlet and P-Static Outlet. Due the domains were divided in six, sectioned per blade, parameter Mass Flow Inlet per passage was chosen, dividing the total mass flow value in six. P-Static Outlet had the value of 0 Pa.

In convergence control, the number of run iterations was 100 iterations. Higher values tested of iterations for the run cases, convergence lines started to run in reverse flow cycle. Besides not achieve the convergence, the obtained values of inlet and outlet mass flow rate respected the convergence criteria of residual target of 1E-4.

Table 3 presents the parameters used for CFX modeling.

Table 3: Parameters used in CFX modeling.

Fluid Models	Turbulence	k-epsilon
Convergence Control	Max iterations	100
	Timescale control	Auto timescale
	Timescale factor	1.0
Convergence Criteria	Residual type	RMS
	Residual target	1.E-4

3 RESULTS

All the cases were run separately, varying the mass flow rate. The results reached values of flow between Q = 0.01148 and Q = 0.04400 and efficiencies in order of 66% for Q = 0.04200 at 800 rpm. CFX results do not consider losses per friction and losses per mechanical components used in the test stand. Previous tests in test stand were done to obtain the mechanical losses of friction. The results are shown in Figure 2. The results of total rotor efficiency were calculated subtracting the friction power from the total power calculated by CFX, and are shown in Figure 3.

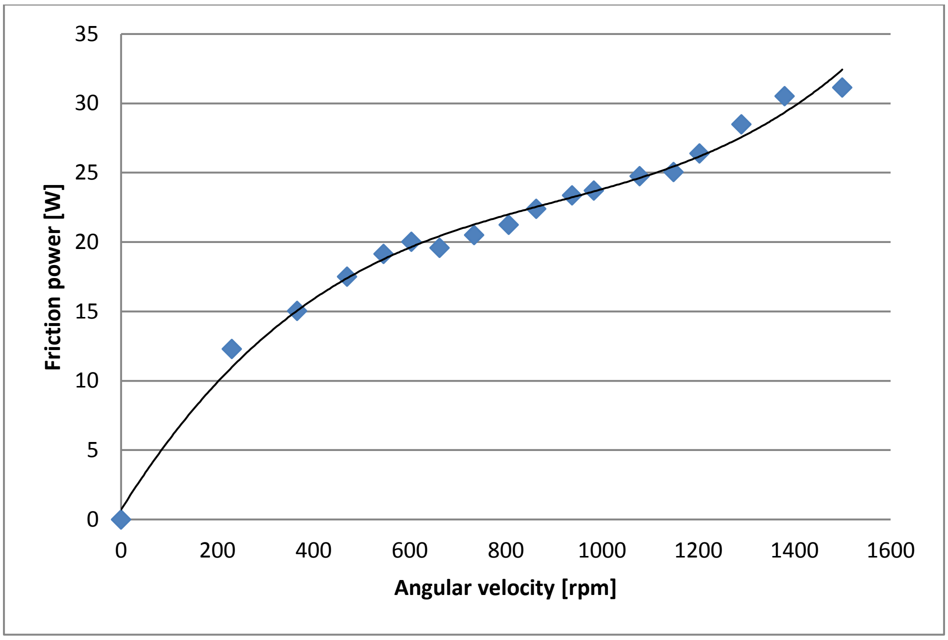


Figure 3 – Friction power.

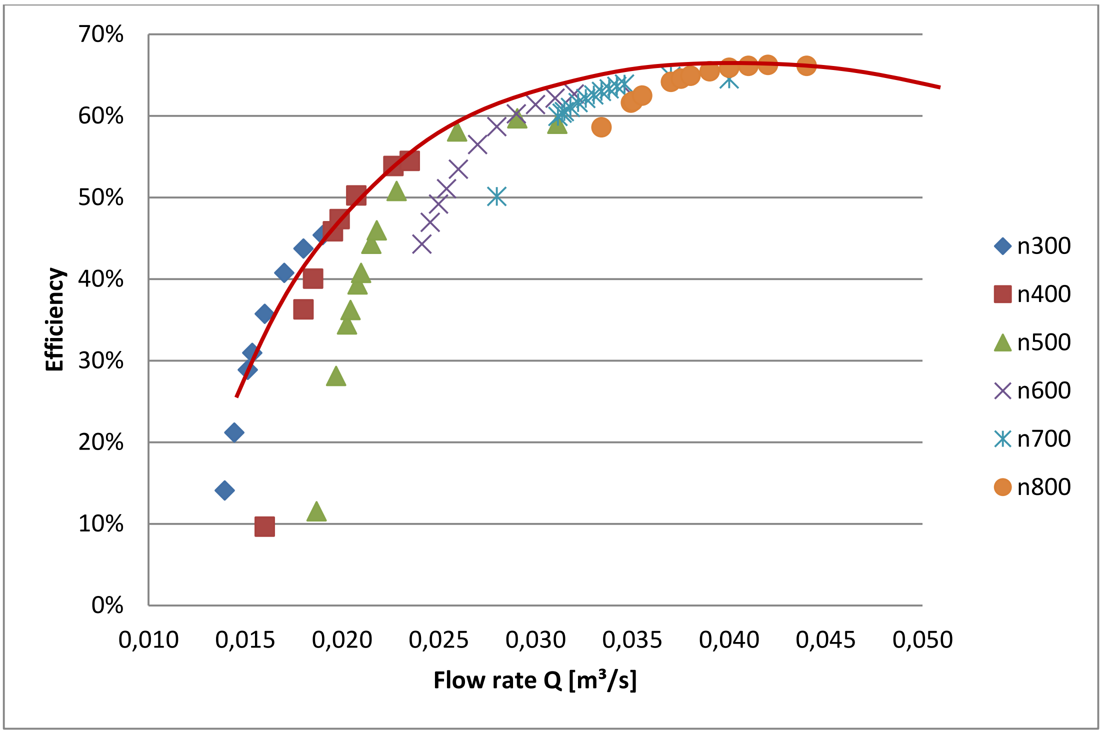


Figure 3 – Total Efficiency of the model of rotor.

In Figure 3 can be observed the tendency that how the angular velocity is raised, the respective curves get closer one of the other, tending to an efficiency limit. The difference between the higher efficiency of n = 700 and n = 800 rpm is 1.4%, and between n = 600 rpm and n = 700 rpm is 2.2%, resulting in a proportional increase according the angular velocity is raised, and limiting the higher rotation speed for the model.

Figures 4 and 5 present the differences of pressure in all domains. On upstream side pressure reaches value in order of 16653 Pa, with lower value at downstream. On the edge of the blade the value reaches 2.112 Pa, and -2.00×10^{4}Pa on the lower pressure side of the blade. The pressure is higher at the leading edge, but it reaches values lower than zero on the exhaustion surface. At the lower pressure surface of the blade, cavitation happens due the low value of pressure, which generates losses of efficiency on the rotor.

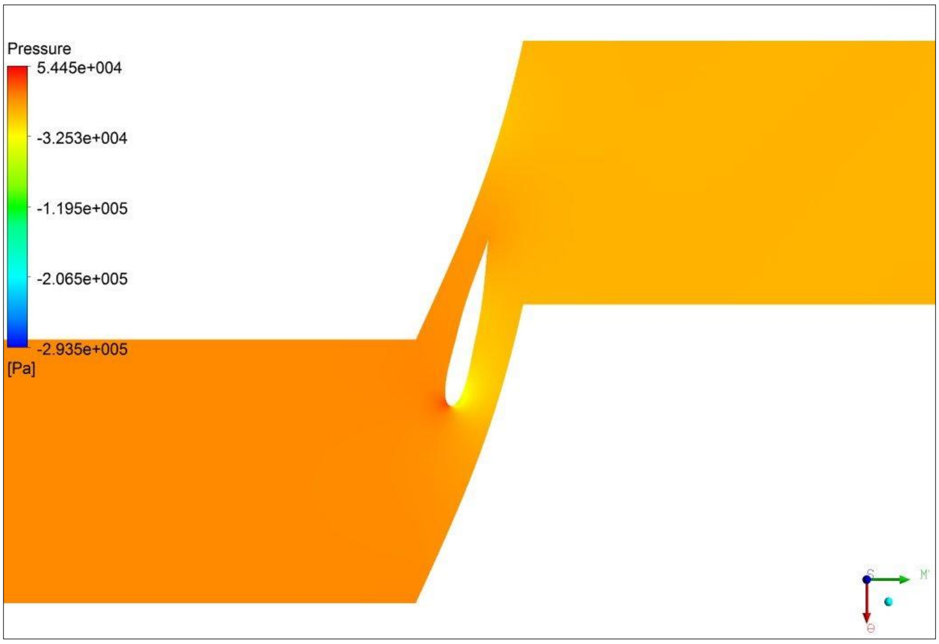


Figure 4 – Pressure on inlet, rotor and outlet domains, at 50% span.

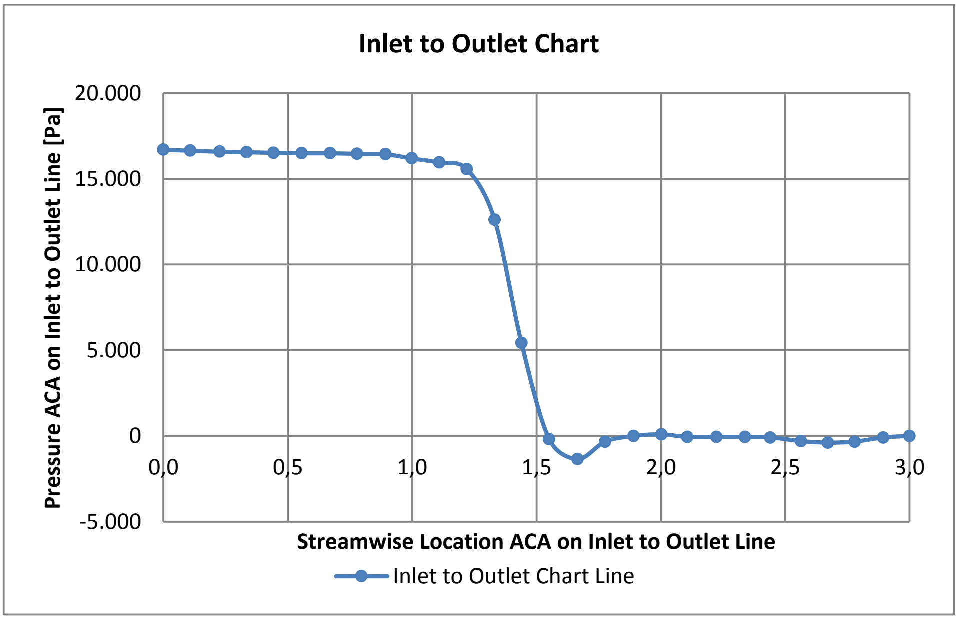


Figure 5 - Pressure on inlet, rotor and outlet domains.

Figure 6 represents the velocity contours at 50% span. Can be noticed the raise of velocity after the rotor, with peaks of velocity vectors on the lower pressure surface, and the streamline with swirl form across the outlet tube, until it reaches the initial velocity, as shown in Figure 7. Figure 8 shows the velocity vectors on the blades.

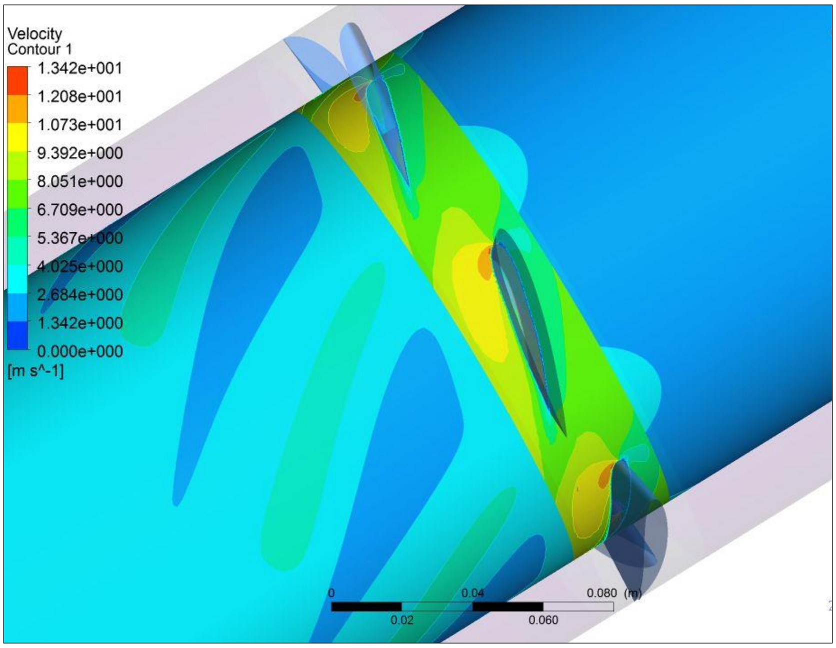


Figure 6 – Velocity contour at 50% span.

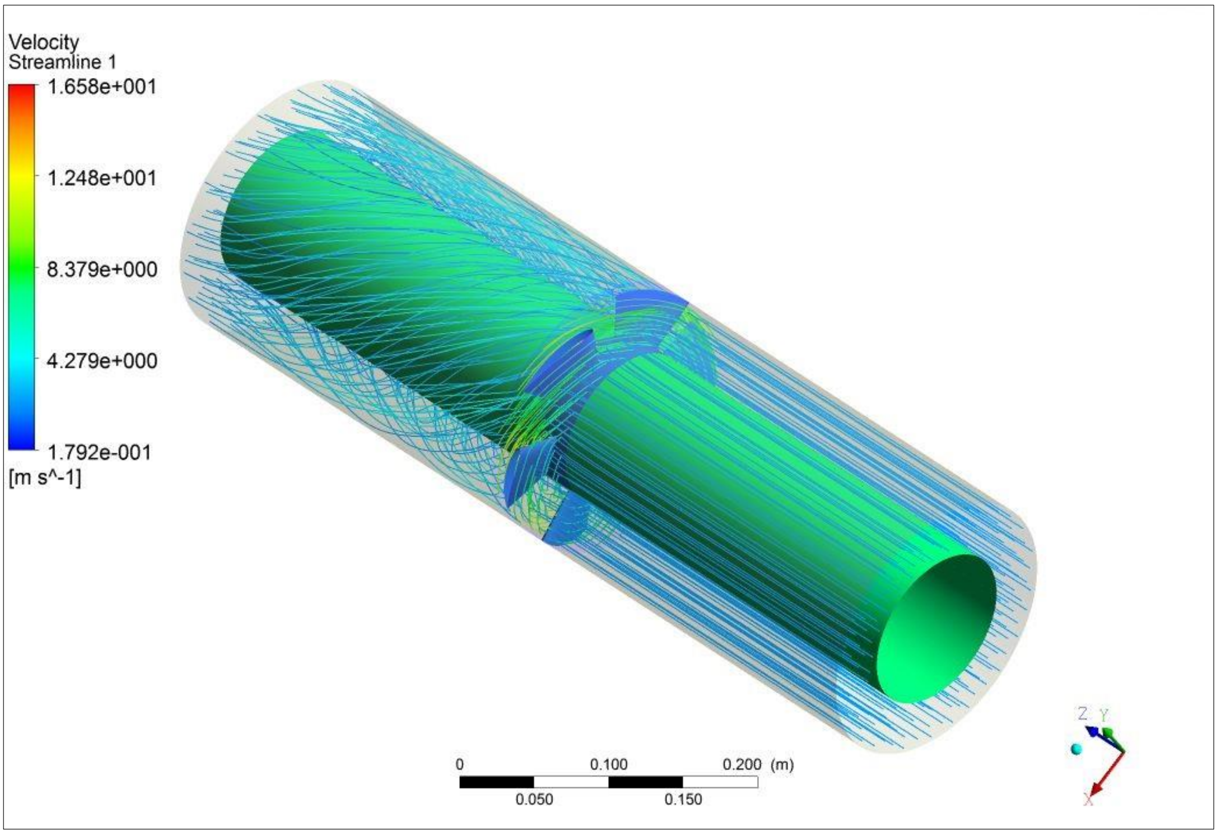


Figure 7 – Velocity streamlines.

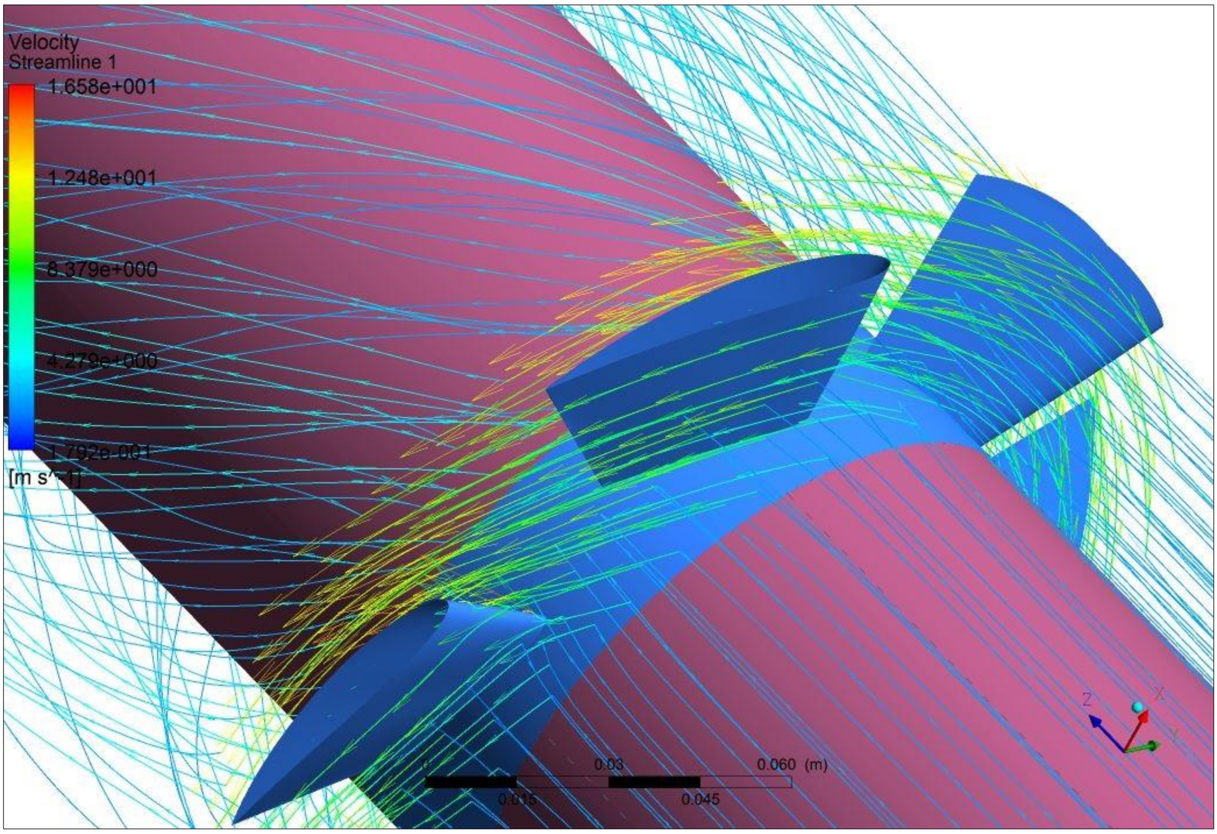


Figure 8 – Velocity vectors on the blades.

4 CONCLUSION AND SUGGESTIONS

The proposed rotor indicated to tidal power plants double effect operation due its symmetric blades, reached efficiency in order of 66% for tested angular velocities, and if tested with higher values of rotation, will not reach values much greater than tested. In other hands, tests were done extrapolating the intended values of flow rate, to reach the peak curve and then its descent. It has resulted in high values of Head, not considerable to cases of tidal turbines. Commercial turbines reach efficiencies in order of 95%, so, the proposed rotor must to be optimized to get a place on the market.

Losses due cavitation, because the angular velocity and probably due the attack angle must be analyzed. According to studies which suggest the use of guide vanes (GARCIA, 2008), future tests should analyze the behavior of the flow on the blade using these devices.

A test using CFX was done only to verify the efficiency in blades assembled 90 degrees to shaft axis, which proved the literature about lower efficiency (DARABI and PORIAVALI, 2007), besides it was not reported in this paper. It is suggested for a future test to identify an appropriate attack angle to optimize the efficiency.

REFERENCES

1. Dhanasekaran, T.S., Govardhan, M., 2005. Computational analysis of performance and flow investigation on wells turbine for wave energy conversion. *Renewable Energy*, vol. 30, pp. 2129–2147.
2. García, B. P. Estudio de una turbina de impulso radial para el aprovechamiento de la energía del oleaje, 2008. PhD thesis, *Universidad de Valladolid*.
3. Jog, M. G. Hydro-electric and pumped storage plants, 1989. *John Willey & Sons*, pp. 133-185.
4. Macintyre, A. J. Maquinas Motrizes Hidráulicas, 1983. *Guanabara Dois*, chapter 13, pp. 269-297.
5. Mohamed, M. H., Shaaban, S. Optimization of blade pitch angle of an axial turbine used for waveenergy conversion, 2013. *Energy*, vol. 56 pp. 229-239.
6. RAGHUNATHAN, S. The Wells Air Turbine for Wave Energy Conversion, 1995. *Prog. Aerospace Sci*., Vol. 31.
7. Rodriguez, C. M. F. Projecto das Pás Directizes Fixas de Uma Turbina Auto-Rectificadora de Acção para Aproveitamento da Energia das Ondas, 2009. Master thesis – *Instituto Superior Técnico, Universidade Técnica de Lisboa.*
8. Setoguchi, T., Santhakumar, S., Takao, M., Kim, T. H., Kaneko, K. A Modified Wells Turbine for Wave Energy Conversion, 2003. *Renewable Energy*, vol. 28, pp. 79-91.
9. Soons, E. J. Analysis of the Wells rotor in oscillating air & water flows, 2006. *Co-Ordinated Action On Ocean Energies Exchange Programme*.
10. Souza, Z. Projeto de máquinas de fluxo: tomo I, base teórica e experimental, 2011. 1 Ed. *Editora* Interciência Ltda. Co-editora *Acta Ltda*.
11. Tease, W. K. Dynamic response of a variable pitch Wells turbine, 2003. *Fifth European Wave Energy Conference Proceedings*. University College Cork, Ireland.