PARAMETERIZATION OF A HEXAEDRIC MESH GENERATION PROCESS FOR THE APPLICATION IN THE FLOW SOLUTION OVER A SMALL SCALE WIND TURBINE.

Carneiro, Francisco Olimpio Moura, olimpiomcarneiro@gmail.com
Rocha, Paulo Alexandre Costa, paulo@posmec.ufc.br
Modolo, Angelo Bezerra, modolo.angelo@gmail.com
Silva, Maria Eugênia Vieira da, eugenia@posmec.ufc.br
Universidade Federal do Ceará – Campus do Pici, S/N – Bloco 715 – Pici – Fortaleza – Ceará

Abstract. *The present work has the main objective of defining parameters to the generation of a group of hexahedric meshes in order to simulate the flow over a small scale wind turbine (diameter of 3 m). In such way, the following stages were accomplished: Primarily the size of the control volume upstream and downstream of the turbine was defined. Afterwards, the mesh in areas of large velocity and pressure gradients was progressively refined. Finally, layers near the surface were added in order to improve the quality of the boundary layer modeling. The first two steps aimed to obtain the best relationship between the quality of the results and the computational effort in the mesh generation likewise numerical solution processes as well. In this investigation task, the control volume size was varied from larger to smaller ones, while the behavior of viscous and pressure moments acquired by the numerical solution were observed. For the analysis of the boundary layer and the mesh refinement close to the surface, the y^+ value (dimensionless distance of the wall) was adopted as a reference parameter. For the turbulence model adopted, it was intended to achieve values smaller than 50 for y^+, what guarantees a good boundary layer modeling according to the logarithmic law of the wall. The software used for this work was OpenFOAM, which is a free package for CFD solutions. To execute this work, applications and solvers were used for the mesh generation and the obtaining process of the flow´s numerical solution, which applied the classic k-ε turbulence model, together with a second order discretization scheme.*
The present work is supported by the FUNCAP/CNPq/PPP.

Keywords*: mesh parameterization; small scale wind turbine; OpenFOAM; k- ε turbulence model.*

1. INTRODUCTION

1.1 The challenge of renewable energy

The electrical energy is one of the most important creations of the humanity. Nowadays it became the essential part for the evolution the world. It is already known that all traditional processes to obtain electrical energy are based in fossil fuel, which emits huge amounts of pollutants per KWh. The point is why this kind of process still generates the most part of the energy in the world. Many years of study turn the traditional process of generation economically and technologically feasible, consequently dominating the market. But no matter how cheap it is, the fossil fuel one day will be over. At the same time the necessity for energy has been growing day-by-day (Evans *et al*, 2009).

The challenge of the future is to use the renewable sources of energy in a sustainable way. To achieve this, intensive research in renewable sources worldwide aimed to become a possible partial or total substitute of the traditional processes.

Nowadays, one of the most economically accessible technologies among the renewable technologies is the wind power. Many countries in the world adopted wind energy as a solution to support any traditional process to supply demand with clean energy. Big companies have been studying this technology for years and today they build the biggest turbines in the world. That is the most able technology to compete with the cost of traditional processes.

All the aerodynamics effects that happen over a wind turbine blade are very complex and still represent a challenge for worldwide researches. Despite the aerodynamics of wind turbine inherits the legacy of airplane and helicopter rotor studies, in this case the system extract energy of the wind. This difference creates specific problems for the design of the turbine blades (Schreck & Robinson, 2007).

The experimental tests support very well the study of the turbine blades, but they are expensive and the instrumentation is difficult. So, for a process of investigation to optimize any system of the wind turbine, it is necessary to repeat several tests. In this context, the Computational Fluid Dynamics (CFD) gains special attention with the evolution of computers, because it turns possible the solution of turbulence models in complex geometries, like a wind turbine blade, and problems with moving mesh, in a admissible time.

The size, type and refinement of a mesh are very important attributes to better use of a CFD model. In this context, the knowledge of how the mesh characteristics influence the accuracy of a solution for a particular geometry is the better way to obtain reliable results in a plausible time.

For problems of fluid flow it is common to use hexaedric meshes, whereas the equations of diffusion and convection are mainly orthogonal. The size is delimited by the influence of boundary conditions in the solution. For example, when analyzing airfoils it is necessary to know how the far conditions of inlet, outlet and the boundaries are to guarantee that the pressure, still in the downstream wake, don't suffer influence or cause discontinuity by the conditions. The refinement influence aims to adjust the y^+ value, which is necessary to be between 30 and 80 (Wilcox, 1994). This guarantees that the wall law is obeyed. To get the necessary adjust, the mesh is modified in stages until no sensible differences in the results occur between successive stages. Results are them called 'grid independent'. (Versteeg & Malalasekera, 2007)

When an analysis is made in wind turbines, it is necessary that the influence of the boundary conditions doesn't affect the variations on velocity and pressure when the flow passes through the plain of the blades (Fig. 1).

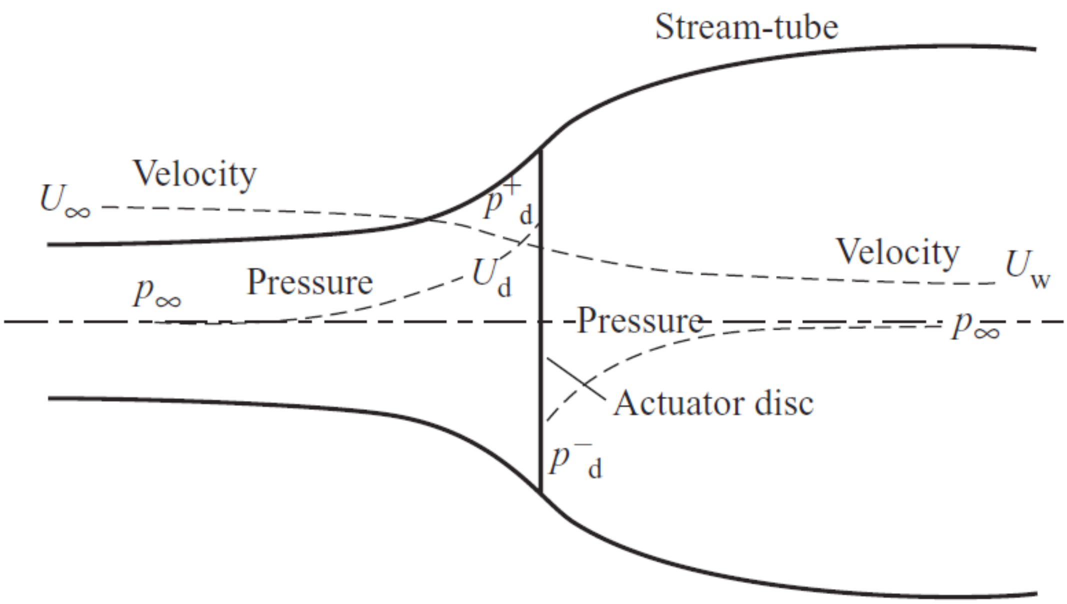

Figure 1 – Axial behavior of the flow characteristics over a wind turbine (Burton *et al*, 2001)

This work presents a grid independence study of a hexahedric mesh created to model the flow over a small wind turbine, in order to achieve the best results with the smallest computational effort. The flow was calculated by a steady, incompressible, turbulent solver, provided by the OpenFOAM (Tabor *et al*, 1998) CFD package.

2. METHODOLOGY

In the present work the blades of the small wind turbine were designed by the Blade Element Moment Theory (BEM) (Alvarez *et al*, 2008). The specific velocity, which is defined as the ratio between the blade tip speed and the wind speed was set to 5. The turbine is tree bladed and has a diameter of 3m. The airfoil profile chosen was the NACA4412, which has been used for years in wind energy assessment (Wright & Wood, 2004).

The parameterization of the mesh started from the investigation of the better total size of the mesh. The upstream and downstream regions of the wind turbine were analyzed. In this task, the first analysis was the downstream length when the upstream region remained constant. After that, it was possible to choose the better length for the downstream region and this value has become constant for the analysis of the upstream length.

With the appropriate length applied to the mesh, the next stage was to improve the accuracy of the simulation by changing the refinement levels adjustment and the investigation of the number of layers upon the surface. These stages have the same relevance that the length analysis because there must be possible adjustments the y^+ value.

The level of discretization of the refinement box was taken up to the maximum possible value could be achieved with the available computational effort. In this task, the mesh generation was performed in levels equal to 0 and 1, which was the furthest that could be reached with the available resources. The refinement level 1 was applied along all the total axial length, from upstream to downstream. In the radial plan of the blades, this refinement region covered an area about 45% bigger than the swept area.

The form of the control volume was a parallelepiped with a section in the radial plan of 10mx10m (Fig. 2).

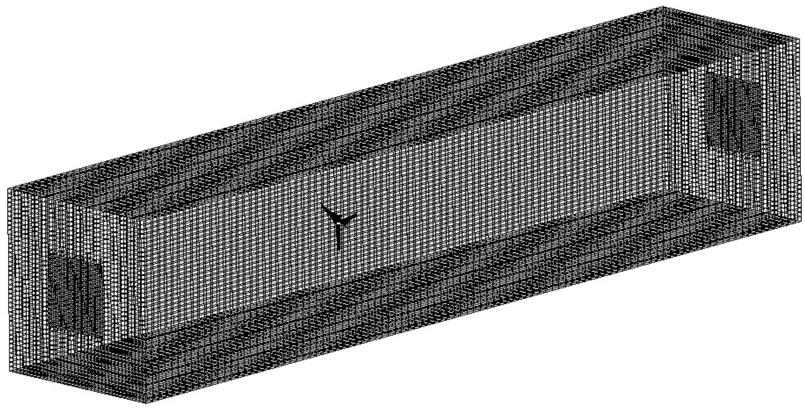

Figure 2 – The control volume geometry

2.1. Downstream analysis

In this first stage the meshes were generated with a constant upstream axial length equal to 10m, the maximum possible refinement box level and different downstream axial lengths which started with the same value of the upstream region until the maximum possible length needed to detect the simulation results behavior.

The initial value for the upstream region length was arbitrary, but started with the smaller one to reduce the time of simulations. The Table 1 shows the different sizes of the generated meshes.

Table 1 – Size data of the generated meshes applied in the downstream analysis stage

Generate Meshes		
Number	Length Upstream	Length Downstream
1	10m	10m
2	10m	20m
3	10m	30m
4	10m	40m
5	10m	50m
6	10m	60m
7	10m	70m
8	10m	80m
9	10m	90m

2.2. Upstream analysis

In this second stage, the choice of the axial length downstream was already done. So the length of the downstream region was kept constant and the refinement level also, while the upstream length varied of a initial value until the same of the upstream chosen (Table 2). The final length was the one that reached the tendency of the results.

Table 2 – Size data of the generated meshes applied in the upstream analysis stage

Generate Meshes		
Number	Length Upstream	Length Downstream
1	5m	30m
2	10m	30m
3	15m	30m
4	20m	30m
5	25m	30m
6	30m	30m

2.3. Number of layers analysis

In the last stage, the meshes were generated with the full length kept constant (both upstream and downstream), while it was varied the number of layers upon the surface (Table 3).

Table 3 – Size data of the generated meshes applied in the layers analysis stage

Generate Meshes		
Number	Full length	Number of Layers
1	50m	10
2	50m	20
3	50m	30
4	50m	40
5	50m	50
6	50m	60

2.4. Simulation parameters

In the present work the OpenFOAM CFD library were used to solve the flow characteristics. It is an open source CFD package, which includes several utilities and solvers to resolve flow problems. Specifically, the solver used was a steady, incompressible and isothermal code, with the possibility of different turbulence models.

For the parameterization of the mesh, it was indispensable to guarantee that the simulations were made with the same parameters. So the adjust of all cases had to consider the most instable one. Table 4 presents the boundary conditions applied to all the domains studied, while Table 5 shows the numerical parameters applied:

Table 4 – Applied boundary conditions

Region	Condition			
	Velocity (linear / angular)	Pressure	k	ε
Inlet	7.5 m/s	Zero gradient	0.844 m^2/s^2	10.617m^2/s^3
Outlet	Zero gradient	0 Pa	Zero gradient	Zero gradient
Contour	Zero gradient	Zero gradient	Zero gradient	Zero gradient
HAWT	-30.0 rad/s	Zero gradient	Zero gradient	Zero gradient

Table 5 – Applied numerical parameters

	Relaxation Factors	Discretization scheme
p	0.4	GammaV
k	0.4	Upwind
ε	0.4	Upwind
v	0.4	Upwind

In this work all the simulations were adjusted with the same physical and numerical parameters. To model the turbulent effects the traditional k-ε model was used. It is a 2-Equation model that uses the Boussinesq approximation to evaluate the effects of the Reynolds turbulent stresses. The k-ε model was already deeply studied and adjusted and, in spite of some known deficiencies, its results are satisfactory for engineering purposes. (Wilcox, 1994)

3. RESULTS AND DISCUSSIONS

The first analysis, which has evaluated the downstream length, demonstrates a cyclic result and the loop started at 30m of length (Fig. 3). This cyclic behavior can be explained by the moment of the wake that passes through the outlet boundary, i.e., as long as it is a steady state solution, the outlet boundary cuts different planes orthogonal to the free stream, perceiving different states of the vortex generated by the HAWT. This configuration indicates that 30m is a reasonable choice, once the results would still oscillate due to the continuous wake vortex shedding. Confirming that this tendency is limited to the downstream analysis, once in this region there is significant vorticity, Figure 4 presents a smooth convergent result, in both pressure and viscous momentum. This occurs because the inlet boundary only perceives the HAWT's presence when it is near enough, what clearly happens until a 15m distance from it. Finally, the most reasonable choice was 20m length, resulting a total length of 50m used in the domain axial distance.

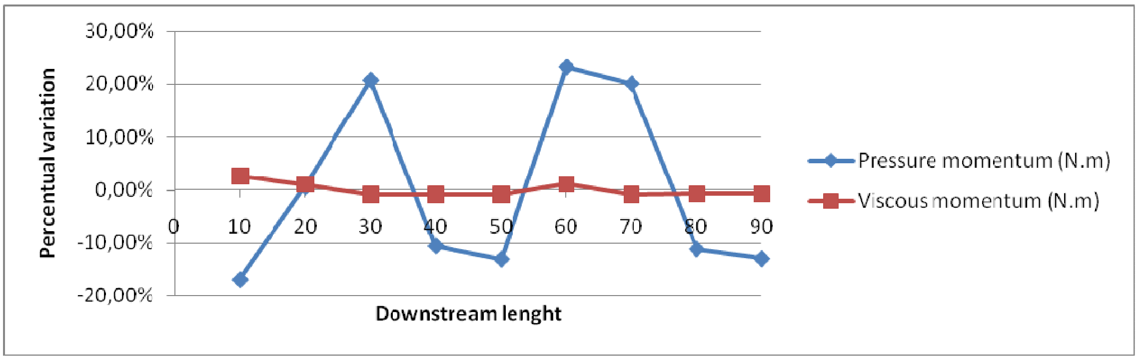

Figure 3 – Downstream variation of the momentum results

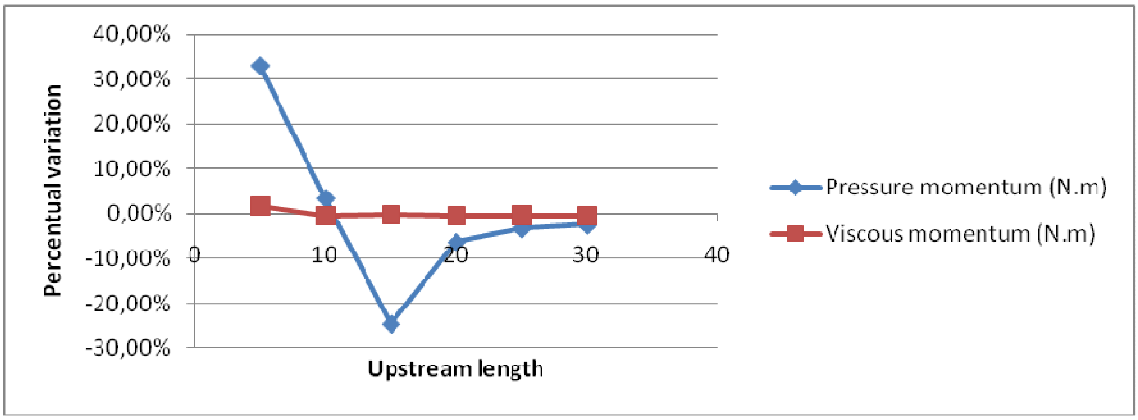

Figure 4 – Upstream variation of the momentum results

The accuracy of a simulation is closely linked to the refinement level of the mesh. With this in mind, a 'refinement box' was created with the maximum number of refinement cycles allowed by the computational effort available. In this work, a maximum refinement level of 1 was reached. Further studies must be carried on, once better resources are available.

As a final mesh improvement strategy, it was created over the blade surfaces several layers. This task aimed to accomplish the correct modeling of the boundary layer. In this work the k-ε turbulence model was used, so it was necessary to adjust the value of y^+, because this model do not resolve the flow inside the boundary layer. Nevertheless it is assumed that the flow follows the logarithmic law of the wall. To guarantee this assumption it must have the values of y^+ between 30 and 80 after the solution convergence. Figures 5 and 6 show the convergence behavior of the momentum results and the y^+ values. A number of 30 layers was found to be reasonable for numerical accuracy.

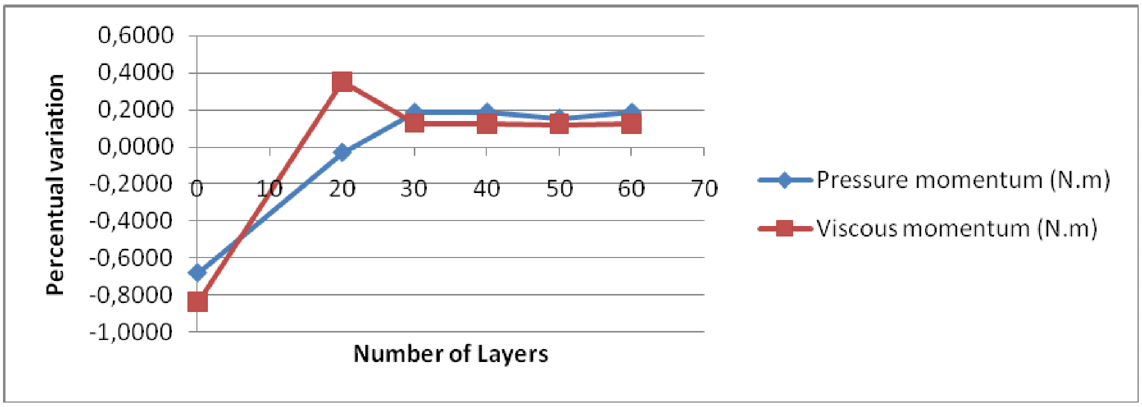

Figure 5 – Momentum convergence behavior due the number of layers

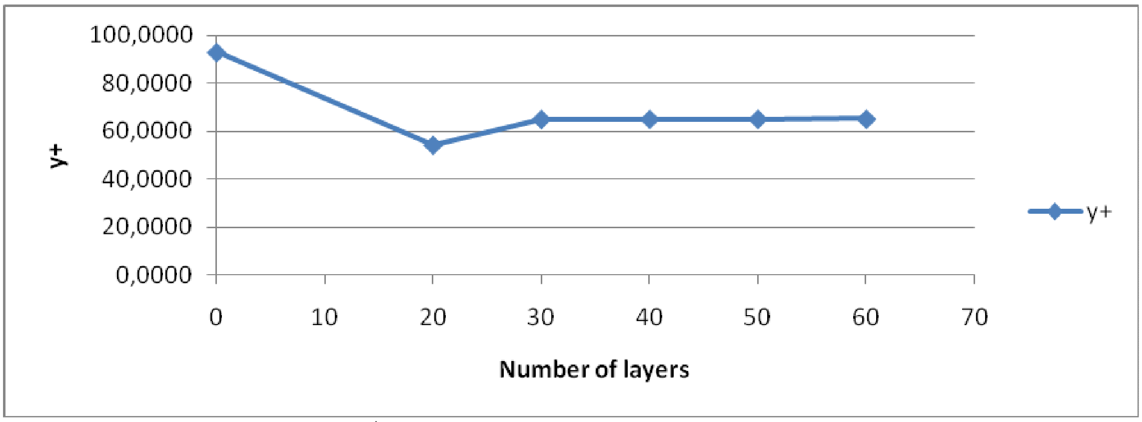

Figure 6 – y^+ convergence behavior due the number of layers

After all the mesh creation parameters were set, the flow was solved for the blade design conditions (tip speed ratio of 5). The streamlines are presented (Fig. 7) and they clearly show the angular momentum transfer to the wake, which is converted into torque to the shaft. It can be seen that the tangential induction factor varies with radial direction, because the vorticity level near the axis of rotation is bigger than in the periphery. This behavior is determined by the BEM theory as a aerodynamic design criteria.

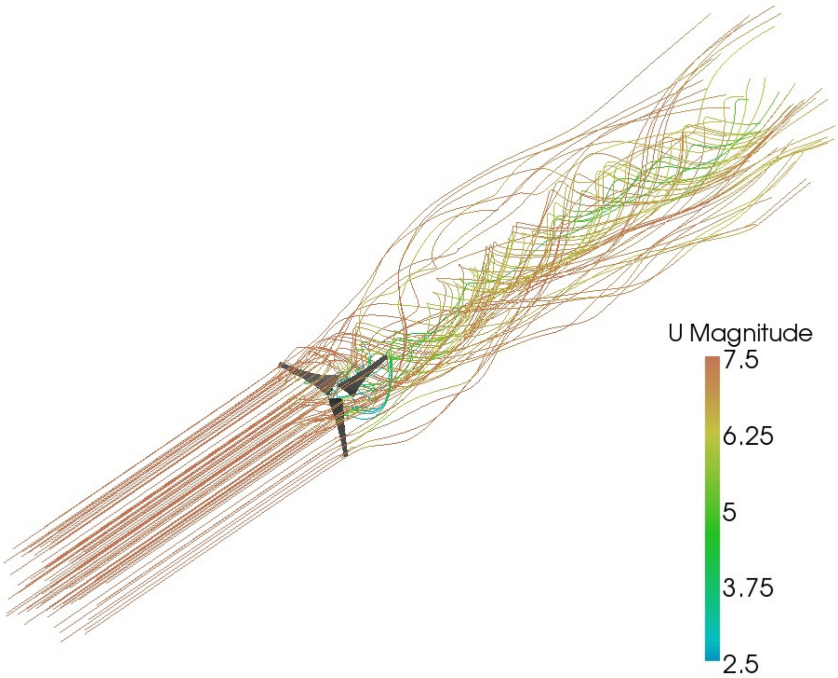

Figure 7 – Streamlines over the wind turbine

Figure 8 presents the pressure and velocity profiles over the axis of rotation. Once again the OpenFOAM code shows agreement with the theory, since there is a pressure drop through the rotating turbine, and the velocity modulus starts to get smaller before the turbine. The velocity loss indicates the transformation kinetic energy into the blades rotational movement. The overall behavior of the stream tube cannot be clearly seen downstream, since the values plotted are over the centerline, that are perturbed by the rotational effects. It can be observed the tendency of the pressure level to return to the free stream value, once the wake is re-energized downstream by the surrounding fluid. This indicates that the chosen length is sufficient for the outlet boundary condition do not propagate into the numerical solution. The same happens with the upstream distance, where the results of velocity and pressure remain stable since 10m.

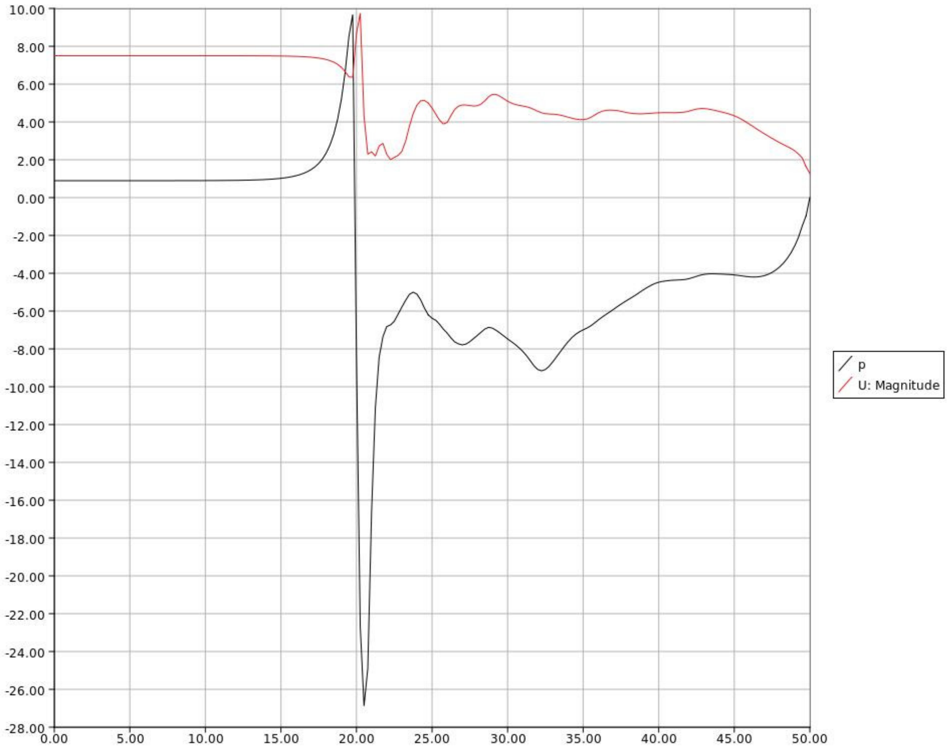

Figure 8 – Pressure and velocity along the axial direction

4. CONCLUSION

For the CFD analysis of a small scale wind turbine, the mesh creation parameters were set. The convergence results showed that the grid independence was obtained to the downstream and upstream lengths. Furthermore, the layers addition process also showed convergence, and the appropriate y^+ to the turbulence model conditions used was reached.

Once the number of refinement loops reached was one, further studies with improved refinement will be done. As long as more computational resources are available, the refinement analysis can be carried on.

The results obtained in the simulation of the flow using the parameterized mesh indicate a good agreement with the predicted by the theory. This was evidenced in the vorticity variation along the radial direction, which is caused by the different tangential induction factor along the blade. This variation is predicted by the BEM theory as a way to optimize the turbine performance. Regarding the upstream results, these present stable values until the inlet boundary, evidentiating that its position does not interfere on the flow over the blade.

3. ACKNOWLEDGEMENTS

The present work is supported by the FUNCAP/CNPq/PPP.

4. REFERENCES

Alvarez, A. C. C., Rocha, P. A. C. and Carneiro, F. O. M., *Aerodynamic project of a threebladed wind turbine for low intensity wind, using the blade element momentum theory*, Annals of the V National Congress of Mechanical Engineering, Salvador, Brazil.

Burton, T., Shape, D., Jenkins, N., Bossanyi, E., *Wind energy handbook*, John Wiley & Sons Ltd., 2001.

Evans, A., Strezov, V., Evans, J. T., *Assessment of sustainability indicators for renewable energy technologies*, Renewable and Sustainable Energy Reviews 13 (2009) 1082-1088.

Schreck, S. J., Robinson, *M., Horizontal axis wind turbine blade aerodynamics in experiment and modeling*, IEEE Transactions on Energy Eonversion, vol. 22, No. 1, march 2007.

Tabor, G., Weller, H. G., Jasak, H., Fureby., C.,*A tensorial approach to computational continuum mechanics using object-oriented techniques*, Computers in Physics, vol. 12, no. 6, nov/dec 1998.

Versteeg, H. K., Malalasekera, W., *An introduction to computational fluid dynamics: The finite volume method*, Prentice Hall, Second Edition, 2007.

Wilcox, D. C., *Turbulence Modeling for CFD*, DCW Industries Inc., La Canada, CA, 1994.

Wright, A. K., Wood, D. H., *The starting and low wind speed behaviour of a small horizontal axis wind turbine*, Journal of Wind Engineering and Industrial Aerodynamics 92 (2004) 1265–1279.