ANALYSIS OF THE FLUID-STRUCTURE INTERACTION EFFECT ON THE FLUID DYNAMIC FORCES OF A WATER FLOW AROUND A NACA 0012 FLEXIBLE BLADE

Franciele Stail Bordin
Adriane Prisco Petry
franciele.bordin@ufrgs.br
adrianep@mecanica.ufrgs.br
Department of Mechanical Engineering – Universidade Federal do Rio Grande do Sul
Rua Sarmento Leite, 425, 90050-170, Porto Alegre, RS, Brasil

Abstract. *Fluid dynamic forces are affected by any change in the shape of the structure. Often, its displacement and deformation are induced by the fluid dynamic forces themselves. Elements of flexible materials are employed in several engineering applications, for instance, in wind turbine blades. This paper presents the study of an external flow using fluid-structure interaction (FSI). The fluid dynamic forces cause the structure to deform, thereby disturbing the initial flow. Numerical simulations are performed in order to evaluate the effect that this deformation has on the fluid dynamic forces. The ANSYS Workbench platform is used, combining the software ANSYS CFX for the fluid analysis and ANSYS Mechanical for the structural analysis. As a form of validation of this method, the laminar flow over an elastically mounted cylinder is studied and compared with literature data. The chosen case for this work is a turbulent water flow over a 3D blade element, fixed at one end and free at the other. The blade geometry is rectangular with the NACA 0012 profile and the turbulence model used is the k-ω SST. The results demonstrate the relevant influence that the deformation of the structure has on the fluid dynamic forces of lift and drag, leading to an agreement with the existing literature.*

Keywords: *Fluid-structure interaction (FSI), Flexible blade, Finite wing, Fluid dynamic forces.*

1 INTRODUCTION

The blade is the most important component in a wind turbine, according to Shokrieh and Rafiee (2006). Casas et al. (2005) said that the efficiency of the blade is determinant for the wind turbine performance. The pursuit of the optimal blade shape and its material composition is a complex problem, according to Jureczko et al. (2005), as the mathematical description of aerodynamic load is complex and a number of constraints and objectives must be satisfied. Tojo (2012) says that there were significant improvements in wind turbines performance accomplished by new blade designs, but that in order to achieve a reliable project that pushes the design margins, a new degree of sophistication had to be introduced into the process: the fluid-structure interaction analysis.

When numerical simulations are performed in order to evaluate the aerodynamic forces produced by a fluid flow, they usually do not take into account the effect that the structural deformation, caused by these forces, has on themselves. The solid body is taken out of the problem, only its contour is represented as a wall. A good analysis would consider the fluid-structure interaction, but this would also consume extra time. Depending on the problem, this interaction can be weaker or stronger. According to Zhang and Huang (2011), since the development of Multi-Megawatt wind turbines, this type of analysis has become a critical issue due to longer and more flexible blades.

According to Jensen et al. (2006) the size of wind turbine blades are expected to increase considerably in the future, demanding a better understanding of its structural behavior. Benini (2002) has studied the flutter phenomenon on a fixed wing considering a potential flow and the fluid-structure interaction, but focused on its structural response. The purpose of this work is to verify if this would considerably change the aerodynamic forces also.

This paper aims to study a simpler problem than modeling an entire wind turbine, as a first step to understand the behavior of the aerodynamic forces when evaluated considering the fluid-structure interaction. The studied case is a turbulent water flow over a NACA 0012 flexible blade.

2 METHODOLOGY

Computational fluid-structure interaction is a multi-physics problem, involving the numerical analysis of both solid and fluid mechanics, plus the coupling between them. The ANSYS software has the MFX-Multiple code solver for FSI problems. It uses iterative coupling and solves each physics and matrix equation separately, combining the ANSYS Mechanical and the CFX software. Setup requires the creation of both fluid and solid domain, mesh and physical models separately, the specification of fluid-solid interfaces for coupled load transfer and a command set to configure the problem and define the solution sequencing. The field solvers are coupled using stagger iterations. At each stagger iteration, the field solver collects loads from the other field solver and proceeds to solve its own physics fields. Iterations continue until all physical field solutions and loads transferred across the interfaces converge.

The numerical analysis of the incompressible fluid flow is based on the Finite Volume Method and the Reynolds Averaged Navier-Stokes (RANS) equations. The Shear Stress Transport (SST) model is used to represent the turbulent effect. The numerical analysis of the structure is performed based on the Finite Element Analysis, with the nonlinear updated lagrangian formulation and a linear elastic material.

2.1 Problem definition

A turbulent water flow over a blade that is fixed at one end and free to vibrate on the other is studied. Figure 1 illustrates the problem. The fluid is water at 25°C and the flow is considered to be incompressible, with a Reynolds number of 6×10^6 and a Mach number of 0.004. The blade has an aspect ratio of 6, a cross section of a NACA 0012 airfoil with a chord of 1 m and an attack angle of 6º. To simplify the problem, the entire blade is considered to be made of steel and the structural damping is disregarded. Table 1 presents the fluid and the blade material properties.

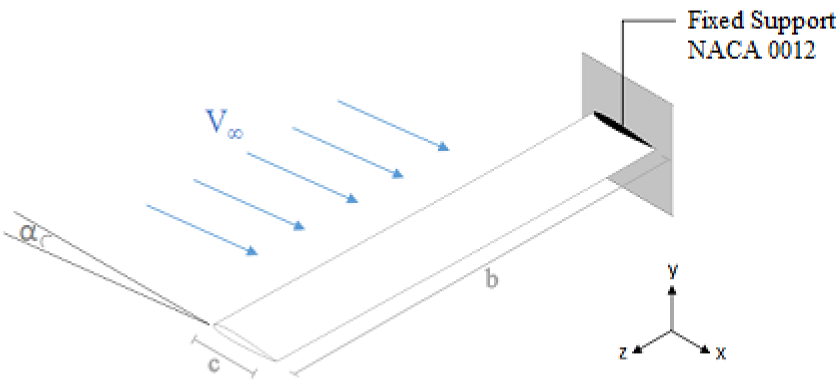


Figure 1. Flow over a flexible blade.

Table 1. Fluid and blade material properties

Water	Steel
Density = 1000 kg/m^3	Density = 7850 kg/m^3
Viscosity = 0.001 Pa·s	Young's modulus = 2×10^{11} Pa
Velocity = 6 m/s	Poisson's ratio = 0.3
Mach Number = 0.004	Yield strength = 2.5×10^8Pa
-	Fracture strength = 4.6×10^8Pa

2.2 Computational domain and boundary conditions

A study is performed and a domain size of 25 m, from the blade to all directions on the plane *xy*, is chosen. The domain size in the *z* direction is 12 m, twice as the length of the blade. The fluid domain is shown in Fig. 2 with its geometry and boundary conditions. Figure 3 shows the structure domain, with its geometry and boundary conditions.

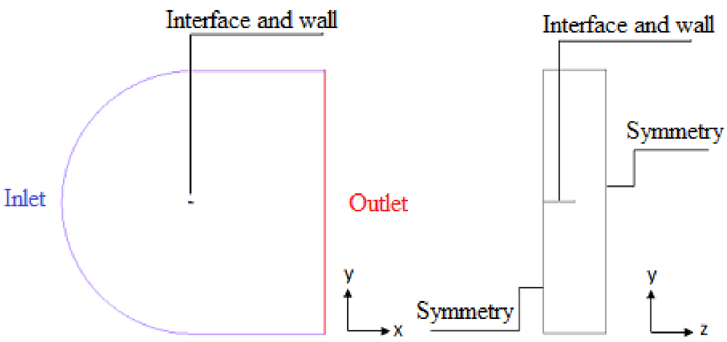


Figure 2. Fluid domain and boundary conditions.

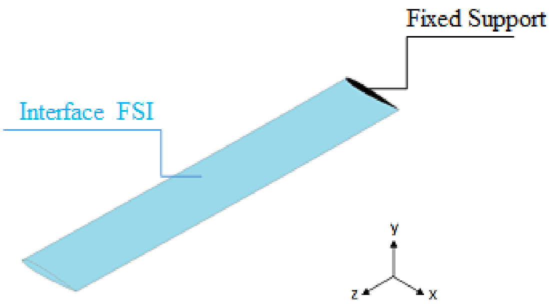


Figure 3. Structural domain and boundary conditions.

3 VALIDATION

With the purpose of validating the methodology, the case of a laminar water flow over a mounted cylinder, allowed to vibrate in only one direction, with a Reynolds number of 105, is studied and compared with the results obtained by Gonçalves (2013), who performed numerical simulations using the Arbitrary Lagrangian-Eulerian method. The initial flow condition is the solution of the last time step of a static cylinder analysis after 30 seconds of flow simulation. Figure 4 illustrates the problem and Tab. 2 presents its details.

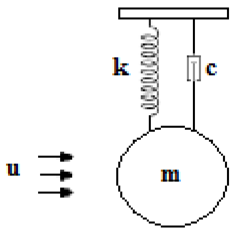
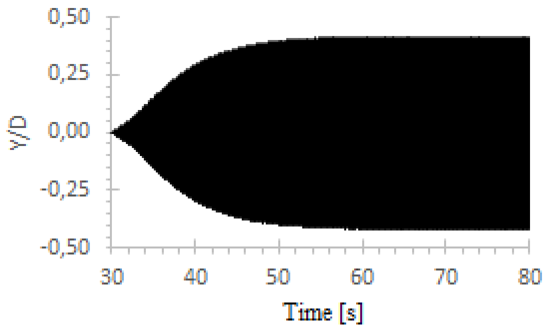


Figure 4. Flow over a mounted cylinder.

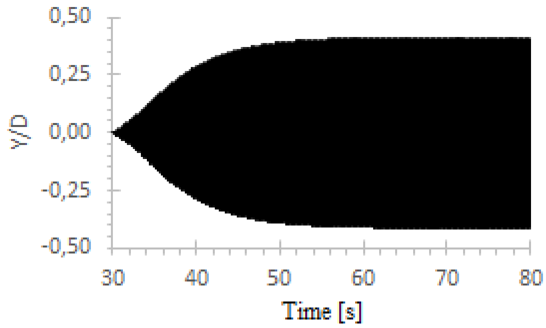
Table 2. Details of the case studied by Gonçalves, 2013

Fluid	Structure
Re = 105	m = 0,2979 kg
V_∞ = 0,065625 m/s	k_a = k 579 N/m
ρ = 1000 kg/m^3	c_a = 0,0325 kg/s
μ = 0,001 kg/ms	D = 0,0016 m

Figure 5 shows the displacement of the cylinder obtained at the present work and by Gonçalves (2013). Table 3 shows the root mean square of the cylinder displacement, drag and lift forces, and their differences in relation to those obtained by Gonçalves (2013).



(a)



(b)

Figure 5. Displacements of the cylinder obtain by (a) this work; and by (b) Gonçalves (2013).

Table 3. Root mean square of cylinder displacement $(y/D)_{rms}$, drag (F_{Drms}) and lift (F_{Lrms}) forces, and their differences in relation to those obtained by Gonçalves (2013)

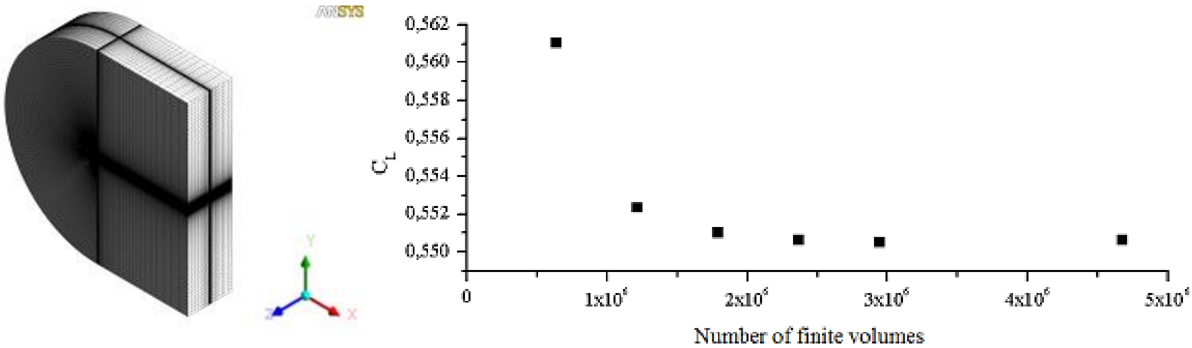
	Present paper [N]	Gonçalves (2013) [N]	Difference [%]
$(y/D)_{rms}$	0,295351	0,292555	0,955841
F_{Drms}	0,006344	0,006151	3,142729
F_{Lrms}	0,001002	0,000969	3,439322

It can be verified that the results are within the expected, agreeing with the data obtained by Gonçalves (2013). Thus, the methodology has been validated for the purpose of this work.

4 RESULTS

4.1 Grid Quality Analysis

A grid quality study is performed for both fluid and solid domains. A simulation of a static blade is conducted for the fluid analysis, with a residual target of 10^{-5}. Since it is considered, for the fluid mechanics analysis, that the blade is symmetric at its fixed end, the value of its lift and drag coefficients are the same as the ones of a blade with an aspect ratio of 12. A curve of its lift coefficient versus number of finite volumes is shown in Fig. 6. The 4th mesh is chosen for the FSI analysis, with 2.291.920 finite volumes.



(a) (b)

Figure 6. (a) Fluid mesh and (b) grid quality analysis for the fluid, with lift coefficient versus number of finite volumes.

For the grid quality study of the structural analysis, the pressure of the fluid on the blade is imported and imposed at all time steps. Figure 7 shows the curve of the number of finite elements versus the *y* displacement of a point situated at the leading edge of the airfoil at the tip of the blade. In this case, the chosen mesh has 4.920 elements. Even though there are other meshes with less number of elements that meet the criteria of convergence, the chosen mesh is the only one that matches the fluid mesh at their common interface, in order to make it easier for the interpolation algorithm.

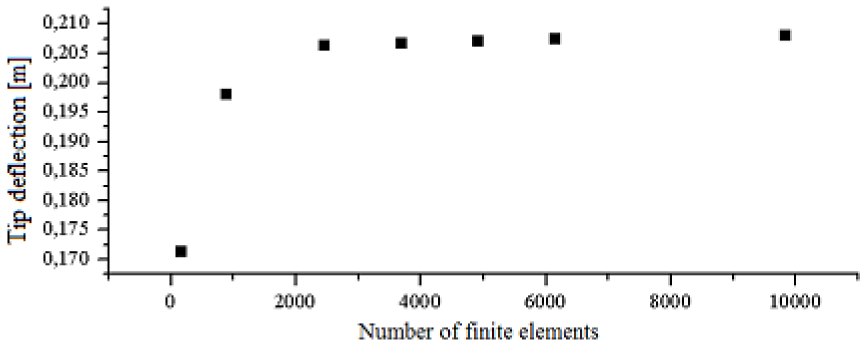


Figure 7. Grid quality analysis for the structure, with tip deflection versus number of elements.

4.2 Structure modal analysis

A modal analysis gives the natural frequencies and mode of vibrations of the blade. Figure 8 and Tab. 4 shows the results obtained by this study.

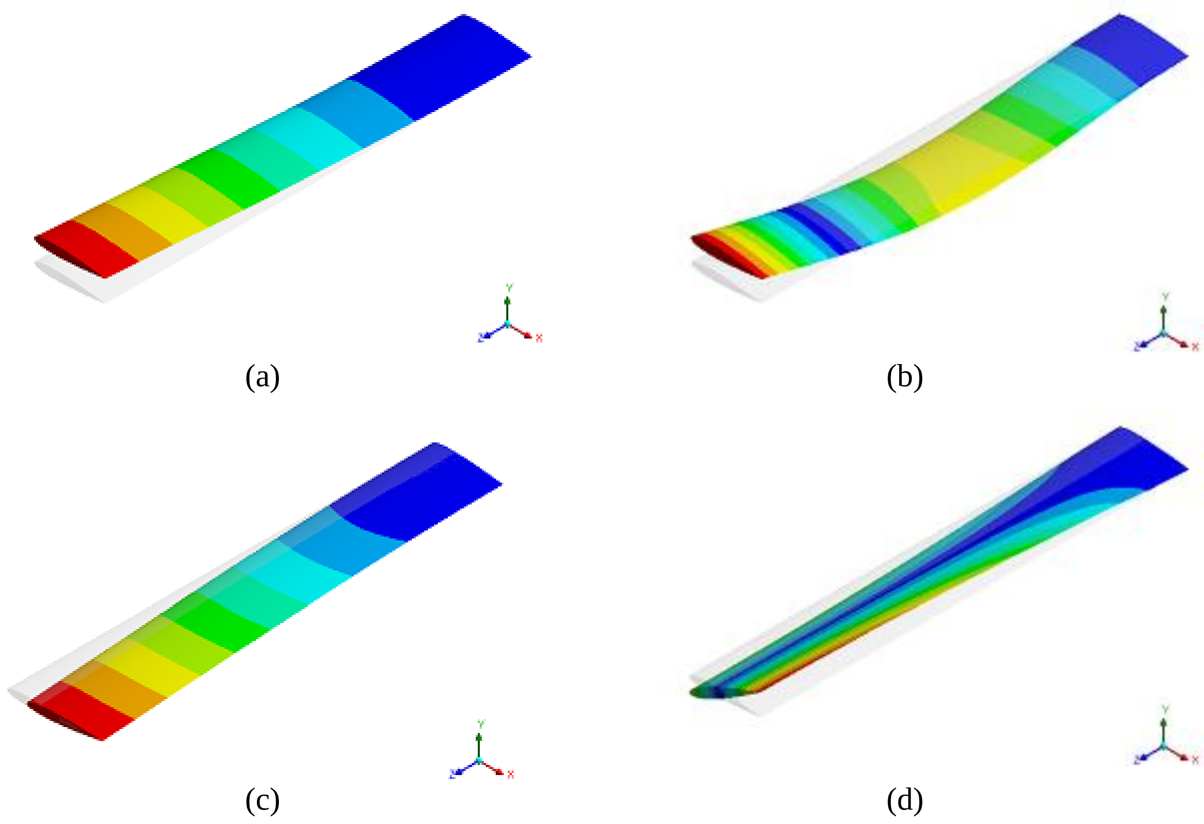


Figure 8. (a) First; (b) second; (c) third and (d) fourth vibration modes.

Table 4. Vibration modes and natural frequencies.

Vibration modes	Natural frequencies [Hz]
1	2,29
2	14,30
3	18,16
4	32,16

4.3 Fluid-structure interaction results

The results obtained by the fluid-structure interaction analysis are presented in this section. The residual target for the fluid simulation variables is 10^{-5} and for the fluid-structure interaction variables is 10^{-3}. The time step is 0.01 s. The initial condition of the flow is the result obtained by the static blade fluid analysis. Figure 9 shows the curve of time versus the *y* displacement of a point situated at the leading edge of the airfoil at the tip of the blade. The geometry of the blade is changed and remains the same after stabilization. The tip of the blade reaches the displacement of 0.134 m at its first cycle. The stabilized configuration shows a displacement of 0.103 m in comparison to the initial one. Figure 10 shows the variation of the angle of attack at the tip of the blade. The tip angle of attack stabilizes at a slightly higher value than the initial, approximately 6.05 degrees.

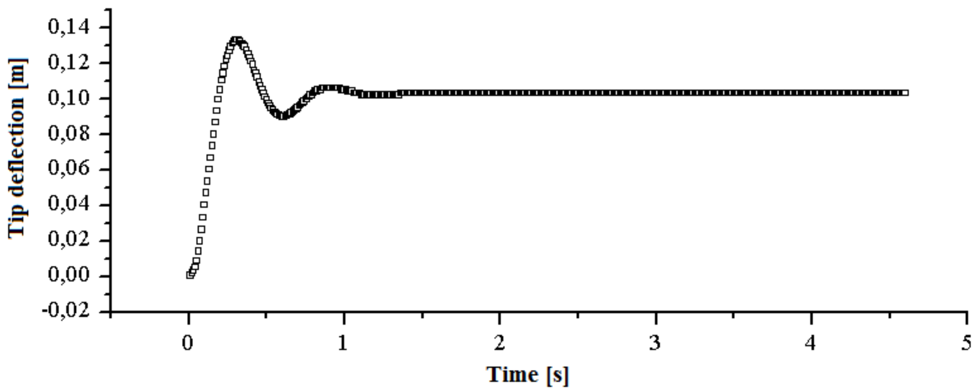


Figure 9. Tip deflection of the blade.

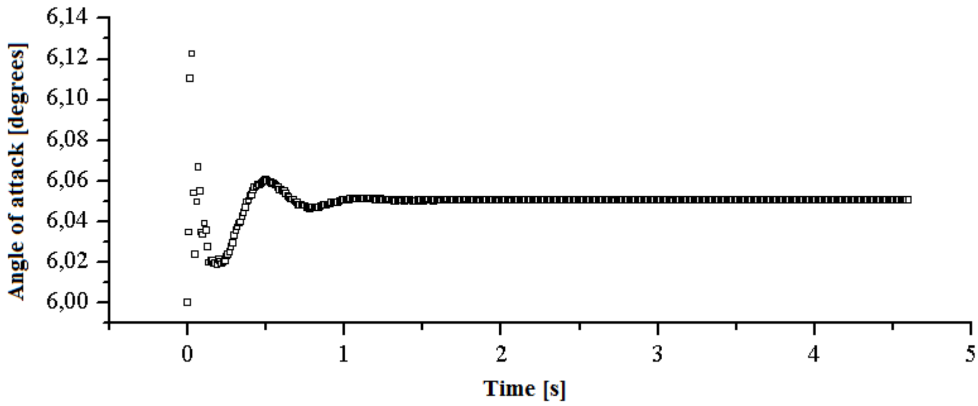


Figure 10. Variation of the angle of attack at the tip of the blade.

Figure 11 shows the variation of the lift force versus time. Figure 12 shows the variation of the drag force. It can be seen that the forces fluctuate at the beginning and then stabilize together with the blade deformation. The lift coefficient stabilizes around 0.8% higher than the static case, with a value of approximately 0.555. The drag coefficient stabilizes around 1.1% lower than expected, with a value of 0.01527.

Figure 13 compares the results obtained in this section with the structural analysis. It is noticed that in the case in question, the disregard of fluid-structure interaction caused the displacements, stresses and strains to be over-estimated. The sudden drop in the lift explains why the displacement of the free end of the blade cannot reach the calculated position in the conventional structural analysis. The difference between the two maximum displacements is 35.5%. The fluid, in this case acts as a damping force on the structure, damping the vibration caused by itself.

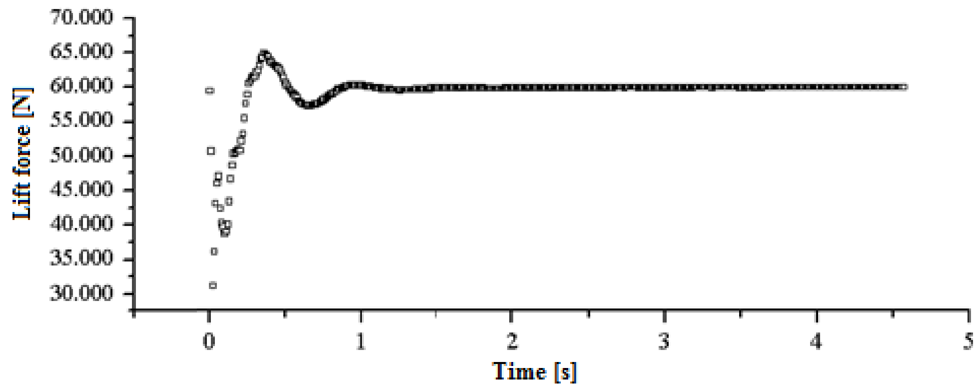


Figure 11. Lift coefficient variation versus time.

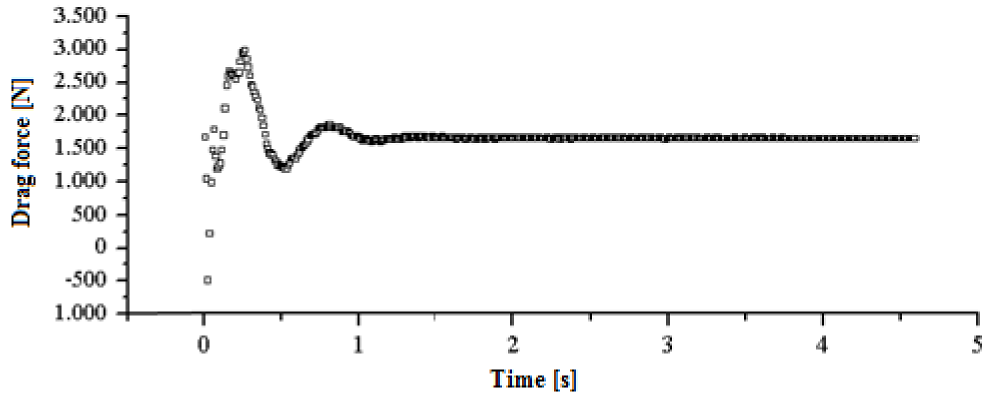


Figure 12. Drag coefficient variation versus time.

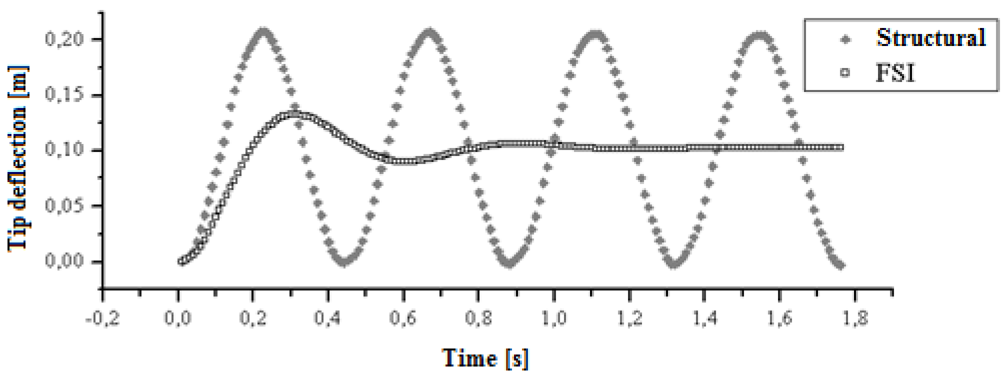


Figure 13. Tip deflection with structural and FSI analysis.

It can be seen that the frequency of vibration of the structure is modified when evaluated through interaction with the water flow. The vibration frequency of the blade tip is 1.64 Hz for the analysis with FSI and the frequency of blade tip angle of attack is 21.19 Hz. Compared to the values of natural frequencies of the structure, you can see a decline in its values. That happens because the water has considerable density, creating an apparent additional mass effect. The structure gains inertia, since it needs to move part of volume of the fluid when it moves. The mass ratio of the problem - the ratio between the mass of the structure and the mass of displaced fluid - is low, with a value of 7.85, indicating the strong influence that the flow has on the inertia of the structure.

5 CONCLUSIONS

A fluid-structure interaction study was performed. In order to validate the methodology, a case of a vibrating cylinder was studied and presented a very good agreement with the data available in the literature. The case of an external flow over a fixed flexible blade was computationally simulated with ANSYS. A grid quality study was performed for both fluid and solid domains. The results show a significant fluctuation of the displacements at the tip blade and its fluid dynamic forces over time. In spite of the fluid of this analysis be water, considering that this analysis was performed with a blade of simpler geometry and material than a real wind turbine blade, design optimization studies are likely to take advantages of the fluid-structure interaction analysis, with geometries and material compositions designed especially for this.

Acknowledgements

The authors gratefully acknowledge financial support from CAPES.

REFERENCES

Benini, G.R., 2002. *Modelo numérico para simulação da resposta aeroelástica de asas fixas*. Master's dissertation, Universidade de São Paulo, São Carlos, SP, Brazil.

Casas, V., Pena, F., Lamas, A. and Duro, R., 2005. "An Evolutionary Environment for Wind Turbine Blade Design." Lecture Notes in Computer Science, Vol. 3512, pp. 1188-1196.

Gonçalves, R.A., 2013. *Análise numérica da interação entre escoamentos a baixos números de Reynolds e cilindros apoiados em base elástica*. Master's dissertation, Universidade Federal do Rio Grande, Rio Grande, RS, Brazil.

Jensen, F., Falzon, B., Ankersen, J. and Stang, H., 2006. "Structural testing and numerical simulation of a 34m composite wind turbine blade." Composite Structures, Vol. 76, pp. 52 – 61.

Jureczko, M., Pawlak, M. and Mężyk, A., 2005. "Optimisation of wind turbine blades." Journal of Materials Processing Technology, Vol. 167, pp. 463–471.

Shokrieh, M.M., Rafiee, R., 2006. "Simulation of fatigue failure in a full composite wind turbine blade." Composite Structures, Vol. 74, pp. 332 – 342.

Tojo, B.M.S.A., 2012. *Aero-Structural Blade Design of a High-Power Wind Turbine*. Master's dissertation, Instituto Superior Técnico, Lisboa, Portugal.

Zhang, P. and Huang, S., 2011. "Review of aeroelasticity for wind turbine: current status, research focus and future perspectives". Frontiers in Energy, Vol. 5, pp. 419–434.