Numerical analysis for prototype blade of Horizontal Wind Axis Turbine in ANSYS Static Structural

To cite this article: D E Husaru and P D Bărsnescu 2018 IOP Conf. Ser.: Mater. Sci. Eng. 444 062010

View the article online for updates and enhancements.

You may also like

- 20th Innovative Manufacturing Engineering and Energy Conference (IManEE 2016)
- Antropometric parameters problem solving of shoe lasts by deforming membranes with medium weight A V Albu, C V Anghel Drugarin, E M Barla et al.
- 7th International Conference on Advanced Concepts in Mechanical Engineering
Numerical analysis for prototype blade of Horizontal Wind Axis Turbine in ANSYS Static Structural

D E Husaru¹ and P D Bârsănescu²
¹Mechanical Engineering, “Gheorghe Asachi” Technical University of Iași, Iași, România
²Mechanical Engineering, Mechatronics and Robotics Department, “Gheorghe Asachi” Technical University of Iași, Iași, Romania

E-mail: husarudorin@yahoo.com

Abstract. In the last decades renewable energy domain has seen a significant development. The wind energy is one of the most important sources of renewable energy. Several research and development directions have been identified to continue installation of the wind turbines: off-shore areas, increased rotor dimensions and installation of small wind turbines. The main objective of this paper is to achieve information about torsional and bending resistance of prototype blade for a three-blade HWAT (Horizontal Wind Axis Turbine). The blade assembly is composed from: a metal hub, a stiffening I-shaped element and blade itself manufactured from composite material. Load condition was created with clamping hub, a concentrated moment on the tip of the blade and a concentrated force at 300 mm from tip. Numerical simulations were performed on all three components of the assembly. Results of two cases of numerical simulation are presented for directional deflection on Y axis and equivalent von Mises stress for each component.

1. Introduction

In the last decades the role of Renewable Energy Sources (RES) becomes very important in modern electric power systems due to their environment friendliness, rising fossil fuels prices and favourable legislation. The wind power is the renewable energy source closest to become competitive on the market. Due to technological improvements and the implementation scale, wind power makes up for the largest share of RES electricity production today, apart from large hydroelectric power plants, [1].

On a global scale, it is necessary to reduce carbon emission. For this action is a multitude of different technologies. The three most widely available renewable energy resources are the sun, the wind and flowing rivers, [2]. Furthermore, the renewable energy, based electricity systems have fixed costs, because the natural resource is fully assessed at the time of deployment, whereas fossil fuel based systems have a fluctuating price, [3]. The increasing importance of wind energy is naturally accompanied by an intense design activity by numerical simulation, in particular for new concept designs, [4].

The understanding and simulation of the wind energy conversion process for wind turbines, single placed or in wind farms, requires the ability to model multiple physical processes taking place at diverse spatial and temporal scales. The efficient design of wind energy systems is based on the fidelity with which mathematical models used in simulation are in line with reality. Consequently, there is a need to validate such models and to calibrate their parameters so as to maximize their accuracy, [5]. The Horizontal Axis Wind Turbine (HAWT) blades are generally oriented in such a way that the wind
direction is roughly normal to rotation plane of the rotor. To maximize the wind energy captured by the blade when the wind direction is variable, the blades are typically twisted.

The most important thing in design of wind turbine is aerodynamic lift. This is formed by two components: a torque component, parallel with the rotation plane of the rotor and causes the blade to spin and a thrust component which gives rise to flap wise bending of the blades, [6].

Composite materials are used for more and more industrial applications. Wind turbines require good structural resistance. That can be obtained with lowest weight for blades and for other components of the assembly. In this way, composite materials appear the best choice in order to reduce the inertia of the machine and therefore its cut-in velocity and inertial loads during operation, [7].

In this paper is analyzed a blade prototype for a Horizontal Wind Axis Turbine with three blades in order to determinate a torsional and blending resistance. For this analyze a finite element simulation performed in Static Structural Workbench form Ansys software was used.

2. Analysis of prototype blade in Ansys
The numerical simulation is the best tool from an economic point of view when prototypes is manufacturing. Finite element analysis provides a method for easy and inexpensive study of the design conditions and manufacturing conditions to be evaluated.

2.1. Model construction
The geometry design of blade assembly was performed in Catia V5. The assembly is composed by 3 elements presented in figure 1, (a) hub, (b) stiffening I-shaped element, (c) blade itself. Every component was design on a real scale. Overall dimension of the assembly is presented in figure 2.

Entire assembly have 1730 mm length, active part of the blade is 1180 mm and hub exterior diameter is 100 mm. The stiffening I-shaped element is placed in interior of the blade between upper side and lower side of the blade making common body with this. The assembly between the stiffening element and the hub is made by two M10 screws.

![Figure 1. Assembly component: (a) hub; (b) stiffening I shaped element; (c) blade.](image-url)
2.2. Assembly materials

For this simulation two types of material were used, steel and fibre glass. Material chosen for the hub is Structural Steel provided by Ansys Material Library, and for stiffening I-shaped element and blade itself a composite material with fibre glass, EPOXY E-Glass UD, also from Ansys Material Library was used. Tensile yield strength for Structural Steel material is 250 MPa.

2.3. Mesh features

In order to have possibility to easily modify a mesh option was created a different mesh for each component. Predominant mesh is formed from tetrahedral cells with maximum element size equal with 8 mm. Total number of mesh elements and mesh nodes are 1338693 respectively 832437. For each component the main mesh feature is presented in table 1: Meshed model are show in figure 3 (a), (b) and (c) for all tree components and in (d) is presented a zoom of the blade mesh. In the trailing edge of the blade the maximum element size is 1 mm and for this an edge sizing method was used. In the neighbourhood of crossing area between active and inactive parts of the blade is created a sphere of influence with 350 mm in diameter with maximum element size 1 mm.

| Assembly component | Predominant mesh | Number of mesh elements | Number of mesh nodes | Maximum element size [mm] |
|--------------------|------------------|-------------------------|----------------------|--------------------------|
| Blade              | Tetrahedral      | 1139553                 | 717540               | 8                        |
| Stiffening element | Tetrahedral      | 114790                  | 60456                | 5                        |
| Hub                | Tetrahedral      | 84350                   | 54441                | 5                        |

Figure 2. Overall dimension of the blade assembly.

Table 1. Mesh features for each assembly component.
2.4. Loading condition

The assembly are fixed on the inside of the hub. On the tip of the blade is applied a concentrated moment on Z axis in clockwise direction in order to simulate torsion behaviour of the blade assembly.

At 300 mm of the tip, on Y axis was created a surface equal with 100 mm. On this surface at negative direction on Y axis is applied a concentrated force in order to simulate blending behaviour. This concentrated force replaces the distributed force placed on entire surface of blade on Y axis.

Two cases with different value from force and moment was simulated. The magnitude of loading condition for that two cases are presented in table 2 and figure 4 show the location of loading condition on assembly in accordance with the features outlined above.

| Case | Force [N] | Moment [Nm] |
|------|-----------|-------------|
| Case 1 | 500       | 250         |
| Case 2 | 1000      | 500         |

Figure 3. Tetrahedral mesh of the components.

Figure 4. Loading condition
3. Results and discussions.
For that two cases, simulations with the same meshing features was performing. In this section, equivalent (von Mises) stress and directional deflection on Y axis for each component are presented.

The figure 5 (a) hub, (b) stiffening element, (c) upper side of the blade, shows equivalent (von Mises) stress distribution obtain for loading condition in Case 1, while the figure 6 show the propagation of directional deflection on Y axis for each element of the assembly with nondeformable wireframe model for Case 1. Maximum directional on Y axis deflection have negative value being of blue colour and minimum directional deflection have positive value being of red colour.

![Figure 5. Equivalent (von Mises) Stress [MPa]-Case 1.](image)

![Figure 6. Directional deflection in negative direction of Y axis [mm]-Case 1.](image)
In the figure 7 is presented (a) hub, (b) stiffening element, (c) upper side of the blade, equivalent (von Mises) stress distribution obtain for loading condition in Case 2, and in the figure 8 is presented the propagation of directional deflection on Y axis for (a) hub, (b) stiffening element, (c) upper side of the blade in Case 2.

Figure 7. Equivalent (von Mises) Stress [MPa]-Case 2.

Figure 8. Directional deflection in negative direction of Y axis [mm]-Case 2.
In Case 1, maximum value of equivalent (von Mises) stress is on hub component with value of 170.15 MPa, in elastic domain, under tensile yield stress limit. Maximum directional deflection on Y axis is on blade component with value of 34.99 mm. The value of directional deflection on Y axis is small in comparison with the overall dimensions of the assembly.

In Case 2, maximum value of equivalent (von Mises) stress is on hub component with value of 341.98 MPa over tensile yield stress limit with value of 250 MPa. Maximum directional deflection on Y axis is on blade component with value of 69.98 mm. The value of directional deflection on Y axis is small in comparison with the overall dimensions of the assembly.

In table 3 is presented the simulation results for each component of the assembly in Case 1 and Case 2.

|        | Solution Assembly component | Maximum value |
|--------|----------------------------|---------------|
| Case 1 | Equivalent (von Mises)  | Blade       | 134.86 |
|        | Stress [MPa]               | Stiffening element | 135.15 |
|        | Hub                        | 170.15       |
|        | Directional deflection in  | Blade       | 34.993 |
|        | negative direction of Y axis [mm] | Stiffening element | 29.913 |
|        | Hub                        | 0.1098       |
| Case 2 | Equivalent (von Mises)  | Blade       | 269.72 |
|        | Stress [MPa]               | Stiffening element | 270.30 |
|        | Hub                        | 341.98       |
|        | Directional deflection in  | Blade       | 69.984 |
|        | negative direction of Y axis [mm] | Stiffening element | 59.825 |
|        | Hub                        | 0.2196       |

4. Conclusion

The finite element analysis is indispensable tool in engineering application. This method is used in prototypes and product development ensuring near-reality modelling with minimal cost. In this paper this method is used for determination of equivalent (von Mises) stress and directional deflection on Y axis for a prototype blade assembly for Horizontal Axis Wind Turbine at torsion and blending.

The simulation results of the two cases indicates that the assembly will withstand torsional and bending loads with force of 500 N placed at 300mm from tip of the blade on Y axis and a moment of 250 Nm placed on the tip of the blade on Z axis. For Case 2 where the loading condition was double that Case 1, equivalent (von Mises) stress for hub element exceeded the tensile yield strength for Structural Steel Material. In this case one of the best solutions is to change the material with another which offer better mechanical proprieties with higher tensile yield strength in order to keep loading condition on the blade.

5. References

[1] Bishop J K and Amaratunga G A 2008 Evaluation of small wind turbines in distributed arrangement as sustainable wind energy option for Barbados Energy Conversion and Management 49 pp 1652–1661
[2] Leary J, While A and Howell R 2012 Locally manufactured wind power technology for sustainable rural electrification Energy Policy 43 pp 173–183
[3] Simic Z, Havelka J G and Vrhovcak M B 2013 Small wind turbines—a unique segment of the wind power market. Renewable Energy 50 pp 1027–1036
[4] Zaghi S, Muscarri R and Di Mascio A 2016 Assessment of blockage effects in wind tunnel testing of wind turbines Journal of Wind Engineering and Industrial Aerodynamics 154 pp 1-9
[5] Bottasso C L, Campagnolo F and Petrović V 2014 Wind tunnel testing of scaled wind turbine models: Beyond aerodynamics Journal of Wind Engineering and Industrial Aerodynamics 127 pp 11-28

[6] Grujićić M, Arakere G, Pandurangan B, Sellappan V, Vallejo A and Ozen M 2010 Multidisciplinary Design Optimization for Glass-Fiber Epoxy-Matrix Composite 5 MW Horizontal-Axis Wind-Turbine Blades Journal of Materials Engineering and Performance 1116 Volume 19

[7] Dal Monte A, Castelli M R and Benini E 2013 Multi-objective structural optimization of a HAWT composite blade Journal of Composite Structures 106 pp 362-373

Acknowledgments
The authors would like to acknowledge the technical resources offered by the Laboratory of Computer-Aided Fluid Engineering, from the Department of Fluid Mechanics, Fluid Machinery and Fluid Power Systems, “Gheorghe Asachi” Technical University of Iasi, Romania. The Laboratory of Computer-Aided Fluid Engineering has been equipped with technical resources with the financial support of the grant ENERED, POSCCE-A2-O2.2.1-2009-4, ID 911.