Material Optimization of a Vertical Axis Wind Turbine using Finite Element Analysis

Faraz Ahmad  
Assistant Professor, Department of Mechanical Engineering,  
Dev Bhoomi Institute of Technology,  
Dehradun, Uttrakhand, India.

Ankit Negi  
B.Tech student, Department of Mechanical Engineering,  
Dev Bhoomi Institute of Technology,  
Dehradun, Uttrakhand, India.

Shubham K. Singh  
B.Tech student, Department of Mechanical Engineering,  
Dev Bhoomi Institute of Technology,  
Dehradun, Uttrakhand, India.

Gaurav Rawat  
B.Tech student, Department of Mechanical Engineering,  
Dev Bhoomi Institute of Technology,  
Dehradun, Uttrakhand, India.

Abstract

A wind turbine is an electromechanical device which converts the wind power into electrical energy. The present study demonstrates the design and simulation of a vertical axis wind turbine. An approximation technique (FEA) was used to analyze the working model. The CAD model was design by Catia and analyzed by Ansys. The model was optimized by vibration frequency to find out best suited material. Three different materials were compared and analyzed by frequency variations. The FEA simulation results demonstrate the area, which was most prone to crack and under vibration condition. The simulation results of the analysis will help in finding out the fitness structures for design optimization.

Keynotes: Wind turbine, FEA, Vibration, Material, Optimization.

Introduction

As the demand for the electricity is increasing day by day and its production was not able to fulfill the requirement. Thus, there is a need of alternative source of energy production, which is independent of the fact that the source of the energy should be renewable and non-polluting. A wind turbine produces no environmental harm during the production of energy [1, 2]. It consists of propeller, which rotates the electrical device for power generation. The present study demonstrates the analysis of a vertical wind turbine used at highway dividers. When a vehicle passes near to the wind turbine, it produces the considerable amount of air due to speedy vehicle. This air strikes the blade of the vertical axis turbine and makes the turbine blades to rotate. The rotation of the turbine generates electrical energy. Furthermore the air striking produces vibration, due to sudden impact of air on the blade. Many researchers contribute in design and analysis of wind turbine to improve the performance. Jihad Rishmany et al. [3] demonstrate the design and analysis of wind turbine blade. Xinzi Tang et al. [4] have performed Finite Element Analysis of wind turbine blade to determine the aerodynamic shape. Suresh Mashyal et al [5] designed a portable highway wind turbine for rural areas. The turbine rotates by the power of moving vehicle air impact and generates the non polluting energy. Ehab Hussein et al. [6] present an experimental study using three-bladed helical vertical axis wind turbines. Results show that prototype can produced up to 48-Watts of power on an average wind speed of 4.4 m/s. L. Ayyadurai et al. [7] have performed an analysis with vertical axis highway windmill for highways lightning problem. Sachin Y. Sayais et al [8] performed the analysis with vertical axis wind turbine as well as with solar system. The energy generates by the solar system and the vertical axis wind turbine was stored into the battery, which can be further used for street lighting. Pabut, O. et al. [9] design the glass fibre reinforced plastic wind turbine blade in ANSYS Workbench. Furthermore the manufactured blade was experimentally tested by bending and model analysis. Meng-Kao Yeh [10] have designed and analyzed the sandwich structure composite wind turbine blade by finite element analysis software.

Material properties and FEA Model

In the present study three types of material was optimized to find out the best suited material for future design aspect. These materials have light weight to strength ratio, good fatigue resistance and wide range of reparability. Furthermore these materials were used in aerospace industry for wings production [11]. The name and the mechanical properties of considered material are listed in table 1.
Table 1: Mechanical properties

<table>
<thead>
<tr>
<th>Materials</th>
<th>Density (kg/m$^3$)</th>
<th>Young's Modulus (MPa)</th>
<th>Poission ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>Aluminium (Alloy 6061-T6)</td>
<td>2712</td>
<td>68900</td>
<td>0.3</td>
</tr>
<tr>
<td>Al 2024</td>
<td>2780</td>
<td>73000</td>
<td>0.3</td>
</tr>
<tr>
<td>CFRP (Carbon Fiber Reinforced Polymer)</td>
<td>1600</td>
<td>70000</td>
<td>0.3</td>
</tr>
</tbody>
</table>

Ansys is a design and analysis software which reduces the problem by solving through mathematical model. Ansys was generally used for structural, thermal, modal and CFD analysis [12]. The cad model of the wind turbine was import to Ansys workbench for analysis. Figure 1 shows the CAD model as well the fixed support. The wind turbine was fixed at the bottom, shown by blue color in figure 1. The geometry was divided in the element by using meshing tool. Figure 2 shows the mesh model. The CAD model of the wind turbine has 36438 numbers of node and 17272 numbers of element.

Result and simulation

In present study modal analysis was performed to calculate the natural frequency of wind turbine for different material properties. The material was optimized by the natural frequency variation under the dynamic loading, generate by the Ansys itself. For modal analysis the boundary condition are the material properties and fixed support. The model was fixed at the bottom plate and the simulation result was calculated for first 4th mode shape for each material. The frequency variation and the mode shape for all three materials are given below.

Figure 1: CAD Model and fixed support

FEA result of aluminum alloy material

Figure 2: Mesh Model
Figure 3: Aluminum alloy made turbine mode shape and deformation at different vibration frequency

FEA result of Al 2024
Figure 4: Al 2024 made turbine mode shape and deformation at different vibration frequency

Figure 5: CFRP made turbine mode shape and deformation at different vibration frequency
In figure 3, 4 and 5 we can see that deformation take place at top rim shown by red color. Furthermore the frequency for aluminum alloy, Al 2024 and CFRP was ranging from 21.711 to 363.16 Hz, 22.041 to 368.69 Hz and 28.45 to 475.89 Hz, respectively. The value of frequency at different mode shape for different material was demonstrated in table 2. Figure 6 shows the frequency comparison and conclude that CFRP has higher frequency failure criteria as compare to other two materials. So that CFRP was selected as the optimum material, which can sustain the vibration during the impact of air.

**Table 2: frequency at different mode shape**

<table>
<thead>
<tr>
<th>Mode No.</th>
<th>Natural Frequency</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Aluminum alloy T6</td>
</tr>
<tr>
<td>1</td>
<td>21.711</td>
</tr>
<tr>
<td>2</td>
<td>21.738</td>
</tr>
<tr>
<td>3</td>
<td>65.448</td>
</tr>
<tr>
<td>4</td>
<td>242.99</td>
</tr>
<tr>
<td>5</td>
<td>362.44</td>
</tr>
<tr>
<td>6</td>
<td>363.16</td>
</tr>
</tbody>
</table>

**Figure 6: frequency comparison**

**Conclusion**

In the present study Preliminary design of wind turbine was developed and the working model of wind turbine was demonstrated by design and analysis. The most prone area of failure was calculated by Ansys workbench, which was shown by red color in mode shape results. The optimum material was found out for batter functioning of wind turbine. Furthermore, in future it will be interesting to check the model experimentally under impact loading of wind.

**Reference**


