MODELLING BEAM VIBRATION UNDER NON-UNIFORM AND NON-STATIONARY TRANSVERSE LOAD

Iyad F. Al-najjar
PhD Student, University of Miskolc, Institute of Machine and Product Design
3515 Miskolc, Miskolc-Egyetemváros, e-mail: iyad.al-najjar@uni-miskolc.hu

Károly Jálics
PhD, Associate Professor, University of Miskolc, Institute of Machine and Product Design
3515 Miskolc, Miskolc-Egyetemváros, e-mail: karoly.jalics@uni-miskolc.hu

László E. Kollár
PhD, Full Professor, Savaria Institute of Technology, ELTE Eötvös Loránd University
Károlyi G. tér 4, 9700 Szombathely, e-mail:kl@inf.elte.hu

Abstract
This paper investigates the vibration of wind turbine blades under transverse load, with the aim of understanding various phenomena that lead to blade vibration such as uneven load due to non-uniform wind; ice on the blade; or sliding of the ice on the blade surface. The subject of this research is a simple model for a wind turbine blade with constant cross section. The model is based on a cantilever beam with a transverse distributed force that changes in time along the blade length. Two models were developed and compared using different approaches. Modal model analysis is employed as an experimental method and a numerical model was also constructed based on the finite element method (FEM). The results show similarity between the behavior of the FEM model and the experimental observations. The results of this study provide insights into the vibration of wind turbine blades and contribute to the development of accurate and reliable models.

Keywords: *transverse vibration, non-uniform loads, modal analysis, FEM, cantilever beam*

1. Introduction

In recent years, the energy of wind power has become an important source of renewable energy due to its environmental sustainability (Wu et al., 2019) and cost-effectiveness (Musgrove, 1987). Horizontal axis Wind turbines (HAWTs) represent the main method of wind power generation, and their efficiency and reliability are crucial to the success of wind energy as a viable alternative to traditional fossil fuel sources. The blades of a wind turbine are the critical component of the wind turbine system. Hence, their performance can drastically impact the overall energy production and maintenance costs.

One of the challenges in designing wind turbines is the vibration of the blades while operating (Igwemezie et al., 2019) due to the effect of uneven loads. Undesired vibration can decrease their lifespan, leads to potential failure (Xie et al., 2020), and in some cases, risks the well-being of Humans and wildlife around it. Therefore, understanding the vibration behavior of wind turbine blades is of great importance in improving their reliability and performance.

Significant research has been carried out in the field of vibration of wind turbine blades, including blade design optimization (Chehouri et al., 2015), the development of vibration control strategies (Awada et al., 2021), and the impact of environmental factors on blade vibration (Leung et al., 2012). However, there is still a need for more research to reveal the effects of transverse uneven loads on the dynamic behavior of wind turbine blades, particularly for a non-uniform load along the beam varying in time.

The present study investigates the effect of a transverse load on the vibration of a cantilever beam. A simple model for a single wind turbine blade is constructed and used to understand its dynamic behavior due to applying a varying transverse load. First, an experimental model is used to find the natural frequencies and mode shapes. Then these frequencies are used to validate a FEM model. Finally, the effect of uneven loads is studied on Ansys.

Overall, this paper aims to contribute to developing accurate and reliable models for wind turbine blades by investigating their vibration behavior under transverse load.

2. Experimental method (modal analysis)

Modal analysis studies the dynamic properties of systems in the frequency domain. Such analysis is carried out to study the free vibration of machines and structures such as a car, a turbine blade, or a wind turbine. There are two methods used for the excitation of the structure for modal measurements. These two methods are the input-output (active excitation) and output only (operational excitation). In the input-output methods of excitation, the procedure involved consists of an excitation function introduced to initiate vibration of the structure. Typically, the forms of excitation entail are impact hammers, drop weights, shakers, or displacement-release. The waveforms used in modal analysis can be of various natures, including harmonic and random input, as well as impulsive excitation. In the output only method, excitation is present if the structure is in service and under some form of external excitation, e.g., traffic or wind loads.

In field testing, dynamic properties are extracted by placing several transducers (commonly accelerometers) at predetermined locations along the structure. To suit the need for mode shape recording, the objective of placing sensors in multiple locations is to attain a sufficient amount of frequency response functions (FRF), such that individual modes can be identified from the modal test. Herein, the highest measurable mode depends largely on the optimal placement of accelerometers; i.e., the extraction of higher modes demands a higher number of accelerometers.

For our experiments a composite specimen was used, which was made of glass fiber reinforced epoxy resin. The main idea was behind, that wind turbine blades are also made of composite material. Table 1 shows the properties and parameters of the specimen, which were also used to build a numerical model in Ansys as described in Section 3. The material parameters were provided by the manufacturer of the composite material.

Table 1. *Parameters and properties of the Experimental Model*

Total length	425 mm	Width	58 mm
Clamped length	22 mm	Height	10 mm
Free length	403 mm	Young's modulus	25 GPa
Density	1850 kg/m^3		

We defined overall 33 measurement points on the plate, to be able to represent also the local mode shapes along the global mode shapes. This amount of points should be sufficient enough for that purpose. The distance of the points was 25 mm along the long side, and 35 mm along the short side of the specimen. Measurement point Nr. 33 was also the excitation point.

For the measurements we used test equipment B&K Photon+ frontend, B&K RT Pro Photon software, 2 pieces of B&K 4397 uniaxial accelerometer, and Endevco 2202-10 impact hammer. We decided to perform the fixed hammer excitation method (roving response method) instead of roving hammer excitation. The schematic representation of the measurement setup is described in Figure 1 (Trendafilova and Heylen 1998).

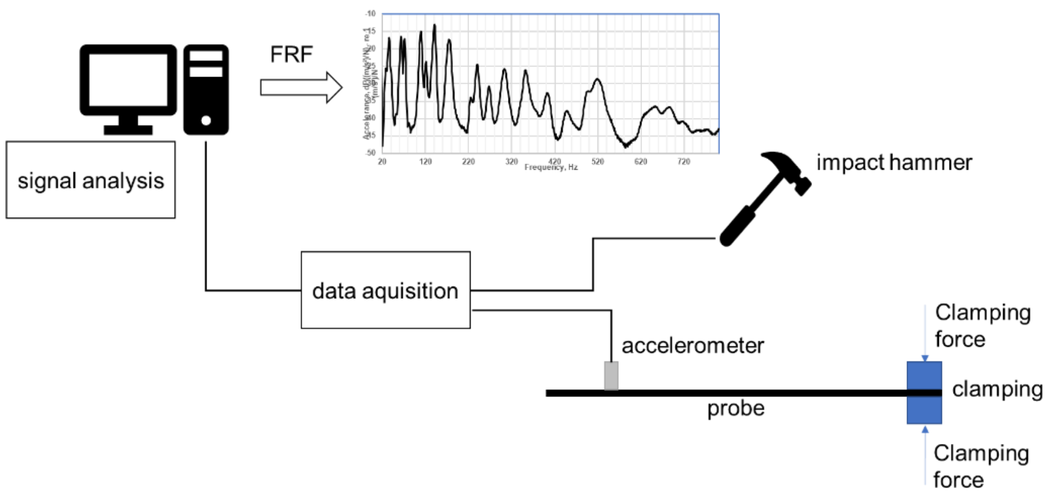


Figure 1. *The schematics of the experimental setup*

Figure 2 presents the experimental set-up. We clamped the specimen in a parallel vise and placed 2 accelerometers on the first and second measurement points. We performed pre – tests regadring the clamping force, where we used a range of clamping forces. We applied a clamping force from the tested range, where the measured transfer functions did not change significantly.

After that we hit on the excitation point, 10 times, the recorded FRFs were averaged. We repeated the measurement also for the remaining measurement points. We checked during the test the quality of the FRFs, the coherences, the spectrum and time signal of the excitation (e.g. to avoid double hits). The measurements are carried out with the following software setup:

1. Frequency range: 0 – 18750 Hz
2. Frequency resolution: 0,732 Hz
3. Excitation signal: impulse
4. Number of averages: 10 hits with linear average
5. Uniform window (the vibration decays within the sample time of one average (1,366 s), so spectral leakage is not to be expexted; exponential window is not necessary)
6. Measuring transfer functions (FRF): a/F; for each FRF also the coherence function

Figure 2. *Experimental setup for a cantilever beam with hammer excitation*

3. The finite element model of the cantilever beam

The finite element method (FEM) is a numerical technique for solving partial differential equations that arise in various engineering and scientific problems. A common application of FEM is structural analysis, where the behavior of complex structures under various loading conditions is analyzed.

To use FEM to analyze the behavior of a cantilever beam with lateral load, the beam is first discretized into a finite number of elements. Each element is then approximated by a simple form, such as a linear or quadratic polynomial function. The behavior of the entire beam can then be approximated by a system of linear equations relating the deformation of each element to the shear load acting on the beam.

The FEM approach allows for efficient and accurate analysis of complex structural systems, including those with non-uniform material properties or complex geometries. By dividing the system into smaller, simpler elements, the FEM approach can capture the behavior of the system at a high level of detail while remaining computationally manageable.

This section will describe details of the finite element model. The commercial software Ansys is applied to construct the model, and different possible transverse loads on the beam will be considered. The mesh will be described including the choice of element size and type. The natural frequencies determined by the FEM will be compared to those obtained in the experiments.

The subject of this study is a beam with constant cross section A as shown in Figure 3. The parameter ρ denotes the beam's mass density, E is the Young's modulus of the beam's material, and I is the beam's cross-sectional moment of inertia about the neutral axis. A composite material was used whose properties were taken from ANSYS engineering data library. Epoxy E-glass wet material was assumed after editing where the density increased to 1857,53 kg/m^3 and the Young’s modulus to 25 GPa based on Table 1.

A 425 mm long beam was turned into a cantilever beam by clamping it at one end. This beam was meshed using an element size of approx. 10x10x5 mm (see equations (1)-(3)) and the default element type SHELL181 element. The meshed beam is illustrated in Figure 4, where the green parts show where the fixed supports are placed with 22 mm length.

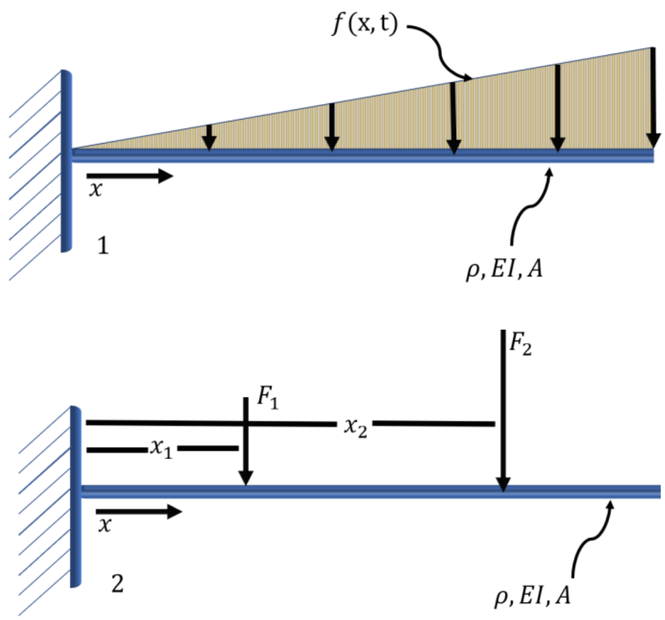


Figure 3. *Cantilever beam 1. Linearly distributed load 2. Concentrated forces*

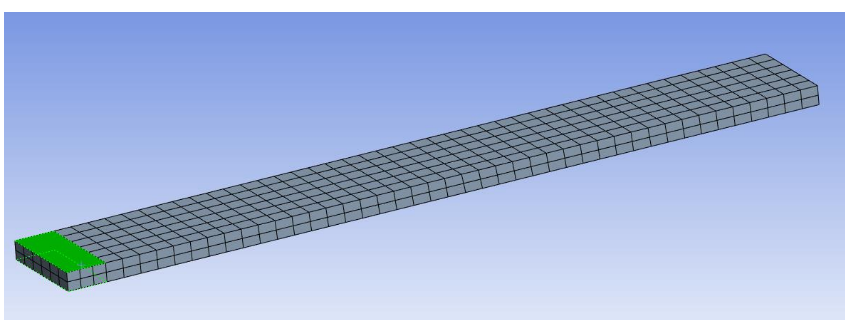


Figure 4. *Demonstration of meshed beam with fixed end*

The choice of element size is based on the following facts. Refining the mesh increases the accuracy of the final result but in return increases the time needed to solve the problem. The following equations should be considered to find the maximum element size d_{max} (Marburg, 2008)

$$d_{max} = \frac{\lambda_B}{N}, \tag{1}$$

where N is the number of elements per wavelength and λ_B is the wavelength of the bending waves in the beam during its vibration related to the highest frequency to be investigated

$$\lambda_B = \frac{c_B}{f_B} \tag{2}$$

with f_B denoting the bending frequency (the highest frequency to be investigated) and c_B denoting the speed of sound (of bending waves):

$$c_B = \sqrt[4]{\omega^2 \frac{B}{m''}} \tag{3}$$

In Equation 3 (Cremer et al., 2005) ω is the natural angular frequency, B is the bending stiffness and m'' is the specific mass.

First, modal analysis is carried out, and the computed natural frequencies are compared to those obtained experimentally. In Table 2, the first three natural frequencies are compared. Numerical and experimental results coincide, the discrepancy is below 3% for each of the three frequencies. Also, Figure 5 to Figure 7 shows the first three mode shapes obtained from the FEM and the experiment. In all cases, the left side of the beam is clamped. Also, the mode shapes obtained by Ansys are to the left side of the figure while the experiment mode shapes are to the right. The first, second and third mode shapes are shown in Figure 5, Figure 6 and Figure 7 respectively. For the FEM the blue color means no deformation while the red parts are the maximum deformation for each case. Hence, the fixed parts are blue. For the experiment part, the points in each figure refer to the local displacement for each accelerometer, so the figure shows the local deformation for each point.

Table 2. *Comparison between FEM and experimental modal analysis*

	Experiment frequency (Hz)	FEM frequency (Hz)	Error %
1.	38	36,899	2,8
2.	227	229,76	1,21
3.	620	637,01	2,25

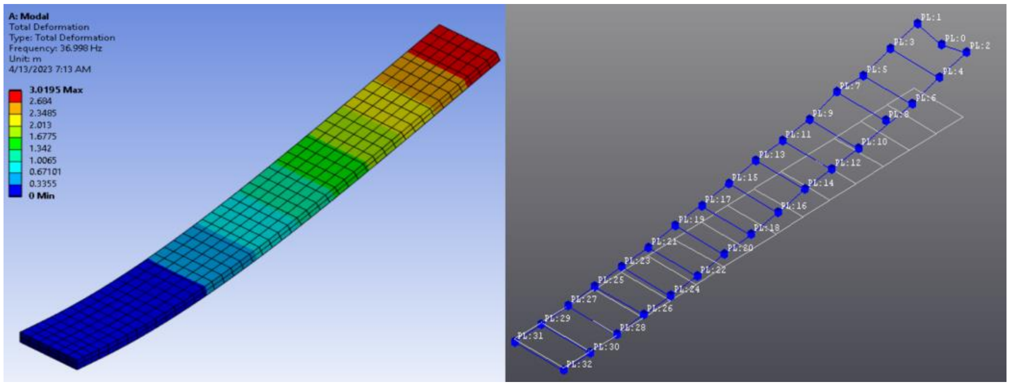


Figure 5. *The first Mode Shape from the FEM and the experiment*

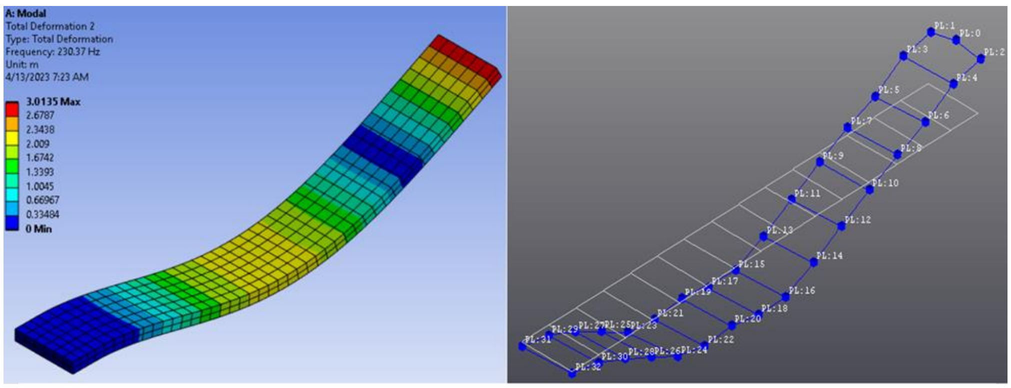


Figure 6. *The second Mode Shape from the FEM and the experiment*

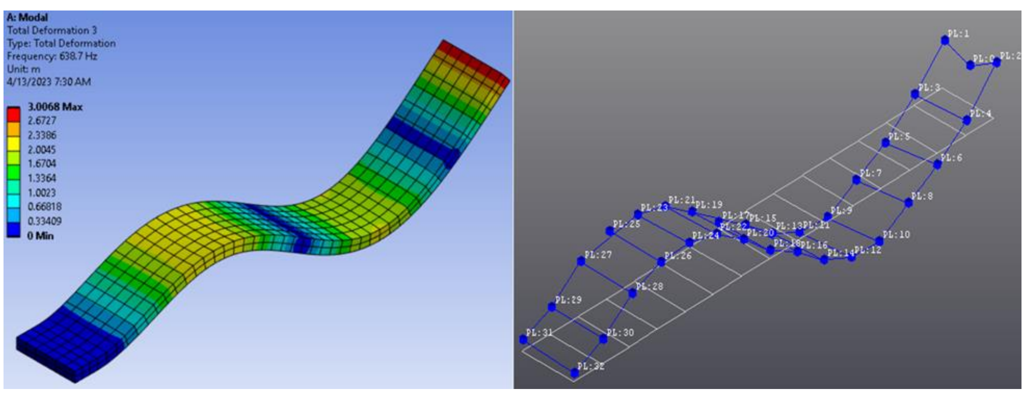
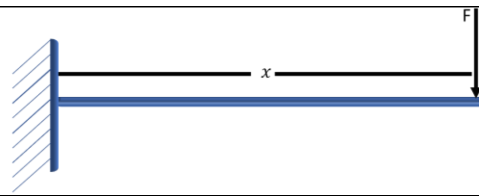
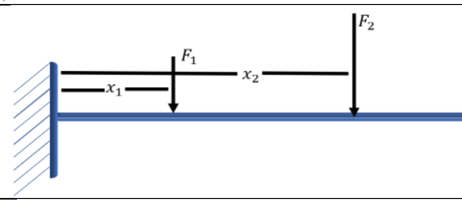
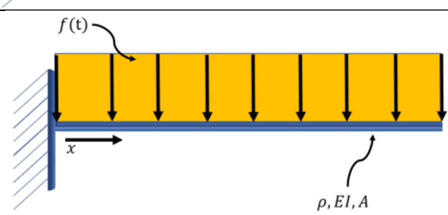


Figure 7. *The third Mode Shape from the FEM and the experiment*

In the second set of simulations, the beam is subjected to loads in the form $f = f(x,t)$ that vary in time and along the beam. Table 3 shows these loads and the maximum deformation obtained from ANSYS for different cases. For the first case, the force was placed at the free end as a function of time. This force may represent the effect of periodic vortex shedding due to wind locally on a shaft or on a turbine blade. For the second case the two forces were placed at two different positions. One, at 0,325 m from the fixed end and one from 0.225 m from the fixed end. Such static loads may be considered partial ice loads on a blade surface.

Finally, the third force was simulated using the tabular option in ANSYS where the force was a table of inputs that describes the force as pressure on an area for each 0,1 second. The magnitude of this force reduces exponentially, which may be the case when ice sheds gradually from a blade surface due to melting.

***Table 3**. Deformation at different load configurations*

	Force	Load case	Max Deformation (m)
1	$F = sin(10\,\pi\, t\ )$ At the free end		After $t = 50$ $1.337\text{x}\ 10^{-4}$
2	$F = \begin{cases} -100\ at\ x_1 \\ 30\quad at\ x_2 \end{cases}$ $x_1 = 0.225$ $x_2 = 0.325$		At the free end $1.977\text{x}\ 10^{-3}$
3	$F(t) = w\ L\ \frac{1}{e^t}$ After 2 sec L: length of the beam; w: width of the beam		At the free end $2.05\text{x}\ 10^{-5}$

For the third case, the force is inserted in Ansys as a pressure along the beam where the force depends on the tabular input. Hence, the final force equals the area of the top surface of the beam multiplied by the value of $\frac{1}{e^t}$ at each input.

4. Summary and conclusion

This paper aimed to build and study vibration of a simplified model for a wind turbine blade, which will serve as a basis for future developments. The blade was modelled as a cantilever beam that was the subject of experimental modal analysis. In the experiment, 33 measurement points were defined on the beam and used to create mode shapes. Then, the finite element model of the beam was also constructed, and the first three natural frequencies were determined and compared to experimental results to validate the FEM model. The FEM model showed less than 3 percent error for the first three natural frequencies. After considering a proper mesh size and type, the FEM can be used to find the deformation for different possible load configurations including distributed and non-uniform loads that change along the beam, i.e., along the x-axis or in time. Further simulations to obtain the characteristics of vibration due to these loads is the subject of ongoing and future research.

Overall, the results can be used to gain insight into the deformation of a simplified blade during the vibration due to time-dependent loads.

5. Acknowledgements

Project No. TKP2021-NVA-29 has been implemented with the support provided by the Ministry of Innovation and Technology of Hungary from the National Research, Development and Innovation Fund, financed under the TKP2021-NVA funding scheme.

References

[1] Awada, A., Younes, R., Ilinca, A. (2021). Review of vibration control methods for wind turbines. *Energies*, 14(11), 3058. **https://doi.org/10.3390/en14113058**

[2] Chehouri, A., Younes, R., Ilinca, A., Perron, J. (2015). Review of performance optimization techniques applied to wind turbines. *Applied Energy*, 142, 361–388. **https://doi.org/10.1016/j.apenergy.2014.12.043**

[3] Cremer, L., Heckl, M., Petersson, B. A. T. (2005). *Structure-borne sound*. Springer Verlag, ISBN 3-540-22696-6 **https://doi.org/10.1007/b137728**

[4] Igwemezie, V., Mehmanparast, A., Kolios, A. (2019). Current trend in offshore wind energy sector and material requirements for fatigue resistance improvement in large wind turbine support structures – A Review. *Renewable and Sustainable Energy Reviews*, 101, 181–196. **https://doi.org/10.1016/j.rser.2018.11.002**

[5] Leung, D. Y. C., Yang, Y. (2012). Wind energy development and its environmental impact: A Review. *Renewable and Sustainable Energy Reviews*, 16(1), 1031–1039. **https://doi.org/10.1016/j.rser.2011.09.024**

[6] Marburg, S. (2008). Discretization requirements: How many elements per wavelength are necessary? In: Marburg, S., Nolte, B. (Eds.), *Computational Acoustics of Noise Propagation in Fluids -Finite and Boundary Element Methods* (pp. 309–332). Springer, Berlin, Heidelberg. **https://doi.org/10.1007/978-3-540-77448-8_12**

[7] Musgrove, P. J. (1987). Wind energy conversion: Recent progress and future prospects. *Solar and Wind Technology*, 4(1), 37–49. **https://doi.org/10.1016/0741-983X(87)90006-3**

[8] Trendafilova, I., Heylen, W. (1998). Fault localization in structures from remote FRF measurements. Influence of the Measurement Points. *Proceedings of the 23rd International Conference on Noise and Vibration Engineering (ISMA)*, pp. 265–272.

[9] Wu, X et al. (2019). Foundations of offshore wind turbines: A Review. *Renewable and Sustainable Energy Reviews*, 104, 379–393. **https://doi.org/10.1016/j.rser.2019.01.012**

[10] Xie, F., Mousaad, A. (2020). Structural control and vibration issues in wind turbines: A Review. *Engineering Structures*, 210, 110087. **https://doi.org/10.1016/j.engstruct.2019.110087**