

Numerical Modeling of the Vibration of Reinforced Concrete Beam

Leonardo Andretta Lobo¹, Elvidio Gavassoni Neto², Amanda Jarek³, Marcos Antônio Teixeira Neri⁴, Rodrigo Augusto do Nascimento Gomes⁵

¹ UFPR Federal University of Parana / landrettalobo@gmail.com

² UFPR Federal University of Parana / gavassoni@ufpr.br

³ Lactec Institutes / amanda.jarek@lactec.org.br

⁴ Neoenergia / marcos.neri@neoenergia.com

⁵ Neoenergia / rodrigo.gomes@neoenergia.com

Abstract

Structural Health Monitoring (SHM) can be vastly used to verify the state of a structure, avoiding the need of destructive tests to do so. One of the many ways of perform SHM is through obtaining and evaluating modal parameters (natural frequencies and mode shapes) of certain structure, collected by dynamic tests, and compare them to pre-established values for structures without damage nor deterioration. Such monitoring can be performed with a reduced maintenance cost and monitoring in the field of electricity distribution, which uses a high number of reinforced concrete light poles. Besides that, the structural health monitoring of those poles contributes to the safety of workers and third party employees who work in the maintenance of power distribution networks. This paper is part of a program that seeks to develop a methodology of structural health monitoring of reinforced concrete light poles using modal analysis. Therefore, values measured in quick tests and compared with predicted data by mathematical models of the pole's vibration can indicate the state of maintenance safety of these structures. One of the first steps for this is the validation of the structural model. This paper pretends to design a numerical model in a Finite Element (FE) software (ANSYS) of a laboratory tested reinforced concrete beam, and after compare the modal parameters obtained numerically to experimental values and the ones calculated by analytical solutions properly adjusted to fit the reinforced concrete features. The model considers the behaviour of a composite material such as reinforced concrete and the free-free boundary conditions identical to the ones used in the first testings. The comparative results between the numerical models and experimental tests are satisfactory in such way that they validate the model as proper in the attainment of reinforced concrete light poles's modal parameters.

Key words

Structural Health Monitoring; Finite Element; Reinforced Concrete; Validation; Modal Analysis; Vibration

Introduction

During the last few years, some private energy supplying groups in Brazil have been suffering several and serious accidents related to its reinforced concrete light poles, which have been presenting failure during the execution of periodic maintenance. Through the reports presented by the companies themselves, a preliminary risk analysis has been made

to guarantee that the execution of such services is safe. Despite that, even though the protocol was followed, there were some signals that the structure was damaged which were not noticed, resulting on its failure and ruin. There were also situations in which the structure seemed to be integer, but when the employee started climbing the pole, it would fail in points underneath the ground.

Being that said, it became necessary to start monitoring the structural health of the light poles in a non-visual manner. According to Zong, Lin, & Niu (2015), in the practice of Engineering, the approximation between statistical analysis and the structural evaluation is a tendency in the future. Because of that, the study of Engineering problems depends even more in mathematical and computational models, instead of empiric and expensive evaluations.

The main objective of this paper is to compare analytical and computational models, which have been designed in ANSYS, allowing the evaluation of the integrity of reinforced concrete prototypes through dynamic tests, and comparing its results with the ones executed in laboratories.

First model: homogeneous material, free-free beam

To perform a computational modeling using Finite Element software, the first step taken was to create a simple beam model, with a homogeneous material, a constant square-shaped cross section throughout the whole extension of the structure, and having a Free-Free boundary condition, which means the structure is able to move freely, without any supports. This model was designed in ANSYS Workbench, using the material and geometrical properties shown in Table 1 and Table 2.

Table 1 - Material Properties

Young's Modulus (E)	200 GPa
Density (ρ)	7850 kg/m ³
Poisson's Ratio (ν)	0.3

Table 2 - Geometrical Properties of the Beam

Area (A)	0.09 m ²
Moment of Inertia (I)	6.75e-4 m ⁴
Length	3.0 m

Using the Modal analysis system in ANSYS Workbench, three different models have been designed, with the intention to verify the variations in the results comparing models with one, two, or three dimensions. Figure 1 shows the meshed 3D model, with 80 cubic elements. The three first flexural vibration frequencies found in each of the models are shown in Table 3, while first torsional and axial vibration frequencies are shown in Table 4 and Table 5, respectively.

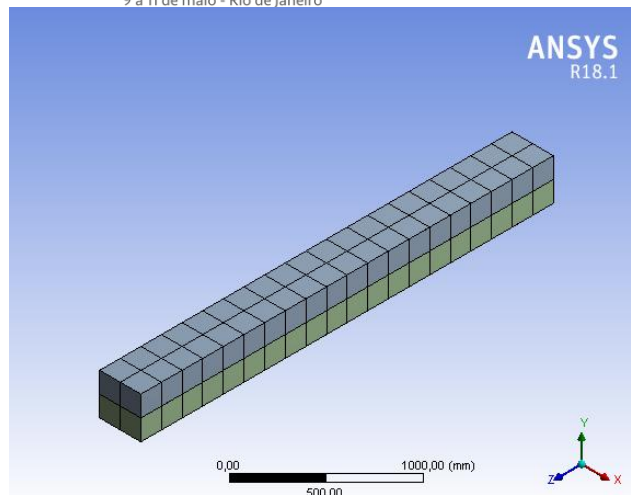


Figure 1 - 3D meshed beam model

Table 3 - Flexural Vibration Frequencies of the Beam (Hz)

Mode (n)	1D	2D	3D
1	167.06	167.46	167.16
2	434.73	438.39	435.6
3	793.37	806.56	796.27

Table 4 – First Torsional Vibration Frequency of the Beam (Hz)

Mode (n)	1D	2D	3D
1	482.64	497.79	480.73

Table 5: First Axial Vibration Frequency of the Beam (Hz)

Mode (n)	1D	2D	3D
1	841.26	841.54	840.63

Having the computational model results been obtained, it was necessary to validate the data with a mathematical model. According to Clough & Penzien (2003), Meirovitch (2001) and J.L.Humar (2001), the equations used to calculate the natural vibration frequencies (ω) of the theoretical beam are written as follows, being Equations (1), (2) and (3) used to determine the first three flexural vibration frequencies of the free-free beam. The equations (4) and (5) were used to calculate the first torsional and axial vibration frequencies of the beam, respectively

$$\omega_{f1} = \frac{(4.73)^2}{L^2} \sqrt{\frac{EI}{\rho A}} \quad (1)$$

$$\omega_{f2} = \frac{(7.853)^2}{L^2} \sqrt{\frac{EI}{\rho A}} \quad (2)$$

$$\omega_{f3} = \frac{(10.996)^2}{L^2} \sqrt{\frac{EI}{\rho A}} \quad (3)$$

$$\omega_{t1} = \frac{n\pi}{L} \sqrt{\frac{G}{\rho}} \quad (4)$$

$$\omega_{a1} = \frac{n\pi}{L} \sqrt{\frac{E}{\rho}} \quad (5)$$

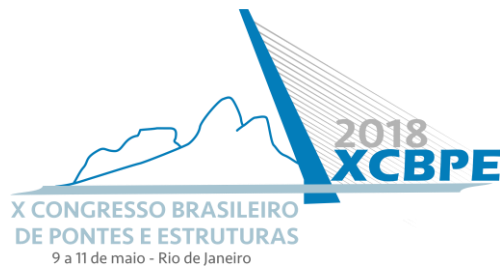
Comparing the results obtained from the analytical solutions and the computational model, it is noticeable that the data calculated by FEM is quite accurate, as can be seen in Table 6. To verify the consistency of the values obtained for the frequencies in ANSYS, the errors between the computational model and the analytical solution were calculated and shown in Table 7.

Table 6- Analytical and Computational Model Frequencies

Vibration Type	Mode	Analytical Solution	Models		
			1D	2D	3D
Flexural	1	172.95	167.06	167.46	167.16
	2	476.72	434.73	438.39	435.6
	3	934.67	793.37	806.56	796.27
Torsional	1	521.73	482.64	497.79	480.73
Axial	1	841.26	841.26	841.54	840.63

Table 7 - Computational Model Errors

Vibration Type	Mode	1D	2D	3D
		Error	Error	Error
Flexural	1	3.40%	3.17%	3.35%
	2	8.81%	8.04%	8.62%
	3	15.12%	13.71%	14.81%
Torsional	1	7.49%	4.59%	7.86%
Axial	1	0.00%	-0.03%	0.07%



With these results, it was possible to update the computational model and add Reinforced Concrete properties to the material, leading to a comparison with an experimental reinforced concrete beam designed, made and dynamically tested in laboratory.

Reinforced Concrete (RC) beam ANSYS and experimental models

In order to perform a Structural Health Monitoring on the light prototypes, it is necessary to have a calibrated and trustworthy computational model that calculates accurately the structure's natural vibration frequencies. To design a model, it requires several parameters and considerations to be done, such as elastic supports, non-constant cross-section, damage and reinforced concrete material. Creating the characteristics of RC in ANSYS is complex, since Workbench doesn't have the feature to create a composite material on its Engineering Data, so the material properties had to be implemented by APDL commands. The element used to simulate the concrete material was SOLID65, which is used for 3D modelling of solids, with or without reinforcement bars (also called rebars). The solid is capable of cracking in tension and crushing in compression. In order to particularize the properties of the material to the model designed, some APDL commands have been implemented into SOLID65's properties, setting the concrete's Young's Modulus to 34.7 GPa, Poisson's Ratio to 0.2 and density to 2289 kg/m³. These values were obtained through laboratory tests, done in concrete cylinders (300 mm height and 150 mm diameter). For the steel reinforcement and hooks, the element used was LINK180, which is a 3D spar useful in a variety of Engineering applications. The element can be used to model trusses, cables, links, springs and so on, and is a uniaxial tension-compression element with three degrees-of-freedom (DoF) at each node: translations in the nodal x, y and z directions. The steel properties considered were 200 GPa Young's Modulus and 0.3 Poisson's Ratio. To create the connection between both elements, the CEINTF APDL command was used, which 'ties' together two regions with dissimilar mesh patterns by generating constraint equations that connect the selected nodes of one region to the selected nodes of the other region. The regions selected were from the elements SOLID65 and LINK180.

The beam models (both experimental and computational, shown in Figure 2 and Figure 3, respectively) have a 1.40 m length, 10 x 10 cm cross-section, fourteen stirrups, four reinforcement bars, two central and one extremity hook, which were used to lift the beam during experimental testing, which can be seen in Figure 4.

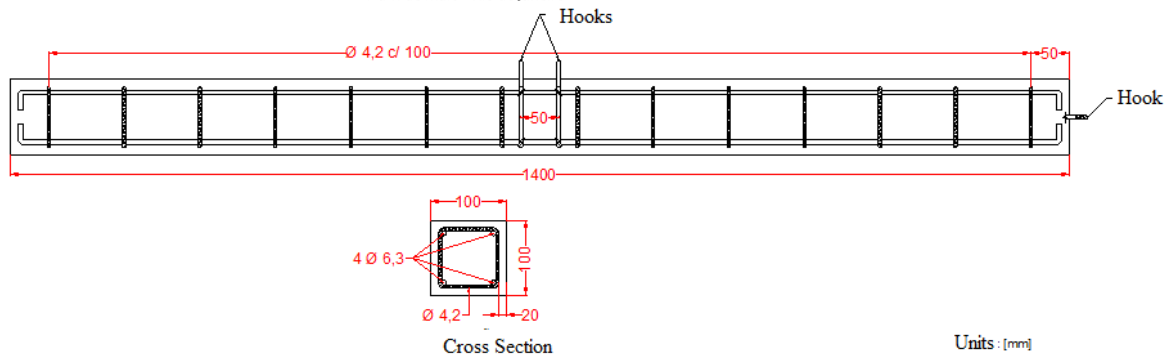


Figure 2 - RC Beam project

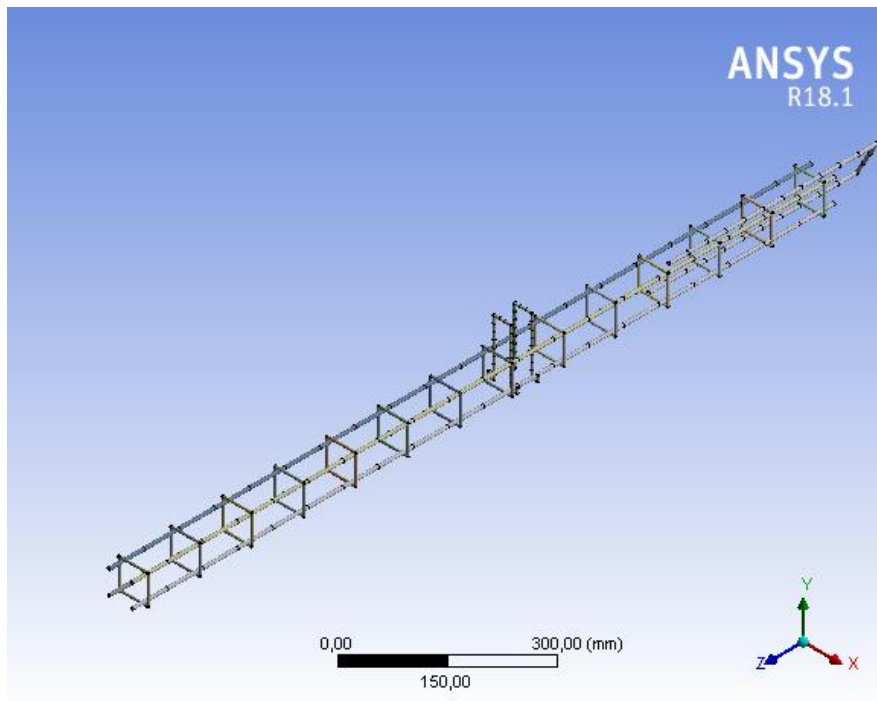


Figure 3 – Meshed steel bars in ANSYS model



Figure 4 - Experimental testing of the beam

Conclusions

The experimental tests executed in laboratory acquired the values of the three first natural vibration frequencies of the beam, measured through the beam's receptance shown in Figure 5. The different colors represent four different accelerometers that were installed on the experimental model. It is possible to verify that the first frequency of vibration of the beam is about 200 Hz, defined by the first peak in the Receptance curves. The second peak indicates a second natural vibration frequency on 540 Hz, and the third, on 810 Hz.

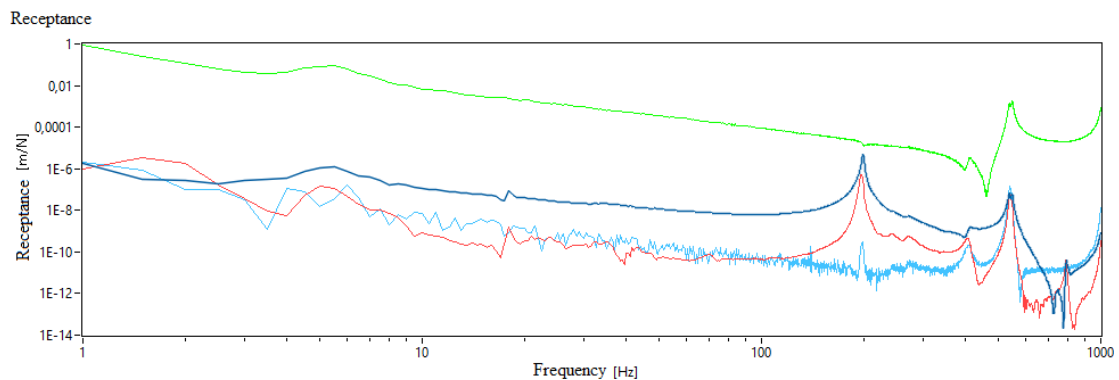


Figure 5 - Receptance of the Reinforced Concrete Beam

Figure 6 shows the first mode shape of the RC beam, which is a flexural mode. Figure 7 shows the second mode shape of the beam and is also a flexural one. On the other hand, Figure 8 shows the third mode shape, but this one is a torsional mode.

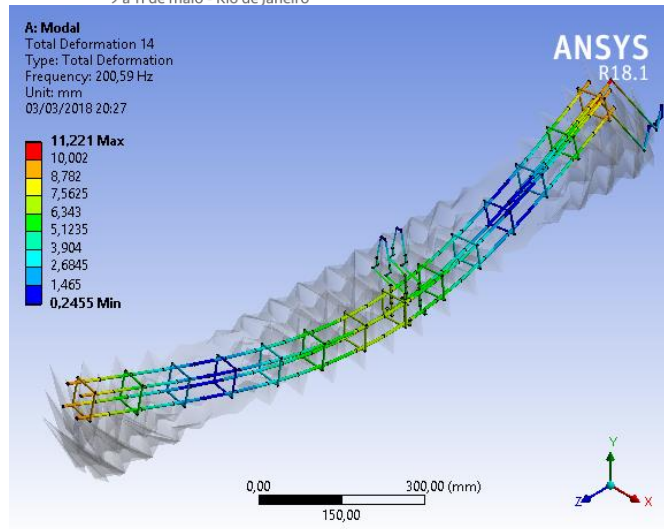


Figure 6 - First mode shape of the beam under free vibration

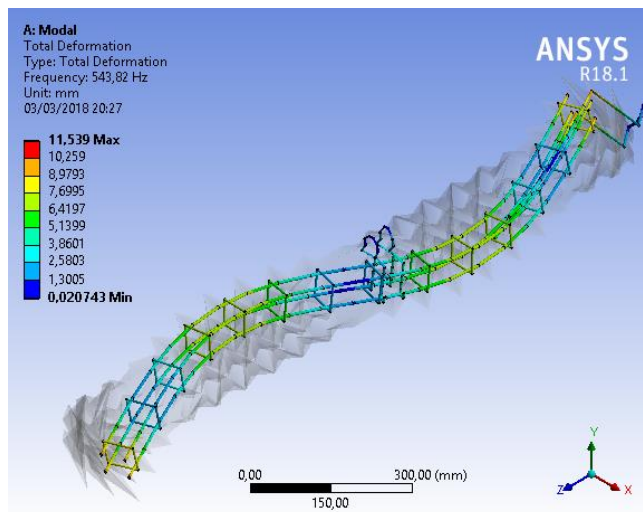


Figure 7- Second mode shape of the beam under free vibration

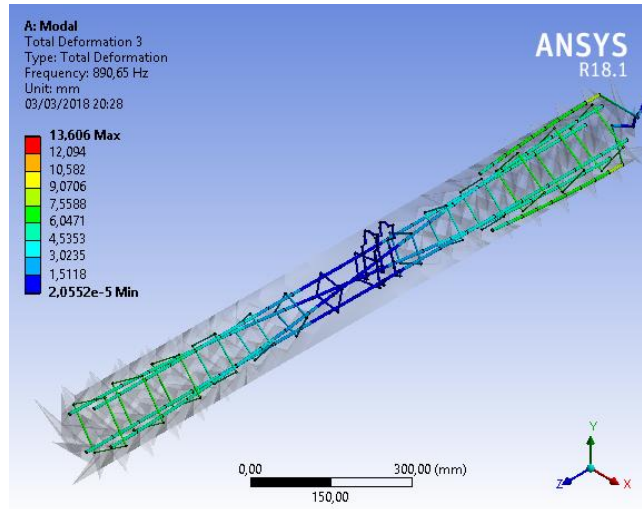


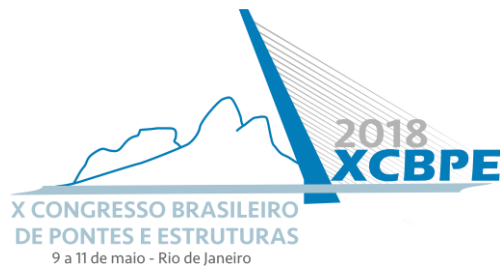
Figure 8- Third mode shape of the beam under free vibration

Table 8 presents the values obtained from the experimental and ANSYS models and compares the data, showing an error that increases with each mode, which is perfectly acceptable considering that ANSYS creates Finite Element models. The next step to be taken shortly is to compare the results obtained in this model with the analytical solutions presented in the previous sections of this paper.

After analyzing the frequencies obtained from the experimental and computational models, it is possible to claim that the Finite Element has a good accuracy, although it is known that both have their errors and simplifications. One of the following steps is to update the model towards the final desired model, which is a RC pole of distribution, with variable cross-section and elastic support. However, it is important to keep validating the procedures implemented in this current paper by designing new computational models based on non-computational analytical models and comparing the results obtained by each one of them.

Table 8- Experimental and ANSYS natural vibration frequencies of the beam

Mode	Frequency		Error
	Dynamic Test	ANSYS	
1	200	200.59	0.30%
2	540	543.82	0.70%
3	810	890.65	9.88%



Acknowledgments

The authors would like to thank LACTEC Intitutes, COSERN and Neoenergy group for the opportunity for developing this work within the research project PeD entitled as - "Evaluation of the structural integrity of distribution poles by means of dynamic monitoring".

Referências

- Clough, R. W., & Penzien, J. (2003). Dynamics of Structures. *Dynamics of Structures*, 752. <https://doi.org/10.1002/9781118599792>
- J.L.Humar. (2001). Dynamics of Structures. *A A Balkema Publishers*.
- Meirovitch, L. (2001). Meirovitch Leonard Fundamentals of Vibrations.
- Zong, Z., Lin, X., & Niu, J. (2015). Finite element model validation of bridge based on structural health monitoring—Part I: Response surface-based finite element model updating. *Journal of Traffic and Transportation Engineering (English Edition)*, 2(4), 258–278. <https://doi.org/10.1016/j.jtte.2015.06.001>
- ANSYS Workbench for Windows, version 18.1: Finite Element software.