MODAL ANALYSIS OF ALFE POWER LINES USING NEW 3D FGM BEAM FINITE ELEMENT

Roman Gogola, Justín Murín, Juraj Hrabovský, Juraj Paulech, Vladimír Kutiš, Vladimír Goga

Institute of Automotive Mechatronics, Faculty of Electrical Engineering and Information Technology, Slovak University of Technology in Bratislava, Ilkovičova 3, 812 19 Bratislava, Slovak Republic

E-mail: roman.gogola@stuba.sk

Received 05 May 2015; accepted 14 May 2015

1. Introduction

Power lines under certain conditions are exposed to dynamic loads in addition to static ones, which can cause elastic vibrations. If a frequency of the harmonic dynamic loads is equal to the eigenfrequency of the line, the resonance vibrations can arise, which can result in mechanical damage of the power line. Therefore, the modal analysis, by which the eigenfrequencies and eigenmodes are stated, is needed. The power line is from mechanical point of view a 3D system. It can vibrate in longitudinal, horizontal, vertical direction and the torsional vibrations are possible as well. But in the technical calculations it is mostly simplified to the one dimensional system. In the literature, e.g. [1], an analytical method is used for the power line free vibration in its vertical plane. The analytical methods are not much effective for the general modal analysis. Most effective are the numerical methods, over all the finite element method. For dynamic analysis the beam finite element can be used. Because the material of the power line is inhomogeneous, the homogenization of material properties is needed.

For modeling and simulation of the problem a new 3D composite beam finite element is used, which was developed at our institute [2]. The second order beam theory has been used for the finite element stiffness and mass matrix formulation. In this paper, the tensional, flexural and torsional eigenmodes of chosen power lines are analysed.

2. Modeling of the AlFe power line

The symmetric power line marked as 450AlFe6 has been considered. This power line consists of 3+9 steel and 11+17 aluminium wires (see Fig. 1).

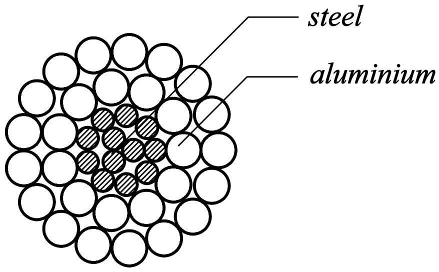

Fig. 1. *Heterogeneous cross-section of the used AlFe power line*

The diameter of the aluminium wires is d_{Al} = 4.5 mm and the diameter of the steel wires is d_{Fe} = 2.8 mm. The effective cross-sections of the power line parts are: A_{Fe} = 73.89 mm^2,

A_{Al} = 445.32 mm^2 and the effective cross-sectional area of the whole power line is A = 519.21 mm^2. The span is L = 300 m with differing attachment levels (see Fig. 2). For bundled conductors there are used spacer dampers with cross-section area A_{SD} = 225 mm^2.

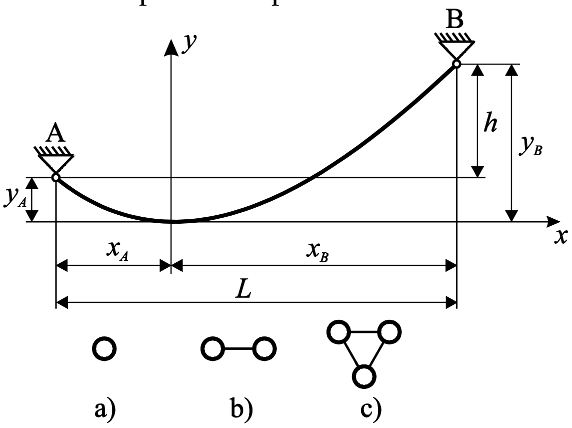

Fig. 2. *Span with differing attachment levels – a) single power line, b) double bundle power line, c) triple bundle power line*

Material properties for power line are: steel – Young's modulus E_{Fe} = 210 GPa, Poisson's ratio ν_{Fe} = 0.28, material density ρ_{Fe} = 7850 kg.m^{-3}; aluminium – Young's modulus E_{Al} = 70 GPa, Poisson's ratio ν_{Al} = 0.32, material density ρ_{Al} = 2700 kg.m^{-3}. Material properties for spacer dampers are: Young's modulus E_{SD} = 75 GPa, Poisson's ratio ν_{SD} = 0.33, material density ρ_{SD} = 2730 kg.m^{-3}.

To create a FEM model for modal analysis it is important to calculate the place of maximal deflection and maximal length of the power lines after extension in static state. At first the gravity load q [N.m^{-1}] of the power line is calculated using formula [1], [3], [4]:

$$q = m.g \tag{1}$$

where m [kg.m^{-1}] is the nominal weight of the power line, g [m.s^{-2}] is the standard gravity. The catenary parameter [1], [3], [4] is:

$$c = \frac{F_H}{q} = \frac{\sigma_H}{\gamma.z} \tag{2}$$

where F_H [N] is the horizontal force at the point of maximal deflection of power line, σ_H [Pa] is the horizontal stress in power line, γ [N.m^{-3}] is the unit weight per cross-section and z [-] is the weather load factor (eq. icing, wind or combination of these loads). For this case we do not consider with these loads so z = 1. The distances of maximal deflection from the fixing point A x_a [m] and from the fixing point B x_b [m] are calculated using formulas [1], [4]:

$$x_a = \frac{L}{2} - c.\frac{h}{L} \tag{3}$$

$$x_b = \frac{L}{2} + c.\frac{h}{L} \tag{4}$$

where L [m] is the length of the span, h [m] is the height difference of the attachment points A and B. The maximal deflection with respect to fixing point B y_b [m] of the power line is [1], [4]:

$$y_b = c.\left(\cosh\left[\frac{x_b}{c}\right] - 1\right) \tag{5}$$

and the power line length after extension L_D [m] is [1], [4]:

$$L_D = c.\left(\sinh\left[\frac{x_a}{c}\right] + \sinh\left[\frac{x_b}{c}\right]\right) \tag{6}$$

The axial forces N_A^{II} [N] and N_B^{II} [N] in the power line at the fixing points A and B are:

$$N_A^{II} = c.\left(\cosh\left[\frac{x_a}{c}\right].q\right),\ N_B^{II} = c.\left(\cosh\left[\frac{x_b}{c}\right].q\right) \tag{7}$$

3. Finite element equations of the 3D composite beam finite element and their solution

The local beam finite element equation for modal analysis has the following formal form:

$$\left[K - \omega^2 M\right] \cdot [U] = [0] \tag{8}$$

where K is the stiffness matrix, M is the mass matrix, ω is the eigenfrequency, and U is the nodal points displacement vector. The stiffness matrix contains also the axial force, which in the case of power line is the tensional force caused by the own weight. The detailed description of this equation can be found in our previous publications, *eg* in [2]. The global finite element equations of the beam structures can be generally established by a classical method. In modal analysis the eigenvalue problem is solved. For a given axial force N, geometrical parameters, material and boundary conditions, the natural frequency ω will be increased until the determinant of the matrix of the global finite element equations tends to zero. The natural frequency is the natural eigenfrequency from which the eigenfrequency can be calculated. Again, the eigenforms can be calculated. The derived equations were implemented into the computational program in the environment of the code MATHEMATICA [5]. With this program following modal analysis of the single and bundle power lines were done. Comparative calculation of the same tasks were also done with software ANSYS [6].

4. Numerical experiments

In modal analysis the single and double bundle power lines according to Fig. 2 has been considered. The effective quadratic moments of inertia of the power line's cross-section area are: $I_z = I_y = 28528.3$ mm^4. The effective circular cross-section of power line is constant with diameter $d_{ef} = 25.71$ mm, the deformed length of the power line is $L_D = 300.307$ m and the height difference is $h = 10$ m. The axial force is $N^{II} = 49700$ N (for simplicity, a constant maximal value for the whole line length was assumed).
The effective material properties of the used power line are [7]:

$E_L^{NH} = 89923.76\ \text{MPa}$, $E_L^{M_y H} = E_L^{M_z H} = 97146.55\ \text{MPa}$, $G_{L_y}^{H} = G_{L_z}^{H} = 34586.06\ \text{MPa}$,
$G_L^{M_x H}(x) = 28090.28\ \text{MPa}$, $\rho_L^{NH} = 3432.91\ \text{kg.m}^{-3}$, $\rho_L^{M_x H} = 2811.64\ \text{kg.m}^{-3}$

Calculated effective material properties have been used in the modal analyses of the single and double-bundle power lines. The eigenfrequencies f [Hz] of the single power lines have been found with a mesh 800 of BEAM188 elements of the commercial software ANSYS, modal analyses of double-bundle power lines have been done with mesh 1455 of

BEAM188 elements. The same problem was solved using the new 3D beam finite element (3D NFE) for modal analysis of composite beam structures [2] (the calculation has been done in software Mathematica). The first ten calculated eigenfrequencies of single and double bundle power lines are shown in Tab. 1.

Tab. 1. *Eigenfrequencies of the single and double-bundle power line in the xy and xz planes.*

eigenfrequencies f [Hz]		single power line			eigenfrequencies f [Hz]		double-bundle power line		
		3D NFE	ANSYS	Δ [%]			3D NFE	ANSYS	Δ [%]
1st	xz	0,2777	0,2775	0,0684	1st	xz	0,2777	0,2778	0,0036
2nd	xy	0,3781	0,3850	-1,8276	2nd	xy	0,3781	0,3848	-1,7694
3rd	xy	0,5550	0,5545	0,0919	3rd	xy	0,4033	0,4090	-1,4308
4th	xz	0,5554	0,5584	-0,5402	4th	xy	0,5545	0,5548	-0,0649
5th	xz	0,8331	0,8323	0,0948	5th	xz	0,5551	0,5555	-0,0703
6th	xy	0,8379	0,8377	0,0179	6th	torsional	0,6346	0,6493	-2,3150
7th	xy	1,1087	1,1097	-0,0721	7th	xz	0,8327	0,8332	-0,0673
8th	xz	1,1094	1,1099	-0,0451	8th	xy	0,8370	0,8380	-0,1243
9th	xz	1,3885	1,3873	0,0871	9th	torsional	0,9563	0,9845	-2,9520
10th	xy	1,3894	1,3884	0,0713	10th	xy	1,1102	1,1103	-0,0027

The 6th and 9th eigenmodes of single power line are shown in Fig. 3, the 6th and 9th eigenmodes of double bundle power line are shown in Fig. 4.

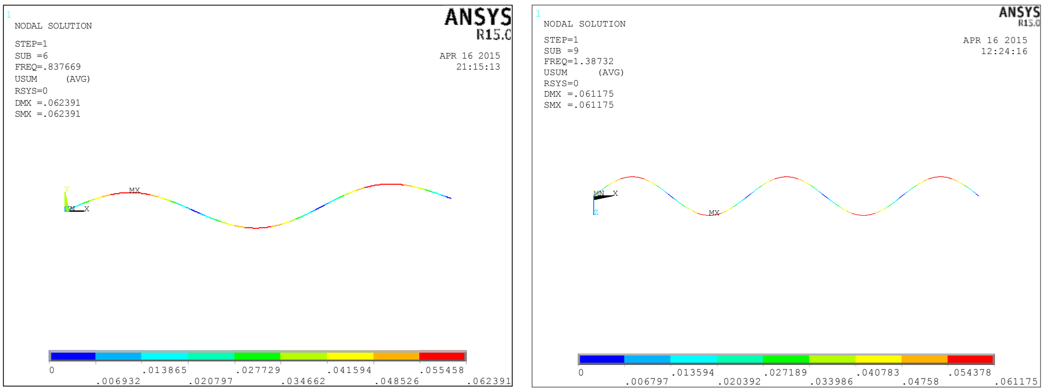

Fig. 3. *The 6th and 9th eigenmode of single power line*

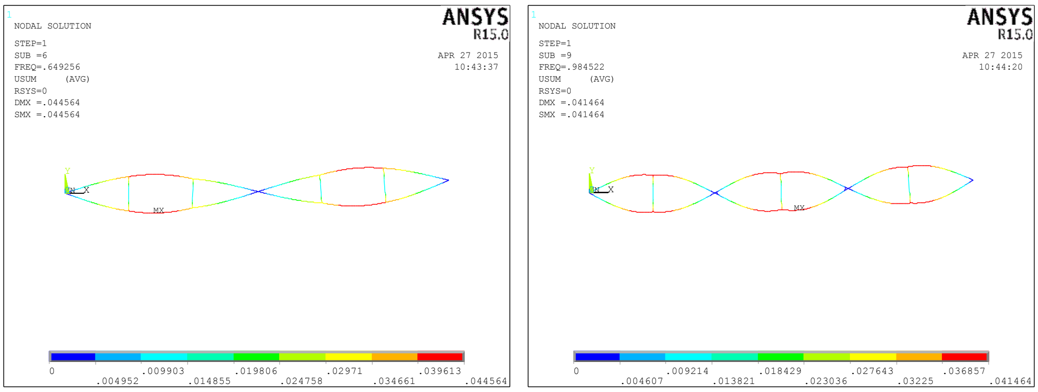

Fig. 4. *The 6th and 9th eigenmode of double bundle power line*

5. Conclusion

In this paper, the results of modal analysis of the single and double bundle AlFe power lines with differing attachment levels are presented. For numerical simulation our new beam finite element and the commercial FEM software ANSYS were used. The axial, flexural and torsional eigenfrequencies and eigenmodes have been studied. The first ten eigenmodes for single power line are flexural and it can be observed from Fig. 3. In opposite, the first ten eigenmodes for double bundle power lines are flexural and torsional. It can be observed from Fig. 4. The calculations confirmed very good accurancy and effectiveness of our new beam finite element. The largest deviation between the NFE and ANSYS solution is occured by the torsional eigenfrequencies. In our new finite element is the effective torsional shear modulus and torsional mass density calculated by the RVE method but in ANSYS solution, the shear elasticity modulus is calculated from the effective elasticity modulus for bending, what is a simplification, which produces low accuracy of the calculation results.

Acknowledgement

This paper has been supported by Grant Agency APVV–0246-12, Grant Agency VEGA No. 1/0228/14 and Agency VEGA No. 1/0453/15.

This contribution is the result of the project implementation: Industrial Research Centre for operating lifetime of selected components of power plants (ITMS: 26240220081), supported by the Research & Development Operational Programme funded by the ERDF.

References:

[1] Š. Fecko, D. Reváková, L. Varga, J. Lago, S. Ilenin, Vonkajšie elektrické vedenia, Bratislava: Renesans, s.r.o., 2010.

[2] J. Murín, M. Aminbaghai, J. Hrabovský, V. Kutš, J. Paulech, S. Kugler, „A new 3D FGM beam finite element for modal analysis," rev. *11th WCCM*, Barcelona, Spain, 2014.

[3] J. Murín, M. Kalousek, V. Kutiš, Mechanika a Termomechanika: vybrané kapitoly pre elektrotechnikov, Bratislava: STU Bratislava, 2006.

[4] L. Varga, „Vonkajšie vedenia," rev. *Vonkajšie vedenia - Návody na cvičenia*, zv. ISBN: 80 2270 28 8, Košice, ES VŠT Košice, 1989, pp. 17 - 49.

[5] S. Wolfram Mathematica 5, Wolfram research, Inc., 2003.

[6] ANSYS Swanson Analysis System, Inc., 201 Johnson Road, Houston, PA 15342/1300, USA.

[7] J. Murin, V. Kutis, Improved mixture rules for the composite (FGM's) sandwich beam finite element., Barcelona, Spain, 2007, pp. 647-650.