58th conference on experimental stress analysis
EAN 2020

Equivalence of Boundary Conditions of FEM Model

HUŇADY R.[1,a], LENGVARSKÝ P.[1,b], KAĽAVSKÝ A.[1,c]

[1]Deparment of Applied Mechanics and Mechanical Engineering, Faculty of Mechanical Engineering, Technical University of Košice, Letná 9/B, 042 00 Košice, Slovak Republic

[a]robert.hunady@tuke.sk, [b]pavol.lengvarsky@tuke.sk, [c]adam.kalavsky@tuke.sk

Keywords: equivalent model, boundary conditions, FEM, modal analysis, stiffness

Abstract. The paper deals with two methods of equivalence of boundary conditions in the finite element model. The proposed methods are based on the determination of the stiffness parameters in the section plane, in which a removed part of the model is replaced by a constraint with elastic elements or bushing connector. In the first case, the stiffness parameters are determined by a series of static finite element (FE) analyzes that are used to obtain the response of the removed part to the six basic types of loading. The second method is a combination of experimental and numerical approach. The natural frequencies obtained by the measurement are used in FE optimization, in which the response of the model is tuned by changing the stiffness parameters of the bushing. Both methods provide a good estimate of the stiffness at the point where the model is replaced by an equivalent boundary condition. This increases the accuracy of the numerical model and at the same time saves computational capacity and the required time due to the reduction of elements.

Introduction

The correctness of the numerical solution results in the finite element method is largely dependent on the accuracy of the FEM model itself. The basis is the correct definition of material properties, initial conditions, loads and boundary conditions such as constraints, connections and contacts in accordance with the real ones. Inaccuracies in the model definition lead to deviations or errors in the load response. Many dynamic problems, such as impact tests [1], [2], fatigue analyzes [3], [4], are quite sensitive to boundary conditions and material parameters. The most common causes of differences in model behavior may be constraint stiffness and structural damping [5], [6]. Computational models are usually verified by correlating the results of numerical calculation and experimental measurement. A suitable approach is to compare modal parameters, i.e. natural frequencies, mode shapes, e.g. using the MAC criterion [7], [8], [9]. In order to refine the model, it is then possible to perform a sensitivity analysis, where the response is adjusted by changing the input parameters.

The paper deals with procedures aimed at improving boundary conditions. Conventional boundary conditions are replaced by equivalent ones, which provide a significant simplification of the model while maintaining their accuracy. Depending on the application, their accuracy may even be increased in some cases. Examples are constraints that are considered to be perfectly rigid in the FEM model, while in point of fact they are flexible. Perfectly rigid constraints distort both the static and dynamic response of the model to external loads. This can lead to a considerable influence on the results and their inconsistency with the results of experimental tests such as vibration analysis, fatigue analysis, etc. For this reason, an equivalence method has been proposed to replace the conventional boundary condition or some part of the model. The equivalent boundary condition is represented by a bushing connector whose stiffness can be determined either by a series of numerical static analyses or experimentally by modal analysis in combination with FE optimization. The similar approach has been used to determine the material constants of a homogenized material [10], [11].

The first equivalence method

The principle of the first method is explained on the example of a beam whose one end is fixed to the frame by means of two preloaded bolts (Fig. 1a). This part of the model will be replaced by bushing whose stiffness parameters are determined in FE static analysis, based on acting of three forces and three moments (Fig. 1b), respectively. Fig. 1c shows the equivalent model of the beam where the part of the model representing the beam attachment is replaced by bushing connector connected to ground. The stiffness parameters were calculated using the following formulas and their values are given in Table 1

$$k_{x,y,z} = \frac{F_{x,y,z}}{\delta_{x,y,z}} \quad \text{(N/mm)}, \tag{1}$$

where k_x, k_y, k_z are equivalent translational stiffness in x-, y-, z- direction, respectively,
δ_x, δ_y, δ_z are maximum translations in x-, y-, z- direction, respectively,
F_x, F_y, F_z are loading forces acting in x-, y-, z- direction, respectively,

and

$$k_{\varphi_x,\varphi_y,\varphi_z} = \frac{M_{x,y,z}}{\varphi_{x,y,z}} \quad \text{(N.mm/rad)}, \tag{2}$$

where k_{φ_x}, k_{φ_y}, k_{φ_z} are equivalent rotational stiffness about x-, y-, z- axis, respectively,
φ_x, φ_y, φ_z are maximum angular deflection about x-, y-, z- axis, respectively,
M_x, M_y, M_z are loading moments acting about x-, y-, z- axis, respectively.

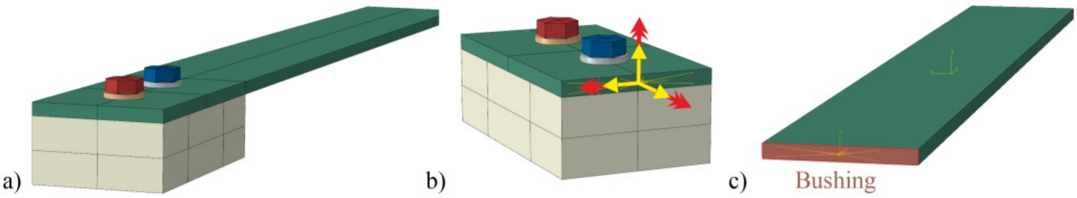
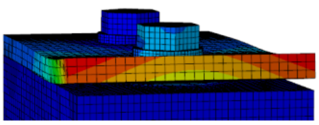
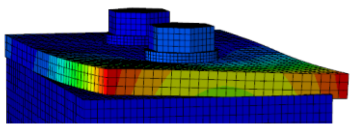
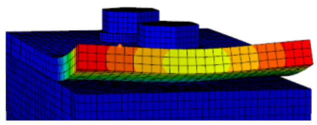
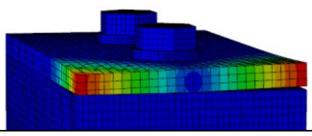
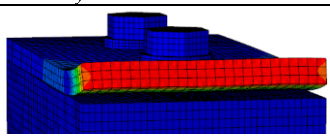
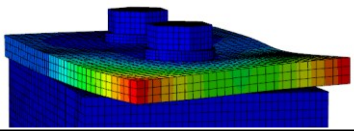


Fig. 1: a) The complete initial FEM model, b) Part of the model used to obtain stiffness parameters, c) The equivalent model with bushing connector.

Table 1: Stiffness parameters of the bushing.

		
k_x = 397e3 N/mm	k_y = 101e3 N/mm	k_z = 9.8e3 N/mm
		
$k_{\varphi x}$ = 1.547e6 N.mm/rad	$k_{\varphi y}$ = 0.972e6 N.mm/rad	$k_{\varphi z}$ = 45.455e6 N.mm/rad

Both FE models, the complete initial model (Fig. 1a) and the equivalent model (Fig. 1c), were subjected to modal analysis to compare them. Their natural frequencies are listed in Table 2. Two selected mode shapes can be seen in Fig. 2

Table 2: Natural frequencies of compared FE models in Hz.

Initial model	Equivalent model	Initial model	Equivalent model
37.6	37.7	1975.4	2058.6
235.5	239.7	2129.2	2194.7
332.6	339.4	2624.3	2610.5
511.2	505.4	3173.7	3256.5
658.7	676.6	3765.5	3736.1
1289.7	1329.7	4111.3	4107.4
1547.5	1533.3	4424.3	4494.1

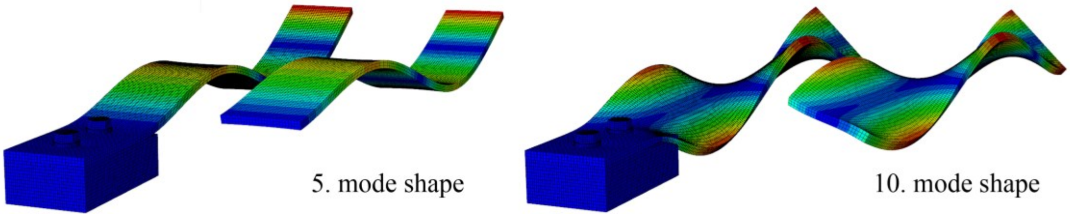


Fig. 2: Selected mode shapes of compared models.

The results obtained show very good agreement, which means that the equivalent model response corresponds to that of the complete initial model. In addition to maintaining dynamic behavior, the model has been significantly simplified. The number of elements decreased from 59747 to 15148.

The second equivalence method

This method uses the values of natural frequencies determined by experimental modal analysis (EMA) to calculate the stiffness parameters. The parameters are determined in the optimization process, which is based on frequency FEM analysis, where the objective function is the known natural frequencies. By changing the stiffness of the bushing, the model is retuned, i.e. to change its modal parameters. The shape, dimensions and material properties of the structure must be known.

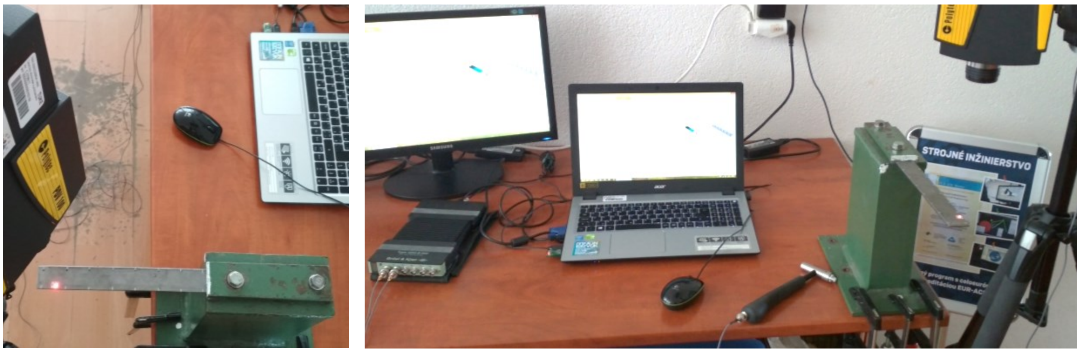


Fig. 3: Experimental modal analysis of the beam.

The method is presented on a similar beam model as used in the previous case. The rectangular steel beam was fixed to the rigid base on one side by means of two bolt connections (Fig. 3). Its natural frequencies and mode shapes were determined by a modal test, in which the beam was excited with a Bruel & Kjaer 8206 impact hammer and its responses were measured with a Polytec PDV100 laser vibrometer. The measurement and the evaluation was performed using a Bruel & Kjaer Pulse system. In the frequency range up to 3.2 kHz, 7 modes were identified, of which 5 were bending and 2 torsion mode shapes of vibration (Fig. 4). Their natural damped frequencies and damping ratios are listed in Table 3.

Table 3: Modal parameters of the beam.

Mode	Damped frequency (Hz)	Damping ratio (%)	Mode shape
1	46.95	2.02	1. bending
2	289.44	0.33	2. bending
3	660.99	0.21	1. torsion
4	810.21	0.17	3. bending
5	1567.15	0.08	4. bending
6	1998.37	0.08	2. torsion
7	2562.31	0.27	5. bending

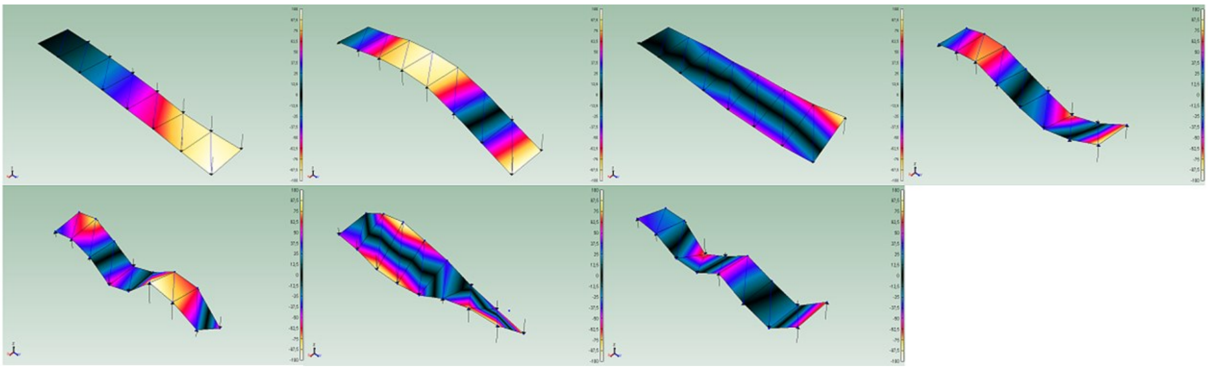


Fig. 4: Mode shapes of the beam.

The parametric optimization was performed in NX Nastran software that uses gradient-based numerical optimization algorithm [12]. The FE model (Fig. 5) consists only of the free part of the beam. The removed part representing the beam attachment was replaced by bushing elements connecting the section plane of the beam to the ground.

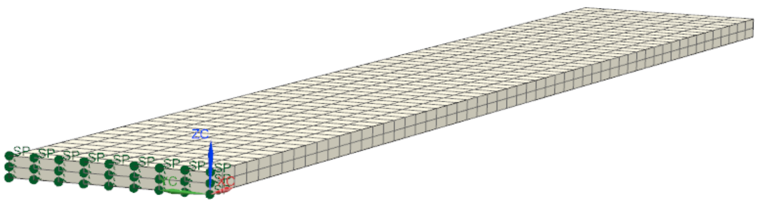


Fig. 5: FE model of the beam.

Since NX allows only one objective function to be defined, this function was the frequency of the first mode. Other frequencies were defined as optimization constraints with a tolerance of +/- 5%. It should be noted that the lateral mode shapes of vibration were not identified by the modal test. However, they occur in FEM analysis (mode 3 and mode 8). For this reason, these modes have not been included among the optimization constraints. The parameters k_x, k_y, k_z representing the equivalent translational stiffness of the bushing in x-, y-, z- direction, respectively, were consider as the design variables.

The optimization parameters were as follows:

```
Design Objective:
Target Model Frequency Mode 1, Target value = 47.000 Hz

Design Constraints:
Model Frequency, Mode 1, Upper limit = 49.000 Hz
Model Frequency, Mode 1, Lower limit = 45.000 Hz
Model Frequency, Mode 2, Upper limit = 304.00 Hz
Model Frequency, Mode 2, Lower limit = 275.00 Hz
Model Frequency, Mode 4, Upper limit = 694.00 Hz
Model Frequency, Mode 4, Lower limit = 628.00 Hz
Model Frequency, Mode 5, Upper limit = 851.00 Hz
Model Frequency, Mode 5, Lower limit = 769.00 Hz
Model Frequency, Mode 6, Upper limit = 1645.0 Hz
Model Frequency, Mode 6, Lower limit = 1488.0 Hz
Model Frequency, Mode 7, Upper limit = 2098.0 Hz
Model Frequency, Mode 7, Lower limit = 1898.0 Hz
Model Frequency, Mode 9, Upper limit = 2690.0 Hz
Model Frequency, Mode 9, Lower limit = 2434.0 Hz

Design Variables:
kx (N/mm), Initial value = 6e6, Lower limit = 500, Upper limit = 10e6
ky (N/mm), Initial value = 6e6, Lower limit = 500, Upper limit = 10e6
kz (N/mm), Initial value = 6e6, Lower limit = 500, Upper limit = 10e6
```

Fig. 6 shows how the value of the objective function has changed in the individual optimization cycles. Fig. 7 shows the changes of the design variables.

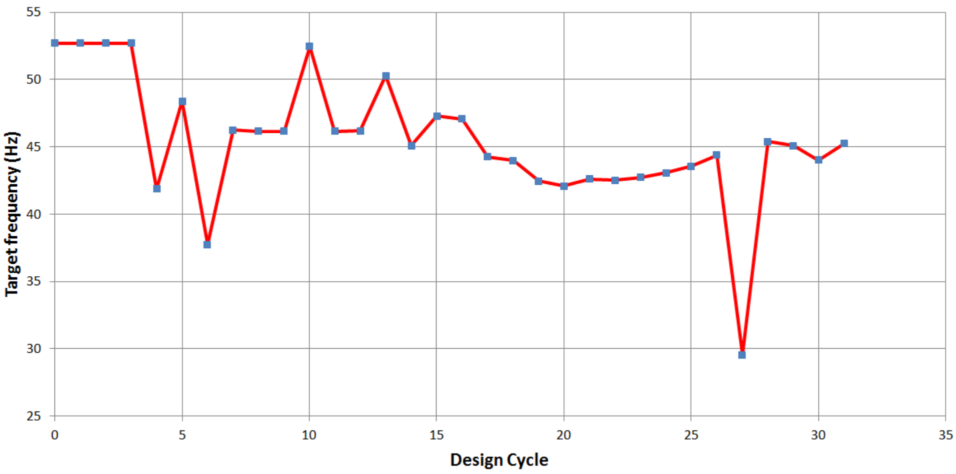


Fig. 6: Objective function plot.

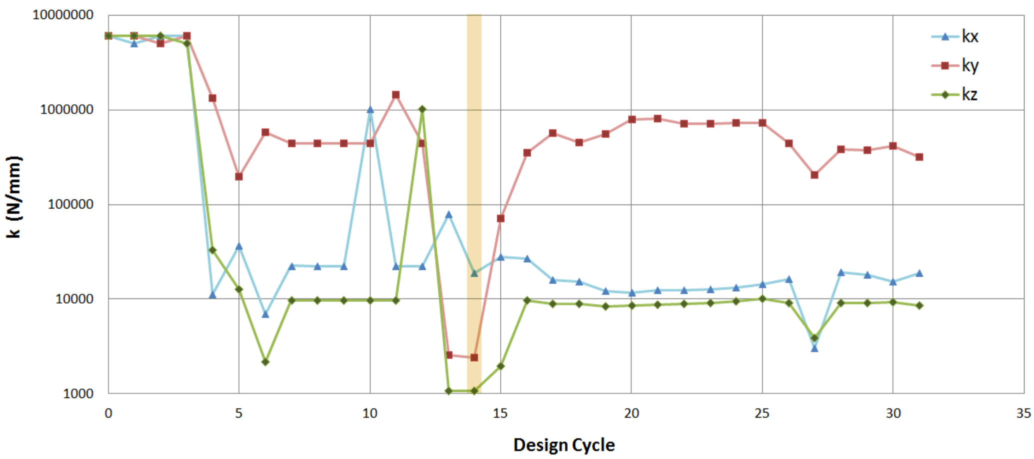


Fig. 7: Design variables plot.

Since not all optimization constraints were met in any cycle, it was necessary to choose a solution where the estimate was closest to meeting them. In the 14. design cycle, five of the seven conditions were met, so this cycle is therefore considered to be the result of optimization. Stiffness parameters in this cycle are k_x = 18654.71 N/mm, k_y = 2428.06 N/mm, k_z = 1068.86 N/mm. For comparison, Table 4 lists the natural frequencies of the beam model with bushing with these parameters, the frequencies of the clamped-free beam, and the frequencies obtained by measurement. The experimentally determined frequencies are considered as reference. Percentage deviations are calculated with respect to them.

Table 4: Natural frequencies of the beam obtained by EMA and FEM.

Mode	EMA	FEM Beam with bushing	FEM Clamped-free beam
1	46.95	45.09 (–3.97%)	52.743 (12,34%)
2	289.44	291.07 (0.56%)	330.11 (14,05%)
3	-	411.26	516.60
4	660.99	682.50 (3.25%)	724.09 (9,55%)
5	810.21	829.92 (2.43%)	924.37 (14,09%)
6	1567.15	1682.42 (7.35%)	1812.71 (15,67%)
7	1998.37	2064.50 (3.31%)	2192.14 (9,69%)
8	-	2463.03	2992.52
9	2562.31	2741.52 (6.99%)	2998.71 (17,03%)

It is evident that the percentage differences are significantly smaller in the case of the model with bushing and increase the accuracy of the FE model compared to the clamped beam with a perfectly rigid constraint. Therefore, it can be stated that the estimation of the stiffness parameters is correct. A more accurate estimate could be obtained by multicriteria optimization.

Conclusions

In the paper, the methods to increase the accuracy of the boundary conditions of the FE model are described. Their basis is the replacement of the boundary condition, bond or part of the model by bushing or elastic connectors with adequate stiffness. The paper presents two approaches to determine these parameters. The first is based on a purely numerical calculation and is applicable when a real physical model does not exist. This approach is very accurate and effective. The second approach is based on experimental modal analysis and parametric optimization using FEM. Its accuracy depends on several factors, such as natural frequencies, material properties, dimensions and geometry of the structure, on the basis of which the FE model is created. From this point of view, it is an estimation method. However, the test results indicate that the estimate is relatively accurate and the procedure is practically applicable.

Acknowledges

This research was supported by project VEGA 1/0355/18.

References

[1] V. Ferreira, L. P. Santos, M. Franzen, et al., Improving FEM crash simulation accuracy through local thickness estimation based on CAD data, Advances in Engineering Software, Volume 71, 2014, Pages 52-62, ISSN 0965-9978.

[2] S. Soni, S. K. Pradhan, Improving crash worthiness and dynamic performance of frontal plastic automotive body components, Materials Today: Proceedings, 2019, ISSN 2214-7853.

[3] A. Kaľavský, R. Huňady, Fatigue analysis of a notched beam in frequency domain, AIP Conference Proceedings 2198, 020007 (2019).

[4] A. Kaľavský, R. Huňady, M. Kicko, M. Kučinský, Calculation of the Fatigue Life of a Dynamically Loaded Component, American Journal of Mechanical Engineering, 2017, Vol. 5, No. 6, 252-256.

[5] M. Goletti, V. Mussi, A. Rossi, M. Monno, Procedures for Damping Properties Determination in Metal Foams to Improve FEM Modeling, Procedia Materials Science, Volume 4, 2014, Pages 233-238, ISSN 2211-8128.

[6] F. S. Iglesias, A. F. López, Rayleigh damping parameters estimation using hammer impact tests, Mechanical Systems and Signal Processing, Volume 135, 2020, ISSN 0888-3270.

[7] D. Siano, M. Viscardi, R. Aiello, Sensitivity Analysis and Correlation Experimental/Numerical FEM-BEM for Noise Reduction Assessment of an Engine Beauty Cover, Energy Procedia, Volume 81, 2015, Pages 742-754, ISSN 1876-6102.

[8] M. Pastor, M. Binda, T. Harčarik, Modal Assurance Criterion, Procedia Engineering, Volume 48, 2012, Pages 543-548, ISSN 1877-7058.

[9] M. Brehm, V. Zabel, Ch. Bucher, An automatic mode pairing strategy using an enhanced modal assurance criterion based on modal strain energies, Journal of Sound and Vibration, Volume 329, Issue 25, 2010, Pages 5375-5392, ISSN 0022-460X.

[10] R. Huňady, P. Pavelka, P. Lengvarský, Homogenization of Material Properties of a Combined rubber/steel Sheet using Modal Analysis, In: EAN 2019 - Conference Proceedings, 2019, p. 146-150.

[11] A. Kaľavský, R. Huňady, P. Pavelka, Determining the young's modulus of a specimen using experimental modal analysis. In: EAN 2018 - Conference Proceedings, 2018, p. 176-83.

[12] Design Sensitivity and Optimization User's Guide (NX Nastran help), Siemens, URL: docs.plm.automation.siemens.com/data_services/resources/nxnastran/10/help/en_US/tdocExt/pdf/optimization.pdf