NUMERIČKA ANALIZA SLOBODNIH OSCILACIJA GREDA

NUMERICAL ANALYSIS OF FREE OSCILLATIONS OF BEAMS

Ena Šahinović [1]
Safet Isić [1]
Ermin Husak [2]
Samir Dizdar [1]
Emir Nezirić [1]

[1] University "Džemal Bijedić" of Mostar
[2] University of Bihać, Faculty of Technical Engineering

Ključne riječi: numeričke metode, metod konačnih elemenata, gredni nosač, slobodne oscilacije, Ansys.

Keywords: numerical methods, finite element method, beam, free oscillations, Ansys.

Paper received:
25. 12. 2024.
Paper accepted:
27. 12. 2024.

Stručni rad

REZIME
Rad analizira slobodne oscilacije grednih nosača, koje su od ključne važnosti za sprečavanje rezonancije u konstrukcijama poput mostova, zgrada i drugih inženjerskih objekata. Cilj rada je analizirati tačnost numeričkih metoda, poređenjem njihovih rezultata s egzaktnim rješenjima za tri slučaja oslanjanja: prosta greda, konzola i greda uklještena na oba kraja. Egzaktna rješenja su prikazana za sve slučajeve. Numerički rezultati za prva tri moda oscilacija dobiveni su u programu ANSYS, nakon čega su upoređeni s egzaktnim vrijednostima.

Professional paper

SUMMARY
This paper examines the free oscillations of beams, which are essential for preventing resonance in constructions such as bridges, buildings, and other engineering structures. The objective of the study is to evaluate the accuracy of numerical methods by comparing their results with exact solutions for three support conditions: a simply supported beam, a cantilever beam, and a beam fixed at both ends. Exact solutions were presented for all cases. Numerical results for the first three oscillation modes were obtained using ANSYS software and compared to the exact values.

1. INTRODUCTION

1.1. Oscillations of Structures

Beam structures are key components in engineering constructions such as bridges, buildings, and machines, serving to transfer loads. Basic oscillations analysis helps predict the behavior of structures under dynamic loads and avoids resonance, which is crucial for safety and durability. Resonance occurs when the natural frequency of a structure matches the frequency of an external force, potentially causing severe failures, as in the case of the Tacoma Narrows Bridge collapse in 1940.

1.2. Methods of Oscillations Analysis

Oscillations analysis can be analytical (exact), numerical, or experimental:

- **Exact analysis** uses differential equations to determine natural frequencies and oscillations modes. For example, the Euler-Bernoulli equation describes the free vibrations of beam structures.
- **Numerical analysis,** such as the finite element method (FEM), discretizes the structure into a finite number of elements to approximately solve mathematical models.
- **Experimental analysis** involves real measurements on physical models to compare numerical and analytical results.

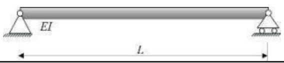
1.3. Free Vibrations

Free vibrations occur when a structure vibrates without external forces acting on it, and their characteristics depend on the material, geometry, and boundary conditions. The oscillation frequency indicates the number of oscillations per second, while the period represents the time of one cycle. Angular frequency shows the rate of phase change in oscillations, and amplitude is the maximum deviation from the initial position.

2. EXACT ANALYSIS OF A BEAM FREE VIBRATIONS

In order to compare the results obtained from numerical analysis, an exact analysis is performed for three characteristic cases of beam supports. The cases to be analyzed include: a simply supported beam at both ends, a cantilever beam, and a beam clamped at both ends.

Table 1. *Formulas for natural frequencies of a beam for three characteristic cases.*

Support conditions	Natural frequency f [Hz]
Simply supported beam 	$f_n = \frac{n^2\pi}{2L^2}\sqrt{\frac{EI}{m}}$
Cantilever beam 	$f_n = \frac{\beta_n^2}{2\pi}\sqrt{\frac{EI}{m}}$ $\beta_n L = \frac{2(n-1)\pi}{2};n \geq 3$
Fixed-fixed beam 	$f_n = \frac{\beta_n^2}{2\pi}\sqrt{\frac{EI}{m}}$ $\beta_n L = \frac{(2n+1)\pi}{2}$

The exact analysis for each of these cases involves solving the differential equation that describes the free oscillations of the beam, applying the appropriate boundary conditions to determine natural frequencies and oscillation modes. Natural frequencies were calculated using the appropriate formula for each support condition**.**

3. FINITE ELEMENT METHOD

3.1. Concept of Finite Element

A finite element, which forms the fundamental concept of the method, can be represented by different geometric shapes (triangle, quadrilateral etc.). There are 1D, 2D, and 3D finite elements, with the simplest being 1D elements, such as beams with two nodes and linear interpolation. These elements interact with each other through common values at their boundaries (nodes).

3.2. Beam Finite Element

When discretizing a beam, the first step is to divide the beam into finite elements, indexing the elements and nodes. The beam is divided into finite elements that are rigidly connected at the nodes. Concentrated forces and bending moments act at the nodes (including forces representing the support). Figure 1. shows the arrangement of finite elements for beam oscillations analysis. A beam-type finite element is an element with two nodes (i, j). Beam elements can be used for 2D planar analysis, as well as 3D spatial analysis for axial load, bending, and torsion.

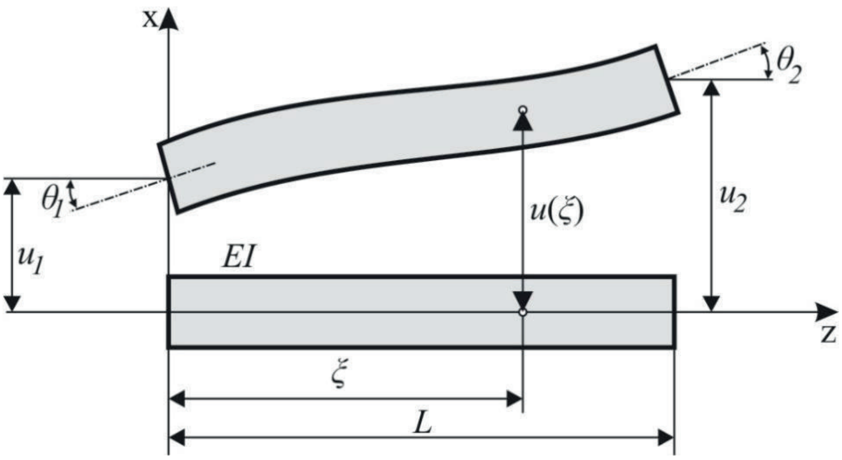


Figure 1. *Linear finite elements for beam bending analysis.*

For slender beams, the bending of the elastic line is described using interpolation with a third-order polynomial, which satisfies continuity and boundary conditions between the elements. Displacements within the element are determined by the following expression:

$$u(\xi) = [\mathbf{N}]\{\boldsymbol{\delta}\}^e \quad \text{... (1)}$$

where [**N**] is the shape matrix, and $\{\delta\}^e$ is the element nodal displacement vector [1-4].

4. EQUATION OF FREE OSCILLATIONS

To determine the natural frequencies and modes for the case of free oscillations using the FEM, the matrix equation of motion for a discretized structure with *n* degrees of freedom is used:

$$[\mathbf{M}]\{\ddot{\boldsymbol{\delta}}\}+[\mathbf{C}]\{\dot{\boldsymbol{\delta}}\}+[\mathbf{K}]\{\boldsymbol{\delta}\}=0 \qquad \dots (2)$$

where are:
[**M**] - mass matrix
[**C**] - damping matrix
[**K**] - stiffness matrix
$\{\ddot{\boldsymbol{\delta}}\}$, $\{\dot{\boldsymbol{\delta}}\}$, and $\{\boldsymbol{\delta}\}$ - vectors of generalized accelerations, velocities, and displacements [6-7].

For a beam finite element with a constant cross-sectional area, the stiffness matrix has the following form [4-5]:

$$[\mathbf{k}]^e=\frac{2EI}{L^3}\begin{bmatrix} 6 & 3L & -6 & 3L \\ 3L & 2L^2 & -3L & L^2 \\ -6 & -3L & 6 & -3L \\ 3L & L^2 & -3L & 2L^2 \end{bmatrix}. \qquad \dots (3)$$

For a beam finite element with a constant cross-sectional area, the mass matrix is:

$$[\mathbf{m}]^e=\frac{\rho AL}{420}\begin{bmatrix} 156 & 22L & 54 & -13L \\ 22L & 4L^2 & 13L & -3L^2 \\ 54 & 13L & 156 & -22L \\ -13L & -3L^2 & -22L & 4L^2 \end{bmatrix}. \qquad \dots (4)$$

The damping matrix is neglected for free oscillations, because it does not significantly affect natural frequencies and mode shapes.

5. ANALYSIS OF BEAM FREE OSCILLATIONS IN ANSYS

The analysis of free oscillations in ANSYS is performed in the Modal Analysis module within ANSYS Mechanical. This module enables the calculation of natural frequencies and corresponding modes. Line elements were used to model the beam and analyze free oscillations. The beam was also modeled in the ANSYS program. Boundary conditions were applied, and results were presented for each of the three cases of free oscillations. Material specifications and geometric parameters of the beam are identical for all three cases:

- Modulus of elasticity (*E*): 210 GPa
- Width (*b*): 0.1 m
- Height (*h*): 0.05 m
- Beam length (*L*): 1.2 m
- Material density (ρ): 8750 kg/m³
- Beam mass (*m*): 47.1 kg
- Moment of inertia (*I*): $1.041\cdot10^{-6}$ m^4
- Cross-sectional Area (A): 0.005 m²

5.1. Example 1: A Simply Supported Beam

The geometry of the beam is defined by modeling in ANSYS program (Figure 2.). The linear model was created, and the specified cross-section was assigned. The beam geometry is the same for all considered characteristic cases. The example of finite element mesh generated for considered beam is shown in Figure 3.

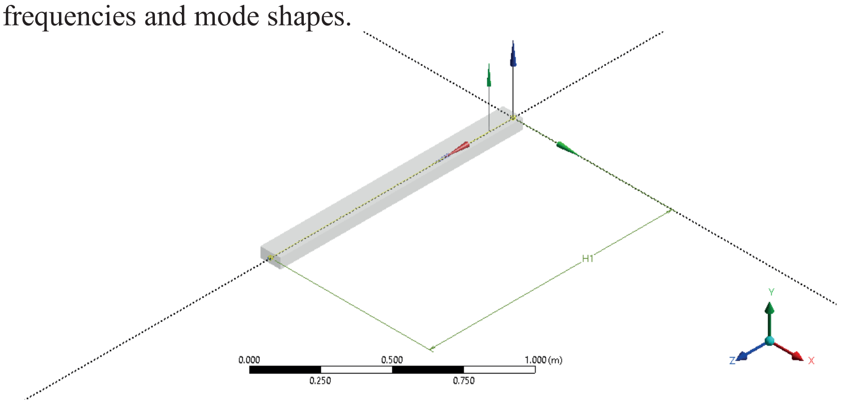


Figure 2. Geometry of the beam

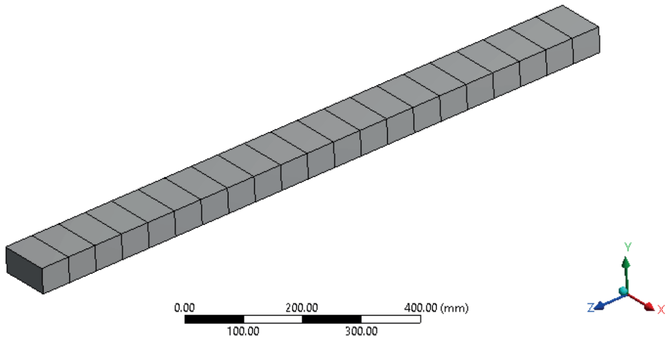


Figure 3. An example of generated finite element mesh

After defining the boundary conditions, the simulation is set up to determine the first three bending oscillation modes by simultaneously increasing number of finite elements. The first three modes of the simply supported beam are shown in Fig. 4. Table 1. shows the error values between the exact frequency calculations and the frequencies numerically obtained in ANSYS.

Table 1. Results for the first three natural frequencies of a simply supported beam

Mode number	f_{exact} [Hz]	f_{numeric} [Hz]	Error [%]
1	81.40	81.22	0.22
2	325.62	322.55	0.93
3	732.65	717.76	1.99

It can be observed that the error between the exact values of the first and second frequencies is less than 1%, while for the third frequency, the error exceeds 2%.
The error for the first and second modes is less than 1%, while the error for the third mode is 1,99%. This demonstrates the high accuracy of ANSYS compared to the exact values. For the model with two finite elements, the error in the calculation of the third frequency is 1.8, while for twenty finite elements, it is 0.98. As the number of elements increases, the results converge toward the exact values, improving the accuracy of the analysis. Figure 5. shows the first three frequencies for different number of finite elements.

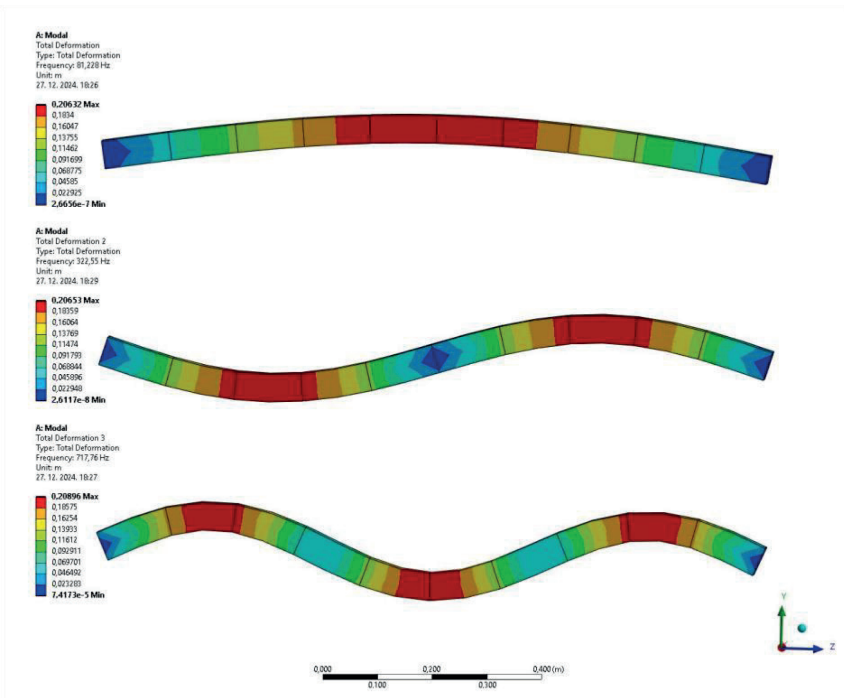


Figure 4. *The first three oscillations modes of simply supported*

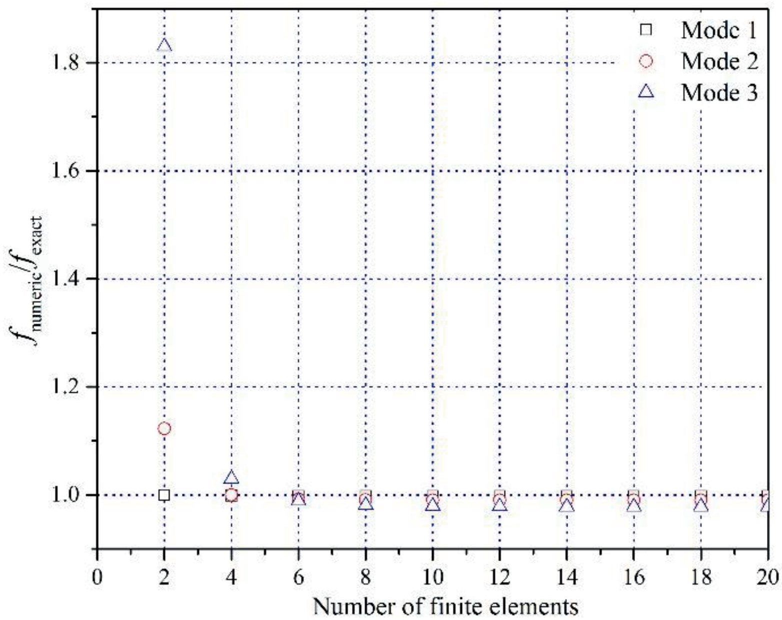


Figure 5. *Convergence of natural frequencies by increasing of finite element number beam*

5.2. Example 2: A Cantilever Beam

The next case is a cantilever beam with one fixed end and one free end. The geometry and mesh are identical to those in the previous example.

The only difference in analysis is in applied boundary condition. Figure 6. shows the first three oscillation modes.

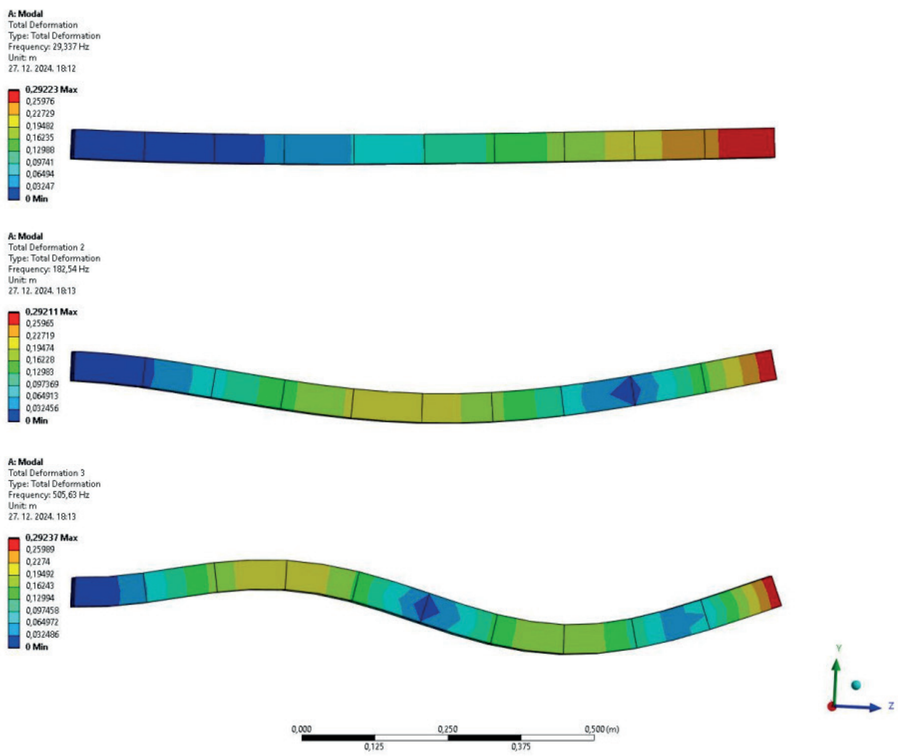


Figure 6. *The first three oscillations modes of cantilever beam*

The frequency calculation error values are presented in Table 2.

Table 2. Results for the first three natural frequencies of the cantilever beam

Mode number	f_{exact} [Hz]	$f_{numeric}$ [Hz]	Error [%]
1	28.97	29.33	1.24
2	181.71	182.54	0.44
3	508.68	505.63	0.59

It is noticeable that, when comparing the exact frequency values with the frequency values obtained from the numerical simulation in ANSYS, the error for the second and third frequencies is below 1%, while for the first frequency, the error exceeds 1%.

Figure 7. shows the change of first three frequencies by increasing the number of finite elements for a cantilever beam.

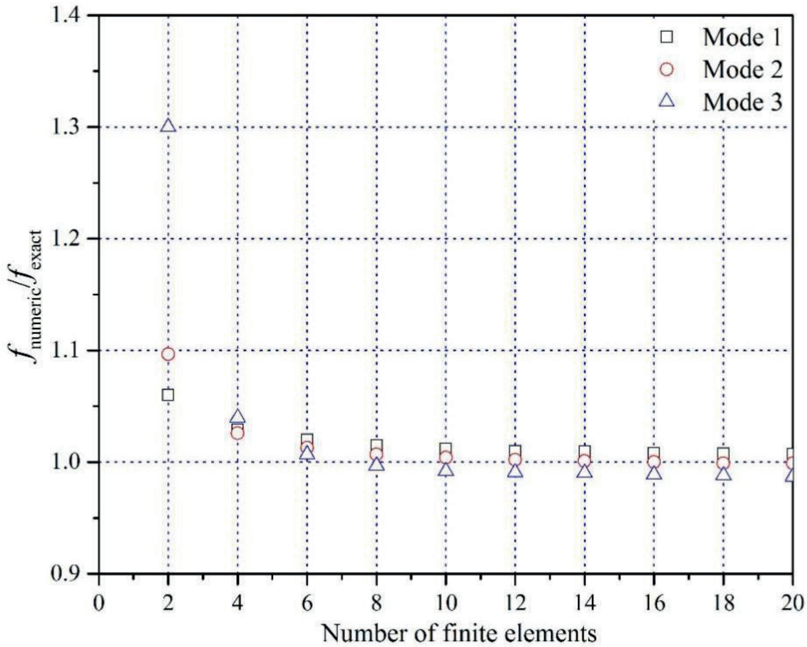


Figure 7. *Convergence of first three eigenfrequencies of cantilever beam by increasing of finite element number*

5.3. Example 3: A Fixed – Fixed Beam

The third analyzed characteristic case is a beam fixed at both ends. After applying boundary condition on the beam model same as in the previous examples, natural frequencies are calculated for different finite elements mesh. The frequency calculation error values are presented in Table 3. Based on Table 3., it can be concluded that the error for the first frequency exceeds 2%, for the third frequency 1,36%, while for the second frequency, the error is significantly smaller, amounting to only 0.34%. Figure 8 shows the first three vibration modes for the beam, obtained from the performed analysis.

Table 3. Results for the first three natural frequencies of the fixed-fixed beam

Mode number	f_{exact} [Hz]	$f_{numeric}$ [Hz]	Error [%]
1	183.06	187.33	2.27
2	508.01	509.77	0.34
3	997.23	983.84	1.36

Figure 9. shows the convergence of the first three natural frequencies by increasing number of finite elements. The results confirm that finer discretization leads to more accurate calculations. These results highlight the importance of increasing the number of elements to achieve higher accuracy in numerical oscillations analysis.

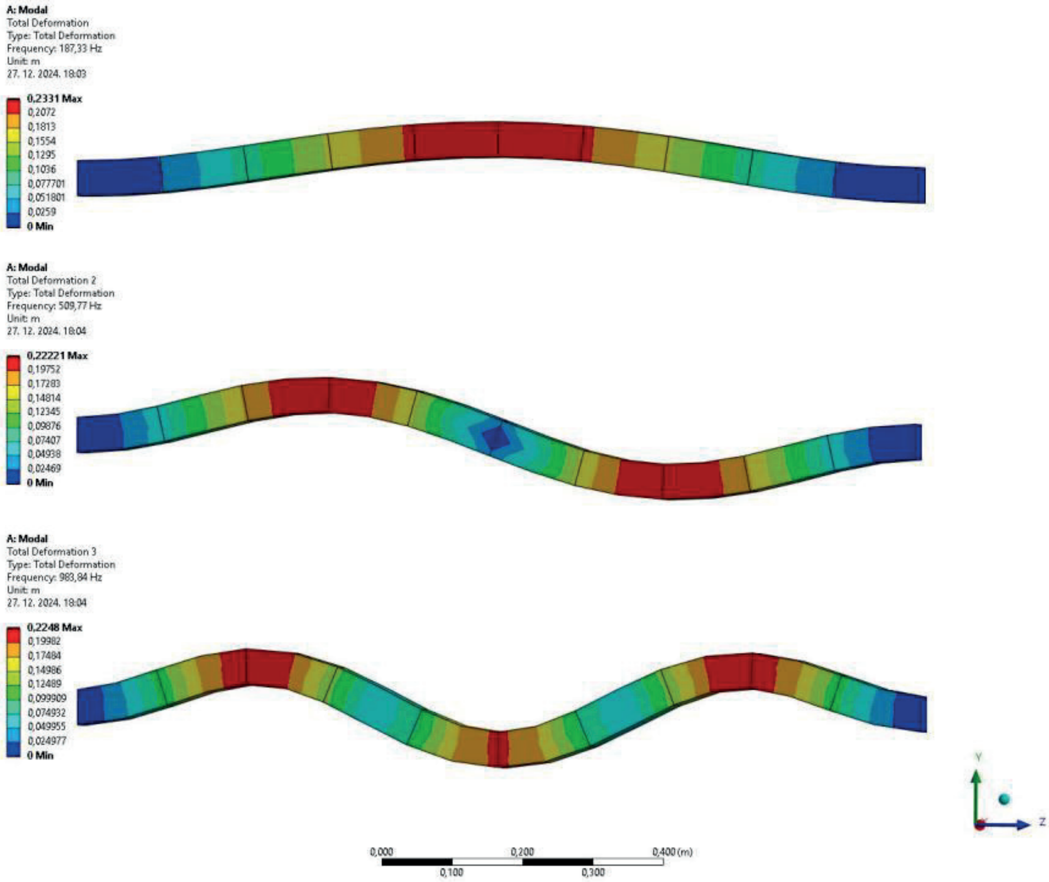


Figure 8. *The first three oscillations modes of fixed-fixed beam*

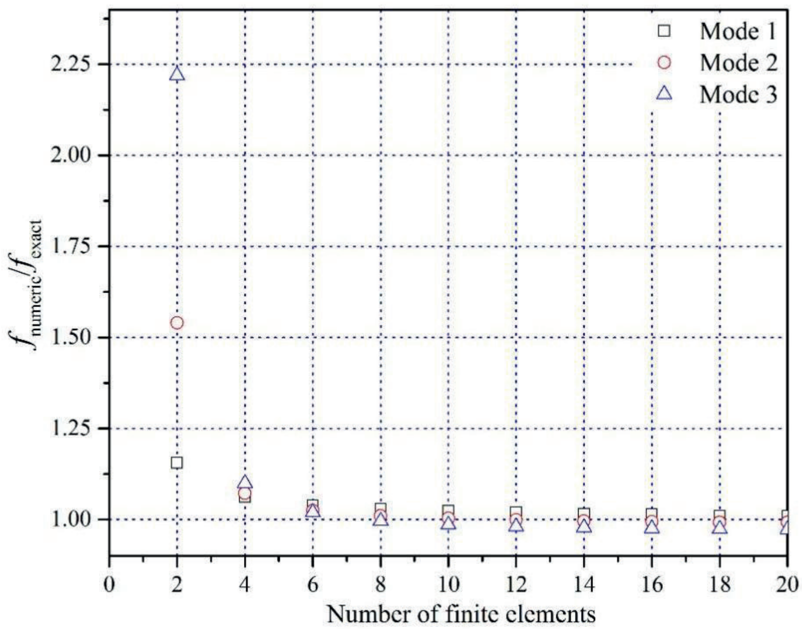


Figure 9. *Convergence of first three eigenfrequencies of fixed-fixed beam by increasing of finite element number*

6. CONCUSION

This paper addressed the topic 'Numerical Analysis of Free Oscillations of Beams' using exact and numerical methods. For numerical analysis finite element method was used in ANSYS software. Three different beam support conditions were analyzed: a simply supported beam, a cantilever beam, and a beam clamped at both ends. The numerical results obtained in ANSYS were compared to exact solutions, with the error not exceeding 3%. The number of finite elements varied in the range from 2 to 20 (2, 4, 6, ... 20).
The analysis of the results showed that increasing the number of finite elements significantly improves the accuracy of the obtained vibration frequencies, confirming the importance of selecting the appropriate mesh density for accurate numerical calculations.

7. REFERENCES

[1] Lemeš, S. "Vibrations of Centrifugal Pumps in Cooling Systems of Automobile Engines." Chapter 5. *Application of the Finite Element Method for Determining Natural Frequencies of Free Undamped Vibrations*. Master's thesis, Zenica, 2002.

[2] Makarios, Triantafyllos K. "Identification of Mode-Shapes and Eigen-Frequencies of Bi-Hinge Beam with Distributed Mass and Stiffness." Aristotle University of Thessaloniki, 2020.

[3] Rana, Hemanta Kumar. "Dynamic Analysis of Fixed-Fixed Beams." Bachelor's thesis, 2012.

[4] Pičuga, A. *Introduction to the Finite Element Method*. Sarajevo: Svjetlost, 1985.

[5] Isić, S. "The Impact of Axial Load on Structural Vibrations Using the Finite Element Method." Master's thesis, 2000.

[6] Craiga, Roya R. Jr. *Structural Dynamics*. 2017.

[7] Zaimović-Uzunović, N., S. Lemeš. *The Finite Element Method*. Zenica: Dom Štampe, 2002.

Corresponding author:
Ena Šahinović
University "Džemal Bijedić" in Mostar
Faculty of Mechanical Engineering
Email: ena.sahinovic00@gmail.com
Phone: + 387 60 34 56 31

The research that has produced this paper has been financially supported by the European Union through the project Introducing Student Research Mobilities to BH Universities (INSTREAM), implemented by a consortium led by International Burch University, but do not necessarily represent the official position of the European Union or International Burch University and is the sole responsibility of the author.