Analysis of a Cantilevered Beam Using FEMCAS

Ştefan Cristian Castravete[2](✉) and Gabriel Cătălin Marinescu[1,2]

[1] University of Craiova, A.I. Cuza, 13, 200585 Craiova, Romania
[2] Fluid Struct, Bvd Ştirbei Vodă. 19A, 200376 Craiova, Romania
scastravete@caelynx.ro

Abstract. This paper presents the analysis of a cantilevered beam using both the commercial FEA software Abaqus and FEMCAS a new software developed for structural analysis. The theoretical approach is described. The results from FEMCAS were comparable with theoretical calculation as well as with Abaqus which shows good accuracy of the solver.

Keywords: Vibration · Modal Analysis · FEA · FEM · Simulation

1 Introduction

FEMCAS is a software based on Finite Element Method which can analyze structures under static and dynamic loads. The software is suitable for creating robust design of structure in 3-dimensional space and is based on Euler–Bernoulli beam formulation. The software can consider defects in the structure based on statistical formulations which leads to a more robust design.

The software includes the following modules:

– DETERMINISTIC LINEAR ANALYSIS MODULE

- Modal Analysis
- Dynamic Analysis

– LINEAR ANALYSIS MODULE WITH PARAMETRIC UNCETAINTIES

- Monte-Carlo Analysis
- Perturbation Analysis
- Neumann Expansion Analysis

This paper will concentrate on the deterministic linear analysis module. The linear analysis module with parametric uncertainties will be presented in a subsequent paper.

I. Dumitru et al. (Eds.): ICOME 2022, AHE 15, pp. 183–189, 2023.
https://doi.org/10.2991/978-94-6463-152-4_21

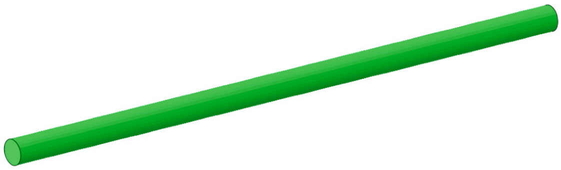

Fig. 1. The geometric model

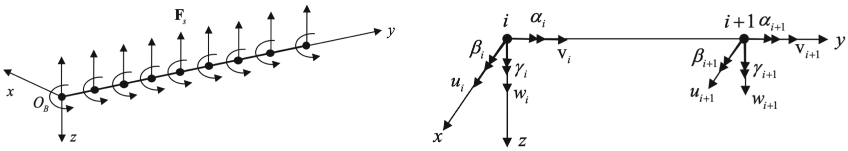

Fig. 2. (a) Finite Element Discretization. **(b)** One Finite Element

2 Theoretical Considerations

2.1 Finite Element Method

Consider a Euler–Bernoulli beam. The governing equations of motion are obtained using the Lagrange equation.

Beam (see Fig. 1) is split into elements like in the Fig. 2 (a) where one element is represented in Fig. 2(b) [1].

Element displacements, $\tilde{u}(y, t)$, $\tilde{\text{v}}(y, t)$, and $\tilde{w}(y, t)$ are expressed in terms of nodal displacements u_i, v_i, w_i in directions x, y, z and nodal rotations β_i, α_i, γ_i around axes x, y, and z.

Equations for one element can be written in matrix form:

$$\mathbf{M}^e \ddot{\boldsymbol{U}}^e + \mathbf{K}^e \mathbf{U}^e = \mathbf{F}^e$$

Assembling elements, we obtain:

$$[\boldsymbol{M}]\{\ddot{\boldsymbol{U}}\} + [\boldsymbol{K}]\{\boldsymbol{U}\} = \{\boldsymbol{F}\}$$

where, $[\boldsymbol{M}]$ is mass matrix, $[\boldsymbol{K}]$ is stiffness matrix, $\{\boldsymbol{F}\}$ exterior applied force vector, $\{\boldsymbol{U}\}$ is displacement vector, and $\{\ddot{\boldsymbol{U}}\}$ is the acceleration vector.

2.2 Modal Analysis

Modal analysis finds the natural frequencies of the system which describe free vibration of a system. Natural frequencies are giving the resonant frequencies of the system and it is very important indicative of dynamic behavior.

Removing external loads, the equations of the system become:

$$[\boldsymbol{M}]\{\ddot{\boldsymbol{U}}\} + [\boldsymbol{K}]\{\boldsymbol{U}\} = 0$$

Considering solution as harmonic vibration:

$$\{\boldsymbol{U}\} = \{\boldsymbol{X}\}sin(\omega t)$$

where, $\{\boldsymbol{X}\}$ is displacement amplitude, ω is rotational frequency, t is time, the equations of motion become:

$$[\boldsymbol{K}]\{\boldsymbol{X}\} = \omega^2[\boldsymbol{M}]\{\boldsymbol{X}\}$$

Before solution is necessary to reduce to standard form [2]

$$[\boldsymbol{A}]\{\boldsymbol{Z}\} = \omega^2\{\boldsymbol{Z}\}$$

where $[\boldsymbol{A}]$ is symmetric, $[\boldsymbol{Z}]$ represents the eigenvector and ω^2 the eigenvalue.

There are several algorithms to solve the equations. Among them, Inverse Iteration Method, Jacobi, Lanczos, or Arnoldi are most used. FEMCAS implements the Jacobi algorithm for modal analysis [2].

2.3 Dynamic Analysis

This module completes the deterministic linear analysis module and analyze linear systems under dynamic loads. This program can be used also for static analysis. The static analysis is a quasi-static where the time is applied slow over the amplitude to reduce the kinetic energy. In quasi-static analysis the kinetic energy should be minimized.

Consider equation of motion in matrix form:

$$[\boldsymbol{M}]\left\{\ddot{\boldsymbol{U}}\right\} + [\boldsymbol{C}]\left\{\dot{\boldsymbol{U}}\right\} + [\boldsymbol{K}]\{\boldsymbol{U}\} = \{\boldsymbol{F}\}$$

where, $[\boldsymbol{M}]$ is the mass matrix, $[\boldsymbol{C}]$ is the damping matrix, $[\boldsymbol{K}]$ is the stiffness matrix, $\{\boldsymbol{F}\}$ is external applied force vector, $\{\boldsymbol{U}\}$ is the displacement vector, $\left\{\dot{\boldsymbol{U}}\right\}$ is the velocity vector and $\left\{\ddot{\boldsymbol{U}}\right\}$ is the acceleration vector. The system is solved by finite element method and the results are displacement, velocity, and acceleration. The equation of motion includes this time the damping therm.

The program is made in Fortran and calculates the displacement of beam elements type under time dependent external forces [2]. The algorithm is using the Newmark direct method which involves the use of time parameters β and γ. The values of these parameters determine the accuracy and stability of the algorithm. By default, FEMCAS uses $\beta = 0.25$ and $\gamma = 0.5$.

It starts from initial state of the bar model were, according to known material data is calculating the initial mass matrix $[\boldsymbol{M}]_0$, initial damping matrix $[\boldsymbol{C}]_0$ and initial stiffness matrix $[\boldsymbol{K}]_0$. The initial conditions of the system $\{\boldsymbol{U}\}_0$ and $\left\{\dot{\boldsymbol{U}}\right\}_0$ are also known.

3 Simulation

3.1 Modal Analysis

The geometric model in Fig. 1 was analyzed with the commercial FEA program Abaqus and a new FEA program called FEMCAS.

In Abaqus the structure was modelled with hexahedral solid elements then modelled with BEAM elements. In FEMCAS the structure was modelled with beam type elements. The model tested is a cantilevered beam as shown in Fig. 1.

Table 1. Natural Frequencies Comparison

Mode No	Frequency (Hz)			
	FEMCAS	Theoretical	Abaqus Beam Elements	Abaqus FEA 3D solid elements
1	28.90	29.04	28.89	29
2	179.09	181.68	178.64	178
3	496.36	509.54	493.44	494
4	962.31	999.24	952.04	958
5	1295.85	1296.53	1295.90	1299
6	1572.65	1651.63	1546.40	1560

Table 2. FEMCAS deviation

Deviation (%)		
Deviation from theoretical	Deviation from Abaqus Beam	Deviation from Abaqus Solid
0%	0%	-1%
1%	0%	-1%
3%	1%	0%
4%	1%	0%
0%	0%	0%
5%	2%	-1%

3.2 Deterministic Linear Analysis

Modal analysis

First, a modal analysis was performed to extract the natural frequencies of the model. The table below show a comparison of the results (Table 1 and 2).

Forced Vibrations The geometric model in Fig. 1 is subjected to forced vibrations. The beam is grounded at one end and at the other end is subjected to a harmonic excitation force (see Fig. 3) in the vertical direction.

The excitation force has the evolution in Fig. 4.

The analysis was performed in FEMCAS and Abaqus and the result is shown in Fig. 5.

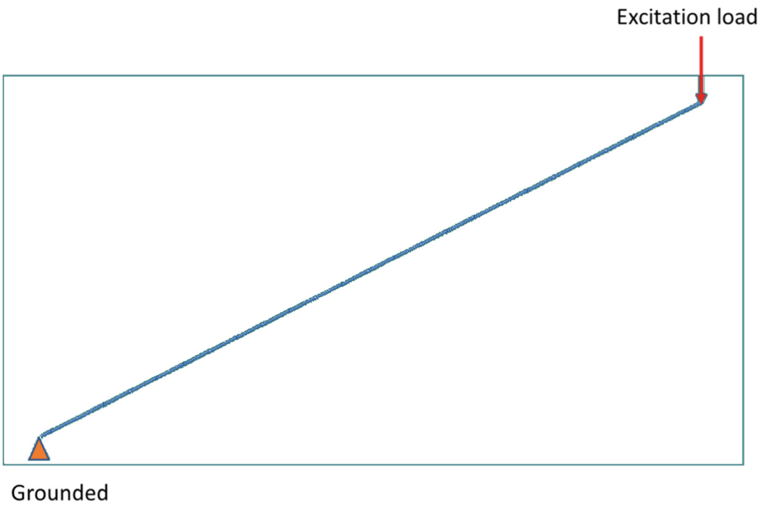

Fig. 3. Boundary conditions and forces

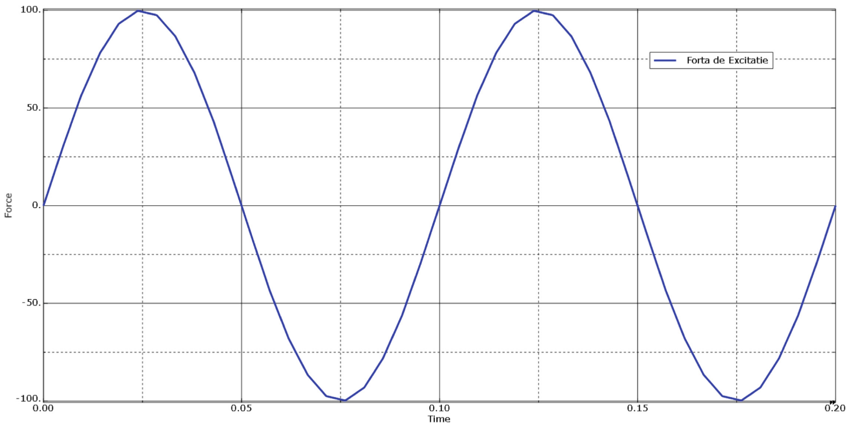

Fig. 4. Excitation force

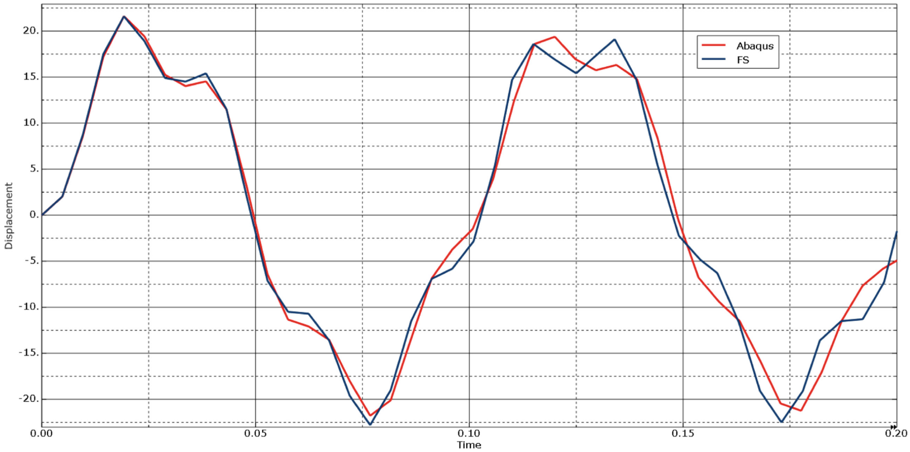

Fig. 5. History of the displacement response in the vertical direction at the beam end where the force is exerted

There can be observed a good correlation between Abaqus and FEMCAS which gives confidence and accuracy to the developed program. The parametric uncertainty linear analysis module will only be tested in FEMCAS as there are no commercial versions of parametric uncertainty linear analysis software on the market.

4 Conclusions

A new structural analysis software is proposed. The software is based on finite element method and can analyze structures in 3-dimensional space, structures that can be approximated by beams of different cross-sections.

A cantilevered beam was analyzed in the absence of parameter uncertainties. In the absence of parametric uncertainties, a modal analysis and forced vibration analysis were performed. The results were compared with analytical calculations and commercial software Abaqus. The results from FEMCAS were comparable with theoretical calculation as well as with Abaqus which shows good accuracy of the solver.

References s

1. Castravete, SC, Nonlinear flutter of a cantilever wing including the influence of structure uncertainties, ProQuest Dissertations Publishing, 3295976 (2007).
2. Margetts, L., Smith, I. M., & Griffiths, D. V. (2014). Programming the Finite Element Method. (5th ed.), John Wiley & Sons Ltd.
3. Simulia Abaqus 2021 documentation, (2021)