Implementation of Finite Element Method in Matlab for assessing modes and natural frequencies of 2D rectangular plates

Augusto Carvalho de Sousa, and Jeferson Rafael Bueno

Citation: Proc. Mtgs. Acoust. **30**, 025005 (2017); doi: 10.1121/2.0000779
View online: https://doi.org/10.1121/2.0000779
View Table of Contents: https://asa.scitation.org/toc/pma/30/1
Published by the Acoustical Society of America

Proceedings of Meetings on Acoustics

Volume 30 http://acousticalsociety.org/

Acoustics `17 Boston

173rd Meeting of Acoustical Society of America and 8th Forum Acusticum
Boston, Massachusetts
25-29 June 2017

Education in Acoustics: Paper 2aEDb3

Implementation of Finite Element Method in Matlab for assessing modes and natural frequencies of 2D rectangular plates

Augusto Carvalho de Sousa
Mechanical Engineering, Katholieke Universiteit Leuven, Leuven, Heverlee, 3001, BELGIUM; augusto.carvalho@kuleuven.be

Jeferson Rafael Bueno
Federal Technology, University of Paraná – UTFPR, Campo Mourão, BRAZIL, Paraná; jrbengenharia@gmail.com

This paper presents results for the assessing of the first six natural frequencies and modal shapes of a 2D rectangular plate with dimensions 200 mm x 500 mm x 2 mm, supposedly a door, with motion restrictions in certain points, supposedly the door ringes and handle. The assessment was made via a self-taught algorithm in MATLAB and the results were compared to COMSOL simulations. The mesh, mass and stiffness matrices, as well as their assembling, is taught through the document. Symbolic language was used to enhance the learning process. A maximum error of 6.02% was found between the two approaches for the 3rd natural frequency. Future research with this approach includes the coupling between the plate and an air cavity and the assessing of the modal shapes and natural frequencies of this system.

Published by the Acoustical Society of America

1. INTRODUCTION

Despite the fact that the engineering students often have a high amount of disciplines and subjects they have to deal with during their degree, most of them do not know how to apply the acquired knowledge in their professional lives.[1] Still, the increase in the number of students in each discipline makes the learning process harder for both students and professors, which brings the need to rethink the "chalk and talk" educational model, where the teacher speaks and the students write, to make this learning process more effective.[2]

The use of softwares might be helpfull to students who are strugling to understand a certain physical concept, but sometimes even though they have a sense of how to get results from a computer simulation, the processes made by the software itself and the mathematical theory behind the results might not be very clear.[3]

When it comes to studies regarding acoustic and vibrations, most simulations are made in commercial softwares available in the market. Such softwares use algorithms to solve the problems given by the user through numerical methods such as the Finite Element Method[4] (FEM) and the Boundary Element Method[5] (BEM). These simulations range from measuring the natural frequencies of a given structure[6] to the analysis of acoustic parameters in rooms.[7] However, these computational tools are very expensive for students to buy, which means that they rely on their universities buying the needed licenses to model physical systems. Furthermore, in the poorer countries, where universities do not have high amount of resources or where the individuals do not have access to the legal procedure of acquiring such softwares, the piracy is the way found to avoid research from stopping.[8–10] Nevertheless, software piracy is bad for both developers and users: for the developers, because it causes money loss and it is not fair with them; and for the pirates, because it is unethical and it might compromise the research, for some features are only present in the legal versions of the computer program.

This paper is the full version of the work previously developed elsewhere[6] and presents a self-taught algorithm to the implementation of the FEM to assess natural frequencies and modal shapes of 2D rectangular plates with different boundary conditions. In addition, the results given by this algorithm are compared with results obtained from a numerical simulation made with a commercial software. The algorithm is an alternative for students to the use of expensive commercial softwares and is written for MATLAB, whose academic version is way cheaper than the ones of such commercial platforms. The paper highlights the main features of the FEM theory and presents parts of the code itself. The code is available for students uppon request by e-mail. People interested in going deeper in the mathematical development of the equations are encouraged to look at ref. [4].

2. FINITE ELEMENT METHOD

The foundations that would lead to the Finite Element Method started to be developed in the 1940's[11,12] and 1950's.[13,14] These ideas were first developed by engineers of aeronautical and civil engineering to solve structural problems.[15]

The method relies in the idea of subdividing a structure in small elements to account for physical quantities such as kinetic and potential energy in each one of these elements. It is then possible to sum everything up to have a final value of these energies, thus leading to the acquisition of the mass and stiffness matrices. From such matrices it is possible to determine the natural frequencies and modal shapes (*eigenfunctions* and *eigenvalues*) of the system.

A typical element for a 2D consideration is shown in Fig. 1. In this figure, the three axis are x, y and z, $2a$ and $2b$ are the dimensions of the element and θ_x and θ_y are the bending moments in their respective axis. The letters ξ and η represent the transposed coordinates x and y to dimensionless or general coordinates, i.e., $\xi = x/a$ and $\eta = y/b$. This transposal of coordinates is useful to deal with the differential equations

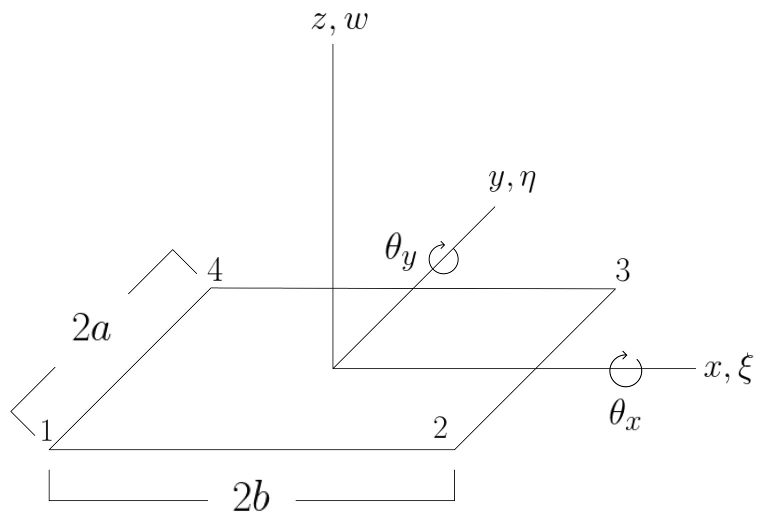

Figure 1: Typical element used in FEM theory (2D).

within the numerical model and is the first step to do before the implementation of the model. The numbers $1, 2, 3$ and 4 are the nodes of the element.

From Fig. 1 it is possible to see that each of the nodes will have three degrees of freedom, i.e., rotation in the x axis, rotation in the y axis, and a transversal motion in the z axis. The three vectors that describe such degrees of freedom are

$$w(\xi, \eta, t) = \{1 \quad \xi \quad \eta \quad \xi^2 \quad \xi\eta \quad \eta^2 \quad \xi^3 \quad \xi^2\eta \quad \xi\eta^2 \quad \eta^3 \quad \xi^3\eta \quad \xi\eta^3\}\{\alpha\}, \tag{1}$$

$$\theta_x = \frac{1}{b}\frac{\partial w}{\partial \eta}, \quad \theta_y = -\frac{1}{a}\frac{\partial w}{\partial \eta}, \tag{2}$$

where t indicates a time dependance and $\{\alpha\}$ is a vector used to determine a general matrix $[\mathrm{A}_e]$ for the system. The process to obtain this matrix can be seen in detail in ref. [4].

Thus, for an element with thickness h, density ρ and surface area A, the kinetic and potential energy of the element can be described as being, respectively,

$$T_e = \frac{1}{2}\int_A \rho h \dot{w}^2 dA \tag{3}$$

and

$$U_e = \frac{1}{2}\int_A \frac{h^3}{12}\{\mathrm{X}\}^T[\mathrm{D}]\{\mathrm{X}\}, \tag{4}$$

where

$$\{\mathrm{X}\}^T = \{\partial^2 w/\partial x^2 \quad \partial^2 w/\partial y^2 \quad 2\partial^2 w/\partial x \partial y\} \tag{5}$$

and [D] (see ref. [4]) is the matrix of material constants; ν is the Poisson's ratio and E' and G are the Young's and shear modulus, respectively.

One can now transpose Eqs. (3) and (4) to general coordinates to find that

$$T_e = \frac{1}{2}\int_{-1}^{1}\int_{-1}^{1} \rho h \dot{w}(\xi, \eta)^2 (ab) d\xi d\eta = \frac{1}{2}\{w_e\}^T[\mathrm{M}_e]\{w_e\} \tag{6}$$

and

$$U_e = \frac{1}{2}\int_{-1}^{1} I\{\mathrm{X}\}^T[\mathrm{D}]\{\mathrm{X}\}(ab)d\xi d\eta = \frac{1}{2}\{w_e\}^T[\mathrm{K}_e]\{w_e\}, \tag{7}$$

where the matrices $[\mathrm{M}_e]$ and $[\mathrm{K}_e]$ are the matrices of mass and stiffness for the element.

Upon excitation of an external force f_e in the element, the virtual work W_e realized by this force is $\delta W_e = \{\delta w_e\}^T\{f_e\}$. Finaly, one can derive a general equation of motion for a system with n finite elements:

$$[\mathrm{M}]\{\ddot{\omega}\} + (1 + j\eta)[\mathrm{K}]\{\omega\} = \{f\} \tag{8}$$

where [M] and [K] are the global matrices of mass and stiffness (i.e., the sum and allocation of the individual matrices of mass and stiffness for each element) and ω is the vector of natural frequencies of the system.

For the sake of clarity, it is important to remind that the process to obtain the matrices $[\mathrm{A}_e]$, [D], $[\mathrm{M}_e]$, $[\mathrm{K}_e]$, [M], [K] and the vectors $\{\alpha\}$ and $\{f\}$ is described in ref. [4], as well as the assumptions and development of the mathematical expressions here exposed. This process will not be repeated here to avoid making the paper very long. It can also be visualized in the MATLAB code used in this work.

A. A FEM PROBLEM

The problem used in this work is illustrated in Fig. 2, where a door with its hinges and handle is pictured with its dimensions in mm. The problem consists in modelling this system, considering that the hinges and handle are fixed and cannot move and that a point force is applied at point "A", to assess its natural frequencies and modal shapes.

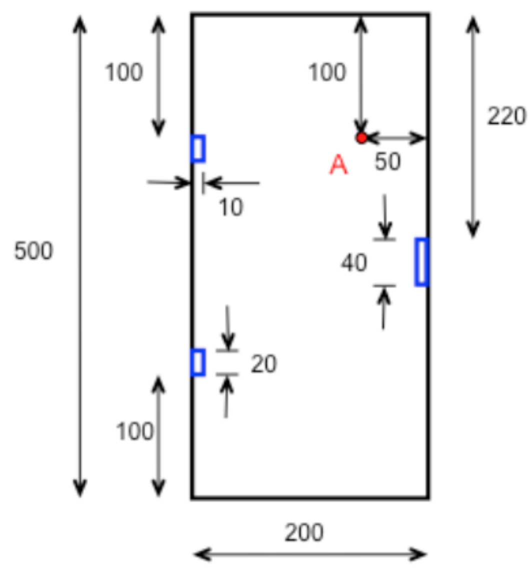

Figure 2: Problem used in the implementation (values in milimeters).

The following points will be evaluated in the implementation of this problem:

- The first six natural frequencies and modal shapes of the plate;
- A comparison of the results obtained with the algorithm with a commercial software.

3. METHODOLOGY

A. MATLAB IMPLEMENTATION

The code itself will not be put in this text, but for the reader to have glimpse of it in advance, the lines for the implementation of the matrices [Me] and [Ke], as well as the vector of forces, are shown in Fig. 3. Symbolic math is used in MATLAB for learning purposes.

```
%% Elementary Mass and Stiffness Matrices and Force Vector
Me = eval((A_e')\(int(int(rho*t*(p)*p'*a*b,csi,-1,1),eta,-1,1))/(A_e));
Ke = eval((A_e')\(int(int(Iz*P'*D*P*a*b,csi,-1,1),eta,-1,1))/(A_e));
fe = eval((A_e')\(int(int(p*a*b,csi,-1,1),eta,-1,1)));
```

Figure 3: Part of the implemented FEM code.

In the above figure $[\mathrm{A}_e]$ is the matrix that relates the degrees of freedom and their changes in coordinates. As an example and for comparison with Fig. 3, the equation for $[\mathrm{M}_e]$ in the theory is given below:

$$[\mathrm{M}_e] = [\mathrm{A}_e]^{-T} \left(\int_{-1}^{+1} \int_{-1}^{+1} \rho h \{p(\xi,\eta)\}^T \{p(\xi,\eta)\} abd\xi d\eta \right) [\mathrm{A}_e]^{-1} \tag{9}$$

i. Mesh

The mechanical properties of the plate were $E = 205$ GPa and $\rho = 7850$ kg/m^3. The thickness chosen for the problem was $h = 2$ mm and the Poisson's ratio was $\upsilon = 0.28$. The first step in this implementation is to model the mesh and its properties in order to determine the number of elements in both x and y axis, thus obtaining the nodes. The element size was chosen as 0.01 m in both axis and each element has 4 nodes and 3 degrees of freedom, as illustrated in Fig. 1. This leads to 20 and 50 elements in the x and y axis, respectively, totalizing 1000 elements in the mesh, which is shown in Fig. 4. The red circle and red elements represent the point A and the door handle and hinges, respectively.

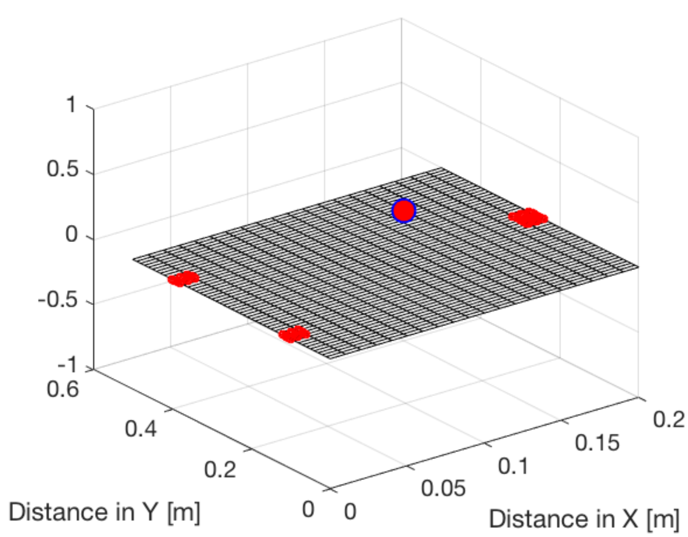

Figure 4: Finite element mesh for the plate.

ii. Connectivity matrix

The connectivity matrix is a mathematical tool that relates each element with its nodes, i.e. it tells the software which node belongs to each element and vice-versa. To explain how to program it, consider the four elements of a mesh shown in Fig. 5.

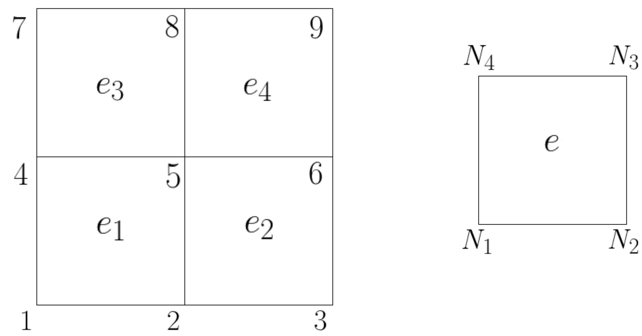

Figure 5: Elements of a typical mesh.

In the above figure, element one has the nodes 1, 2, 5 and 4, element 2 has the nodes 2, 3, 6 and 5 and so forth. A mathemcatical pattern can be used to identify which node belong to each element, as well as the nodes that are shared by two or more elements. Let N_i be the i-esimal node of each element in the above figure ($i = 1, ..., 4$), N_e the number of the element ($e = 1, ..., 4$), L and C be the number of lines and columns in the x and y axis, respectively and N_x the number of elements in the x axis in the mesh (4 for this study case). In this case, $L = 1$ or 2 and $C = 1$ or 2.

One can then easily find that the first node of each element can be found by the expression $N_1 = N_e + (L - 1)$. For the third element, for example, the first node is $N_1 = 3 + (2 - 1) = 4$. Since this third element is in the second row of the mesh, $L = 2$. For the fourth element, $N_1 = 4 + (2 - 1) = 5$ and so forth. The remaining nodes can be found as being $N_2 = N_e + L$, $N_3 = N_e + N_x + 3$ and $N_4 = N_e + N_x + 2$. This information is summarized in Table 1.

Table 1: Guide to find the connectivity matrix.

Element (N_e)	**Node 1** (N_1)	**Node 2** (N_2)	**Node 3** (N_3)	**Node 4** (N_4)
1	1	2	5	4
2	2	3	6	5
3	4	5	8	7
4	5	6	9	8
Expression	$N_e + (L - 1)$	$N_e + L$	$N_e + N_x + 3$	$N_e + N_x + 2$

iii. Global matrices

The matrices for the elements $[\mathrm{M}_e]$ and $[\mathrm{K}_e]$ are obtained considering that each element has 12 degrees of freedom in total, i.e. 3 degrees of freedom $\times$ 4 nodes. If the elements have 12 degrees of freedom, their matrices of mass and stiffness are 12 $\times$ 12 matrices. These are calculated and then allocated in the global matrix.

To study the allocation process, consider the mesh in Fig. 5, which has 9 nodes overall, meaning that the global matrices will have 9 $\times$ 9 "slots", with ij elements ocupying the i-esimal row and j-esimal column.

These slots will be filled with the relation of the degrees of freedom of each element with itself and with the degrees of freedom of other nodes.

Let's consider the first element of Fig. 5. The relation of the first node with itself, i.e., N_1N_1, will be filling the slot in the first line and first column. On the other hand, the relation between the first node and the third node, N_1N_3, will be filling the slot in the first line and fifth column (see in Fig. 5 that the third node of element 1 is marked as 5. In Table 2 it is presented the positions of all the relations of elements 1 (blue), 2 (green), 3 (yellow) and 4 (purple) of the mesh shown in Fig. 5. This table can be used as a guide to find and understand the global matrices of FEM problems.

From Table 2 it is clear that the global matrices of mass and stiffness shall be diagonal, with the non-nodes elements (blank spaces in this table) being equal to zero. Other thing that the reader might find out is that some relations are superposed, for instance the relations N_3N_3, N_4N_4, N_2N_2 and N_1N_1 for elements 1, 2, 3 and 4, respectively. It is explicit that such elements share the comon node N_5, which means that this relation is the same regardless of the element to which the node belongs, i.e. the relation of the node N_5 with itself is not variable, and this is the reason of the superpositioning in the table. The N_{ij} relations are, physically, the degrees of freedom for each node and how these degrees of freedom affect each other. When the reader looks at the ij element in Table 2, i.e. N_1N_1, she or he is looking at the first three degrees of freedom, w_1, θ_{x_1} and θ_{y1}, and how they relate with themselves.

Table 2: Guide to obtain the global matrices.

	1	2	3	4	5	6	7	8	9
1	N_1N_1	N_1N_2		N_1N_4	N_1N_3				
2	N_2N_1	N_2N_2	N_1N_2	N_2N_4	N_2N_3	N_1N_3			
3		N_2N_1	N_2N_2		N_2N_4	N_2N_3			
4	N_4N_1	N_4N_2		N_4N_4	N_4N_3		N_1N_4	N_1N_3	
5	N_3N_1	N_3N_2	N_4N_2	N_3N_4	N_3N_3	N_4N_3	N_2N_4	N_2N_3	N_1N_3
6		N_3N_1	N_3N_2		N_3N_4	N_3N_3		N_2N_2	N_2N_3
7				N_4N_1	N_4N_2		N_4N_4	N_4N_3	
8				N_3N_1	N_3N_2	N_4N_2	N_3N_4	N_3N_3	N_4N_3
9					N_3N_1	N_3N_2		N_3N_4	N_3N_3

iv. Natural boundary conditions (or geometric conditions)

Boundary conditions are the conditions imposed to parts of the system. The reader might ask himself the simple questions: "Can this part of my system move? How?". From the answers she or he will have the boundary conditions. If the edge of a plate can not move at all (i.e. it can not move in the *xy* plan nor can it rotate), it means the the nodes that are part of this edge will also have no motion, so the values of such nodes in the N_iN_j relations in Table 2 will be zero.

In the case of the problem showed in Fig. 2, the nodes that compose the door handle and hinges will be considered as constrained. It is simple do define the position of the nodes that will be "zero-nodes": if the door is 500 mm tall and the discretisation (element size) in the *y* axis is 0.01 m, the plate will have 0.5/0.01 = 50 elements. The easy way to do this is by deleting the rows and columns of the global matrices that correspond to constrained degrees of freedom. One can simply measure the spacial position of the hinges and handle and define the nodes that will be constrained in a similar process as showed in Table 1.

v. Extracting natural frequencies and modal shapes of the system

From Eq. 8, one can easily access the natural frequencies and modal shapes of the system through the *eigs* function in MATLAB, where its input arguments will be the global matrices of mass and stiffness. This function will return two essential parameters in the analysis proposed in this paper: *eigenvalues* (natural angular frequencies) and *eigenvectors* (modal shapes). Natural frequencies can then be assessed by the following equation in the implementation:

$$f_n = \frac{\sqrt{eigenvalues}}{2\pi} \tag{10}$$

4. COMSOL SIMULATION

To compare the results obtained from the MATLAB code the mechanical system was also implemented in COMSOL Multiphysics software. The implementation used the *Structural Mechanics Module* and the *Eigenfrequency Solver*. The discretisation of the mesh respected the six elements per wavelength requirement.[4] The force applied in point A (see Fig. 2) was a normal *Point Load* with 1 N, and, on the door, a *Spring Foundation* was added.

5. RESULTS AND DISCUSSION

Both MATLAB and COMSOL first six natural frequencies are shown in Table 3.

Table 3: Natural frequencies of the system.

Natural Frequency	1st	2nd	3rd	4th	5th	6th
MATLAB	57.98 Hz	59.25 Hz	143.83 Hz	161.88 Hz	169.13 Hz	234.73 Hz
COMSOL	59.13 Hz	62.60 Hz	153.05 Hz	163.24 Hz	167.25 Hz	236.02 Hz
Relative Error [%]	1.94	5.35	6.02	0.83	1.12	0.55

There is an excellent agreement between both results. The highest error is 6.02%, which occurs in the 3rd natural frequency. However, both values for it (143.83 Hz and 153.05 Hz) are within the same third-octave band, whose central frequency is 160 Hz. This means that if one is to treat this mode there would not be a very huge problem if the treatment is for either 143.83 Hz or 153.05 Hz. The other relative errors are so small that the results of the MATLAB code presented are trustworthy and feasible of achieving. The modal shapes for the MATLAB code are presented in Fig. 6.

In Fig. 6 the natural frequencies are rounded up to next integer. It is clear to see that in the positions of the handle and door hinges there is no motion at all, which is expected since they were degrees of freedom that were set to zero in the MATLAB code. When plotting the modal shapes in COMSOL, the agreement was also excellent between both results. For the sake of the length of this paper, COMSOL modal shapes were not put here, but readers can request the model file by email.

The processing time for MATLAB symbolic language is higher than if one uses numbers in the equations. However, the authors believe that using the first strategy is better for code learning and to understand how the software deals with the equations.

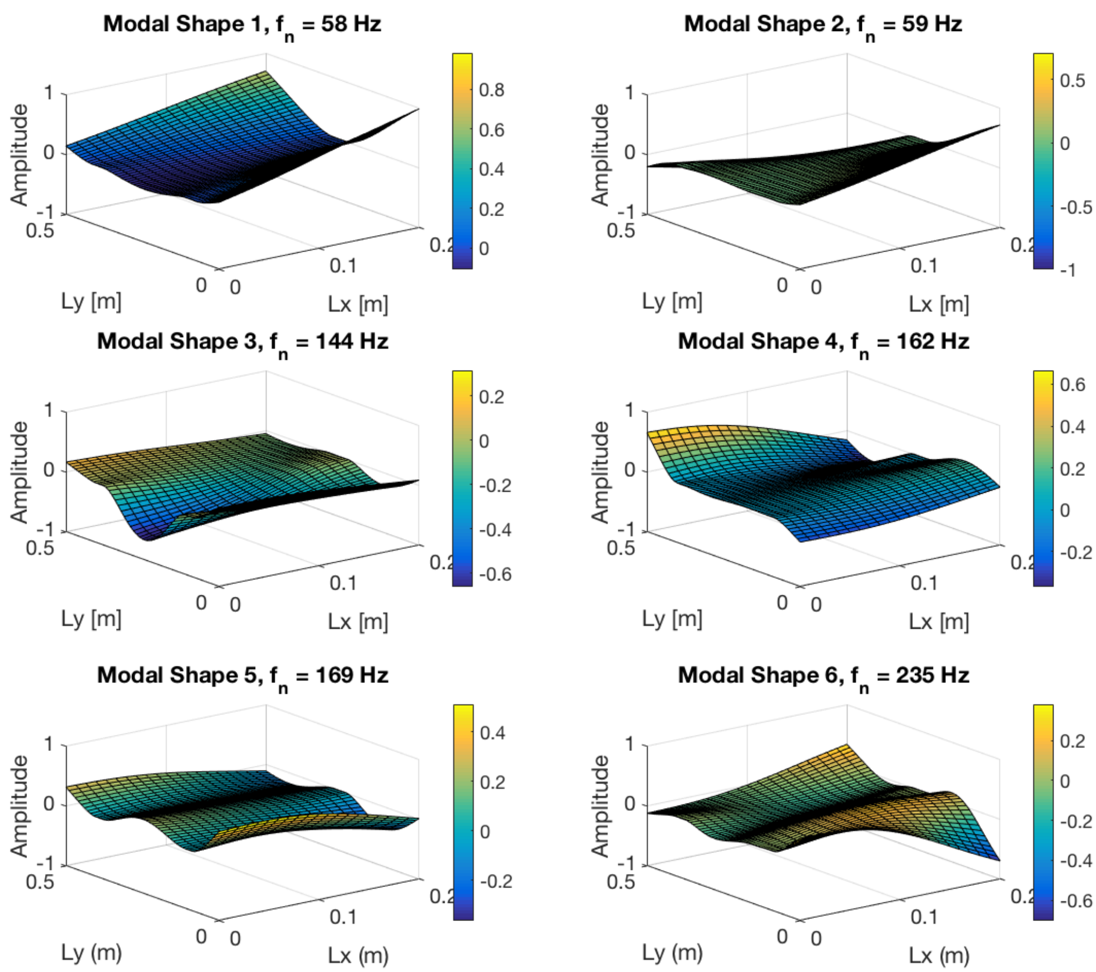

Figure 6: Modal shapes of the first 6 natural frequencies of the system.

6. CONCLUSION

The FEM algorithm presented here shows great agreement with commercial software results and can be used by any person who has the MATLAB code. The code consists of a good way to learn the Finite Element Method in its state-of-art. Its main disadvantage is that it is only valid for rectangular geometries. However, one can modify it to basically any geometry as long as the equations for the new shape(s) are provided.

ACKNOWLEDGMENTS

The authors would like to acknowledge Prof. Julio Cordioli for the lectures and FEESC for the financing of the Master's of the first author.

REFERENCES

[1] J. E. Mills, D. F. Treagust, *et al.*, "Engineering education – is problem-based or project-based? learning the answer," Australasian journal of engineering education **3**(2), 2–16 (2003).

[2] B. A. Mello, "Aumento na quantidade de alunos em disciplinas básicas: Como obter vantagens dessa realidade universitária.," Caderno Brasileiro de Ensino de Física **37**(3) (2015).

[3] C. Fiolhais and J. Trindade, "Use of computers in physics education," in *New technologies for higher education: proceedings*, Universidade de Aveiro (1998), pp. 103–115.

[4] M. Petyt, *Introduction to finite element vibration analysis* (Cambridge University Press, 2010).

[5] P. K. Banerjee and R. Butterfield, *Boundary element methods in engineering science*, Vol. 17 (McGraw-Hill London, 1981).

[6] A. R. Carvalho de Sousa and J. R. Bueno, "Self-taught finite element method implementation to the assessing of natural frequencies and modal shapes in 2d rectangular plates," The Journal of the Acoustical Society of America **141**(5), 3556–3556 (2017).

[7] J. Crompton, L. Gritter, S. Yushanov, K. Koppenhoefer, and D. Magyari, "Analysis of acoustic response of rooms," in *Proceedings of the COMSOL Conference* (2010).

[8] G. M. Eckhardt, R. Belk, and T. M. Devinney, "Why don't consumers consume ethically?," Journal of Consumer Behaviour **9**(6), 426–436 (2010).

[9] H. K. Cheng, R. R. Sims, and H. Teegen, "To purchase or to pirate software: an empirical study," Journal of Management Information Systems **13**(4), 49–60 (1997).

[10] R. R. Sims, H. K. Cheng, and H. Teegen, "Toward a profile of student software piraters," Journal of Business Ethics **15**(8), 839–849 (1996).

[11] A. Hrennikoff, "Solution of problems of elasticity by the framework method," Journal of applied mechanics **8**(4), 169–175 (1941).

[12] R. Courant, "Variational methods for the solution of problems of equilibrium and vibrations," Bulletin of the American Mathematical Society **49**(1), 1–23 (1943).

[13] J. Argyris, "Energy theorems and structural analysis, part l," Aircraft Engineering **26**(1), 383 (1954).

[14] M. Turner, R. Clough, H. Martin, and L. Topp, "Stiffness and deflection analysis of complex structures," Journal of Aeronautical Sciences **25**(1), 805–823 (1956).

[15] J. Meek, "A brief history of the beginning of the finite element method," International journal for numerical methods in engineering **39**, 3761–3774 (1996).