International Journal of Science and Research (IJSR)
ISSN (Online): 2319-7064
Index Copernicus Value (2013): 6.14 | Impact Factor (2013): 4.438

Analysis of Plane Two-Dimensional Structures by the Finite Element Method

F. HAMRIT[1], B. NECIB[2]

[1]Department of Mechanical Engineering, Faculty des Technology, University M'sila
[2]Laboratory of Mechanics - University Constantine1

Abstract: ***The discreet structures in the shape of rod or beam are an application of big importance in many sectors of the industry, as: the spatial mechanical construction or robotics, aerodynamic, the civil genius. The technique of calculation of these structures knew during these last years a considerable development. The method of elements finished is nowadays a powerful tool, available to the reasonable costs, the time of modelling is reduced henceforth, as the hold in hand the use of the computer. to reach of calculations of these structures less expensive and faster, one to develop a program in PASCHAL " language ", that permits the determination of displacements, reactions in nœudses, the axial strengths in elements and the clean fashions of the structure. Examples of verification have been made, under different loads and conditions to limits. The proposed structures have us permits to deepen our knowledge to the application of the method of elements finished in the static or dynamic discreet structure analysis. This developed program gives the best results of calculation compared to the software "SAP 2000 " and Robot 2009 software programs.***

Keywords: Finite element, element plate, matrix of rigidity, thrust loads, nodal displacement, vibration mode.

1. Introduction

The discrete structures are composed of bar, beam elements riveted or welded to each other at points called "nodes", and subjected to external forces or moments. Under the effect of these forces, the structure may be deformed and the internal stresses in each element may occur. These structures are characterized by a finite number of unknown displacements and the forces at the nodes parameters. This method is used also to continuous analysis tree dimensional bodies or cylindrical bodies using plate, triangular or shell elements. In fact, this method is based on the discretization of the structure or continuous body into infinitesimal elements limited by nodes and then by assembling them in order to obtain the overall and the entire structure [1]. Thus, the shape of the structure or the body is obtained in respecting its conditions with the initial limits and applied efforts. In general, the behavior of all the assembled basic members describes all of the behavior of the whole structure or body. The present work consists on the use of this method for static and dynamic analysis of structures under the influence of outside excitation with different boundary conditions. The stiffness and mass matrix of each element is computed, and then assembled to find the overall stiffness and mass matrix of the structure. In fact, the finite element method is known as a very powerful technique used to analysis discrete or continuum structures, in the field of engineering. It is now used in many sectors of the industry, mechanical, civil, aerospace and robotics. This work is devoted to the use of this method for static and dynamic analysis of structures in porches (plate element) due to excitement outside with different boundary conditions. Understanding this method gives necessity in the development of certain scientific knowledge as the theory of elasticity, mechanical environment continues, the strength of materials, structural dynamics, and applied mathematics. If the structure has a complex system of behavior and continues defined by the infinite number of parameters [2], it becomes very difficult to analyze or find the analytical solution. However, the finite element method grows the ability to find the most perfect solution while replacing the continuous system by a discrete system, characterized by a finite number of parameters [3].

In this context, a program of computation based on FORTRAN language has been developed. The displacements, forces and reactions at the nodes as well as the axial strengths in each element and the clean fashions of the all structure have been determinate under different applied loads and boundary conditions in static and dynamic cases. The obtained results are compared to those found using existing programs such as "SAP 2000" and "Robot 2009". A good comparison has been observed.

2. Problem and Methodology

The systems with one degree of freedom are simply illustrated by the system represented on figure (1) supposed in a vertical plan, and X (t) is the displacement of the mass m starting from the equilibrium position [4].

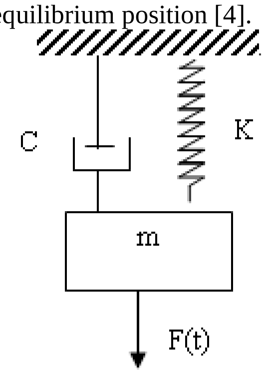

Figure 1: System with one degree of freedom

Its equation of the motion is given by:

$$m\ddot{x}(t) + c\dot{x}(t) + kx(t) = F(t) \qquad (1)$$

and the solutions of the characteristic equations are:

$$r_{1,2} = -\frac{c}{2m} \pm \frac{\sqrt{c^2 - 4km}}{2m}$$

Then, we can rewrite the equation of motion without excitation as:

$$\ddot{x} + 2w_0 \alpha \dot{x} + w_0{}^2 x = 0 \quad (2)$$

When the system mass-spring without damping is submitted to an excitation, the equation of motion becomes [5]:

$$m\ddot{x}(t) + kx(t) = F(t) \quad (3)$$

In our study, the equations of motion of the element plates are formulated in a similar way to equation (3), but with several degrees of freedom, this will be formulated using the matrix form and will be presented as:

$$[m]\{\ddot{x}\} + [k]\{x\} = \{F(t)\} \quad (4)$$

where [k]: matrix of rigidity
[m]: matrix mass
{x}: vector of nodal displacements
{ $\ddot{x}$ }: nodal acceleration vector
{F(t)}: nodal applied forces

3. Mathematical Formulation

3.1 Oscillation Forced

The existence of even low friction, leads irremediably to the extinction of the oscillatory motion by dissipation of energy (internal or external). It is necessary to maintain the movement, to external energy in the form of a so-called power exciter. It will be admitted that (resonator) system does not react on the exciter.

3.2 Harmonic Excitation

The equation of motion for a damped harmonic oscillator subjected to a force external F (t) is written as:

$$m\ddot{x}(t) + c\dot{x}(t) + kx(t) = F(t) \quad (5)$$

The simplest case is that when the excitation is a harmonic force in the form:

$$F(t) = F\cos(wt + \theta)$$

The general solution of the equation of motion is then a linear combination of the general solution of the equation without second member (free oscillations plan), and a particular solution of the equation with second member. As previously, we can rewrite equation 5 as follows:

$$\ddot{x}(t) + 2\alpha w_0 \dot{x}(t) + w_0{}^2 x(t) = \frac{F(t)}{m}$$

3.3 Definition of the generalized Eigen values problem

Assuming that the discretized system is stationary (coincidence of fixed and linked axes) and neglecting the depreciation of viscous matrix, the free discrete associated equation of motion will be expressed in the following form [6]:

$$M\ddot{x}(t) + Kx(t) = 0 \quad (6)$$

4. Modeling of the Plate by Finite Elements

We consider the mechanical behavior of each element separately and then we assemble these elements so that the balance of forces and the compatibility displacements are met in each node. FEM uses simple approximations of variables unknowns in each element for transform partial differential equations into algebraic equations. The nodes and elements do not necessarily have particular physical significance, but are based on approximating precision considerations. Define nodes and elements (Create Mesh). So for each item, determine the elemental stiffness matrix linking the degrees of freedom (travel) and nodal forces, applied to nodes [7]. Assembly of elementary matrices in a system global [K] {U} = {F} to satisfy the equilibrium conditions at the nodes, the boundary conditions are applied to solve the system [K]{U} = {F}.

4.1 Application of the finite element method to a rectangular element in a plane state of stress and stain deformation:

The finite element of the plane state simplest is a rectangular element as shown on the figure (2). The element has a length (a) and a width (b) with a constant thickness (h); each of the four corners or nodal points has two degrees of freedom: *u* and *v* displacements in *x* and *y* direction respectively. Thus this element has eight nodal masses (four pairs of m_x and m_y) and eight nodal displacements or nodal degrees of freedom (four pairs of *u* and *v*).

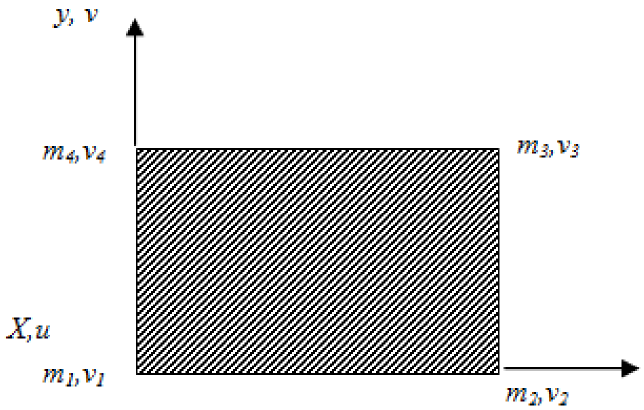

Figure 2: Rectangular finite element with eight degrees of freedom at a plane state of stress or strain deformation

As a result, the mass matrix of the element can be found as follows:

$$[M_e] = \rho.a.b.h \begin{bmatrix} \frac{1}{9} & \frac{1}{18} & \frac{1}{36} & \frac{1}{18} & 0 & 0 & 0 & 0 \\ \frac{1}{18} & \frac{1}{9} & \frac{1}{18} & \frac{1}{36} & 0 & 0 & 0 & 0 \\ \frac{1}{36} & \frac{1}{18} & \frac{1}{9} & \frac{1}{18} & 0 & 0 & 0 & 0 \\ \frac{1}{18} & \frac{1}{36} & \frac{1}{18} & \frac{1}{9} & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{9} & \frac{1}{18} & \frac{1}{36} & \frac{1}{18} \\ 0 & 0 & 0 & 0 & \frac{1}{18} & \frac{1}{9} & \frac{1}{18} & \frac{1}{36} \\ 0 & 0 & 0 & 0 & \frac{1}{36} & \frac{1}{18} & \frac{1}{9} & \frac{1}{18} \\ 0 & 0 & 0 & 0 & \frac{1}{18} & \frac{1}{36} & \frac{1}{18} & \frac{1}{9} \end{bmatrix} \quad (7)$$

4.2 Rigidity matrix of the rectangular element:

The elementary matrix of rigidity can be derived by initially formulating the strain energy deformation U for the element and then by carrying out the differentiation partial equation of this energy with respect to each degree of freedom according to the theorem of Castigliano given by:

$$F_i = \frac{\partial U}{\partial u_i}$$

Where the strain energy deformation of the finite element is found using a linear the displacement at (x,y) position [8],

$$u(x,y) = c_1 + c_2 x + c_3 y + c_4 xy$$
$$v(x,y) = c_1 + c_2 x + c_3 y + c_4 xy \quad (8)$$

Where c_i are constants found using boundary conditions at each nodal point of the element. That is the equation 8 becomes:

$$u(x,y) = f_1(x,y)u_1 + f_2(x,y)u_2 + f_3(x,y)u_3 + f_4(x,y)u_4$$
$$v(x,y) = f_1(x,y)v_1 + f_2(x,y)v_2 + f_3(x,y)v_3 + f_4(x,y)v_4 \quad (9)$$

Where the four functions f (x,y) are called the displacement functions obtained as:

$$f_1(x,y) = \left(1 - \frac{x}{a}\right)\left(1 - \frac{y}{b}\right)$$
$$f_2(x,y) = \frac{x}{a}\left(1 - \frac{y}{b}\right) \quad (10)$$
$$f_3(x,y) = \frac{xy}{ab}$$
$$f_4(x,y) = \left(1 - \frac{x}{a}\right)\frac{y}{b}$$

As a result, the tensor strain deformation can be found and expressed as[9]:

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \begin{bmatrix} -\frac{1}{a}\left(1-\frac{y}{b}\right) & 0 & \frac{1}{a}\left(1-\frac{y}{b}\right) & 0 & \frac{y}{ab} & 0 & -\frac{y}{ab} & 0 \\ 0 & -\frac{1}{b}\left(1-\frac{x}{a}\right) & 0 & -\frac{x}{ab} & 0 & \frac{x}{ab} & 0 & \frac{1}{b}\left(1-\frac{x}{a}\right) \\ -\frac{1}{b}\left(1-\frac{x}{a}\right) & -\frac{1}{a}\left(1-\frac{y}{b}\right) & -\frac{x}{ab} & \frac{1}{a}\left(1-\frac{y}{b}\right) & \frac{x}{ab} & \frac{y}{ab} & \frac{1}{b}\left(1-\frac{x}{a}\right) & -\frac{y}{ab} \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \\ u_4 \\ v_4 \end{Bmatrix} \quad (11)$$

Or in the matrix form:

$$\{\varepsilon\} = [A]\{q\} \quad (12)$$

The equations of rigidity for the rectangular element at eight degree of liberty can be found as[10]:

$$\begin{Bmatrix} F_{x1} \\ F_{y1} \\ F_{x2} \\ F_{y2} \\ F_{x3} \\ F_{y3} \\ F_{x4} \\ F_{y4} \end{Bmatrix} = \begin{bmatrix} C_1 & & & & & & & \\ C_2 & C_3 & & & & & & \\ C_4 & -C_5 & C_1 & & & & & \\ C_5 & C_6 & -C_2 & C_3 & & & & \\ -\frac{C_1}{2} & -C_2 & C_7 & -C_5 & C_1 & & & \\ -C_2 & -\frac{C_3}{2} & C_5 & C_8 & C_2 & C_3 & & \\ C_7 & C_5 & -\frac{C_1}{2} & C_2 & C_4 & -C_5 & C_1 & \\ -C_5 & C_8 & C_2 & -\frac{C_3}{2} & C_5 & C_6 & -C_2 & C_3 \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \\ u_4 \\ v_4 \end{Bmatrix} \quad (13)$$

Then:

$$C_1 = \left(\frac{b}{3a} + \frac{1-\nu}{6}\frac{a}{b}\right)\frac{Et}{1-\nu^2} \quad C_2 = \left(\frac{\nu}{4} + \frac{1-\nu}{8}\right)\frac{Et}{1-\nu^2}$$
$$C_3 = \left(\frac{a}{3b} + \frac{1-\nu}{6}\frac{b}{a}\right)\frac{Et}{1-\nu^2} \quad C_4 = \left(-\frac{b}{3a} + \frac{1-\nu}{12}\frac{a}{b}\right)\frac{Et}{1-\nu^2}$$
$$C_5 = \left(\frac{\nu}{4} - \frac{1-\nu}{8}\right)\frac{Et}{1-\nu^2} \quad C_6 = \left(\frac{a}{6b} + \frac{1-\nu}{6}\frac{b}{a}\right)\frac{Et}{1-\nu^2} \quad (14)$$
$$C_7 = \left(\frac{b}{6a} + \frac{1-\nu}{6}\frac{a}{b}\right)\frac{Et}{1-\nu^2} \quad C_8 = \left(-\frac{a}{3b} + \frac{1-\nu}{12}\frac{b}{a}\right)\frac{Et}{1-\nu^2}$$

5. Examples of Applications

For the resolution of a plane elasticity problem by the finite element method, we will use the Pascal program and the result will be compared to our developed program based on the sap2000 software for an isotropic solid plate in a plane state of stresses. An elastic problem is solved once the displacement vector in any point of the solid and the forces at each point of solid are known, that is the problem with its boundary conditions is known. Some examples are considered to examine our developed program. One considers a homogeneous plate which we will discretize in rectangular equal elements, (n) element in the X axis (m) element in the Y axis, for a static case we can calculate the nodal displacements (u_i, v_i), the reactions at each nodal points and the stress in the medium of each element; then for the dynamic case, we will calculate the Eigen frequencies and the Eigen values, and a comparison of the obtained results between our developed program and SAP2000 will realized.

5.1 Modeling of a rectangular plate by the finite element method

The study of the examples relates to a rectangular plate in plane state of stress under various boundary conditions, to analyze this structure (thin plate) by the finite element method by discretizing this plate in very small rectangular elements (fig 2).

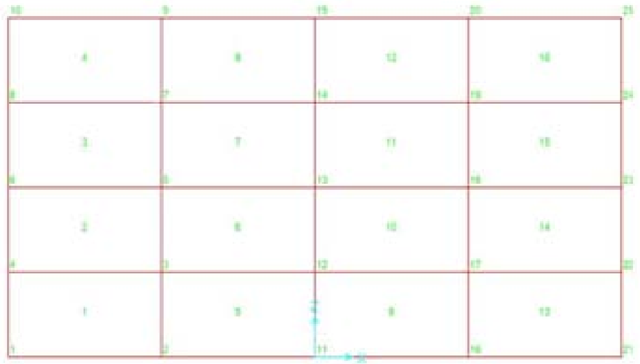

Figure 3: Discretization of the thin plate (rectangular finite elements)

5.1.1 Example of application N°1:

We consider a steel rectangular plate in a plane State of stress with a Young's modules E = 2.10^{11} N/m^2, a Poisson's ration, a density, a length of 2 m and width of 1 m, clamped on one side and free on the other sides. The plate is discretized into 16 elements and 25 nodes, while the applied forces are primarily concentrated to the node 25, linearly then uniformly distributed on each node (figures 3).

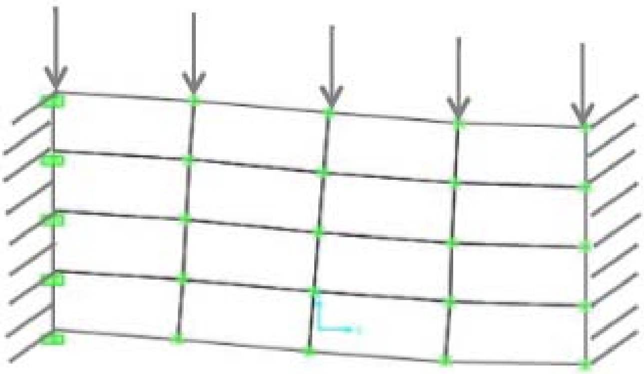

Figure 4: Mode of vibration N°1, with a frequency of w= 2045.41HZ

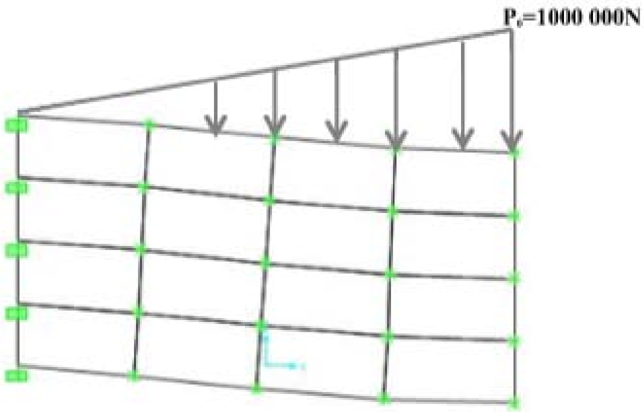

Figure 5: Mode of vibrationN°2 with a frequency of w= 7132.37HZ

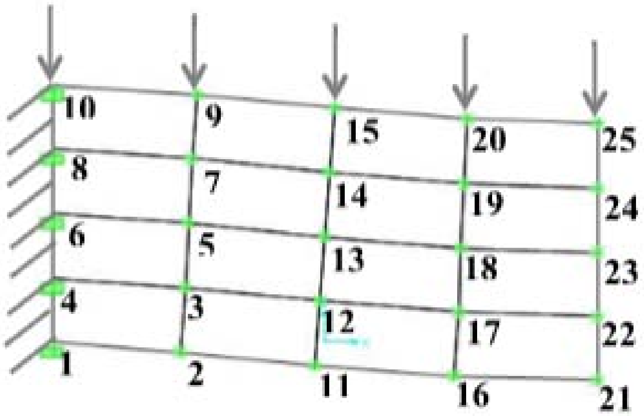

Figure 6: Mode of vibration N°3 with a frequency of w= 13186.3HZ

$P(x) = -P_0\ (P/x)$ *avec* P_0=1000 000 N

$F_{y9}=F_{y10}=F_{y15}=F_{y20}=F_{25}$=-1000 000 N

5.1.2 Exemple of application N°2 :

The same plate is considered but under different types of boundary conditions (figures 7, 8 and 9) and different external forces

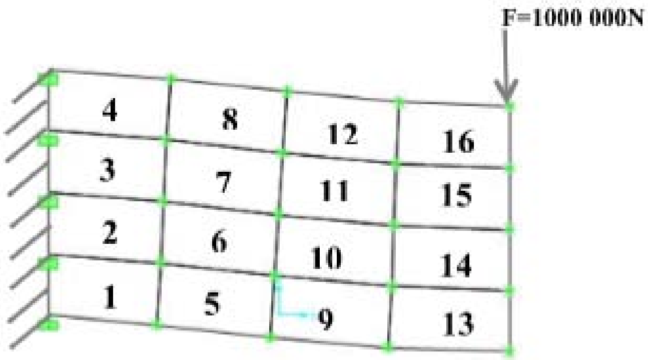

Figure 7: Mode of vibration N°4 of a frequency w= 14067.89HZ

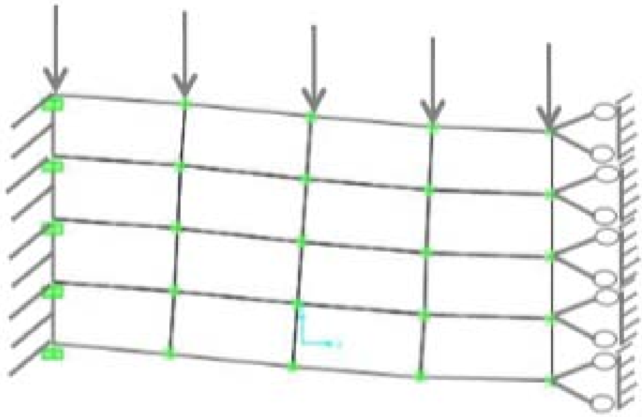

Figure 8: Mode of vibrationN°4 of a frequency w= 15632.62HZ

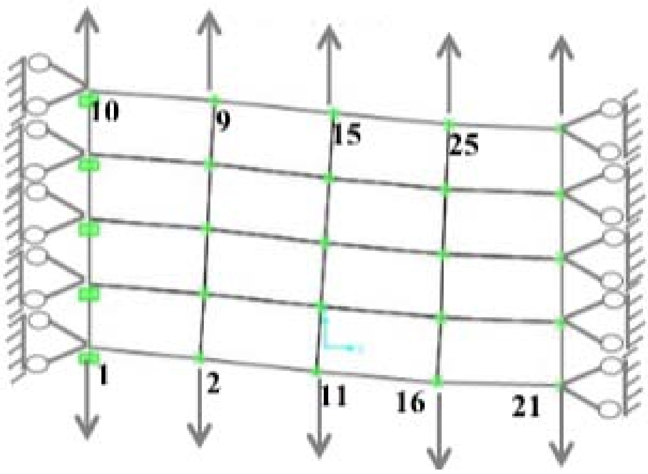

Figure 9: Mode of vibration N° 4 of a frequency w= 17218.97HZ

$$F_{y1} = F_{y2} = F_{Y11} = F_{y16} = F_{y21} = 1000000.N$$

The displacement of each node, reaction and the stresses in the middle of each element of the plate are obtained comparatively with our developed method and the software Sap 2000 (figures 9-13); the first two specific modes of vibration of each case are shown in figures 14 and 15 and good results are obtained.

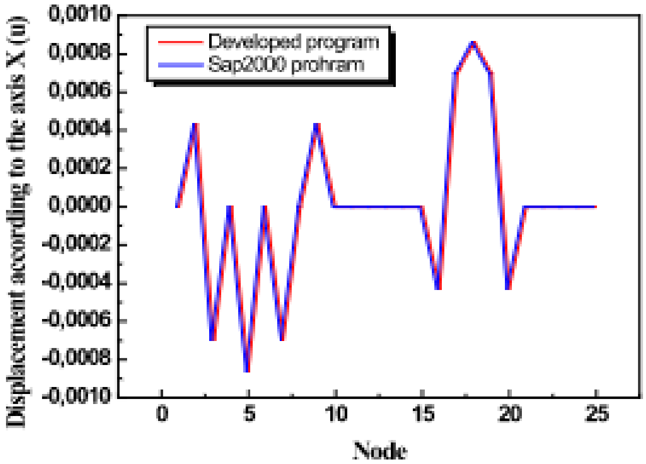

Figure 10: Nodal displacements in X direction

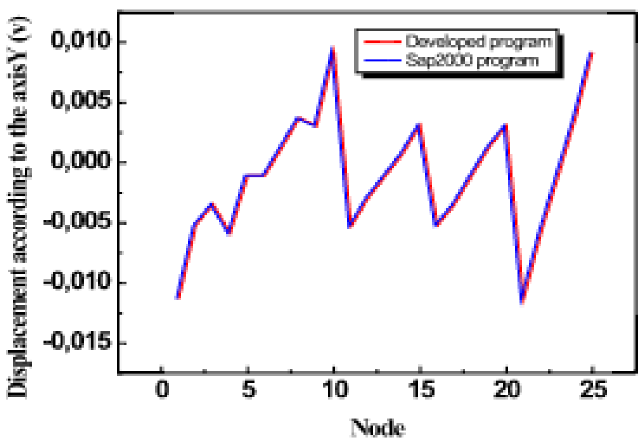

Figure 11: Nodal displacements in Y direction

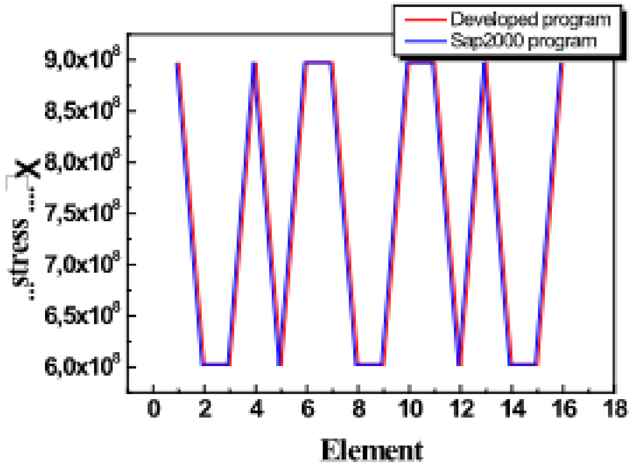

Figure 12: Stress comparison (δ_x)

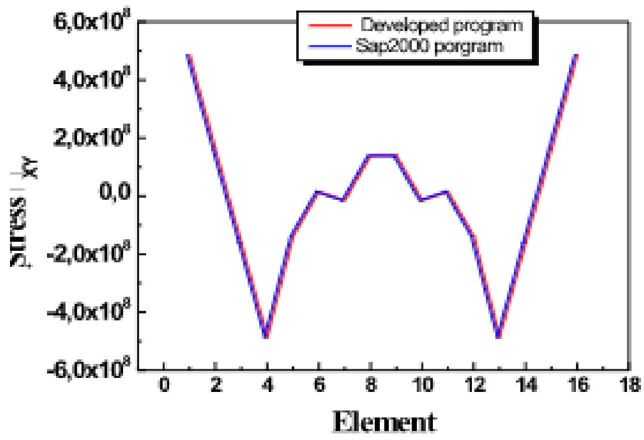

Figure 13: Stress comparison (τ_{xy})

Table 1: Calculating the natural frequencies and mode for X axis

Nodes	The movement of nodes following the X axis		
	Mode N0 1 W1=2045.41 HZ	Mode N^0 2 W_2=7132.37 HZ	Mode N^0 3 W_3=13186.3 HZ
1	0	0	0
2	-0.007354	0.019429	-0.006662
3	-0.003022	0.011947	-0.006874
4	0	0	0
5	-0.000483	0.016534	-0.004897
6	0	0	0
7	0.002054	0.020507	-0.003044
8	0	0	0
9	0.0006341	0.011875	-0.004844
10	0	0	0
11	-0.00886	-0.0144619	-0.007366
12	-0.004303	0.006157	-0.009593
13	-0.000831	0.01975	-0.008065
14	0.002663	0.032639	-0.004369
15	0.007327	0.049855	-0.002283
16	-0.005875	-0.031099	-0.00658
17	0.003004	-0.0055	-0.006183
18	-0.000731	0.008959	-0.006481
19	0.001578	0.023023	-0.003963
20	0.004577	0.048455	0.001913
21	0	0	0
22	0	0	0
23	0	0	0
24	0	0	0
25	0	0	0

Table 2: Calculating the natural frequencies and mode for Y axis

Nodes	Movements following nodes Y axis		
	Mode N^0 1	Mode N^0 2	Mode N^0 3
1	0	0	0
2	-0.009154	0.057522	-0.003704
3	-0.008463	0.057283	-0.00192
4	0	0	0
5	-0.008247	0.056437	0.000972
6	0	0	0
7	-0.008282	0.054446	0.003259
8	0	0	0
9	-0.008795	0.050765	0.004186
10	0	0	0
11	-0.020449	0.042572	0.005808
12	-0.020513	0.046345	0.006103
13	-0.020427	0.048072	0.007725
14	-0.020395	0.050475	0.010243
15	-0.020259	0.049979	0.011129
16	-0.029708	-0.029065	0.007213
17	-0.0302	-0.028315	0.006101
18	-0.030371	-0.025226	0.004527
19	0.030287	0.021498	0.003895
20	-0.029912	0.017089	0.004076
21	-0.033192	-0.069242	0.002982
22	-0.033882	-0.07169	0.001247
23	-0.034144	-0.069691	-0.001507
24	-0.034106	0.066365	-0.004206
25	-0.033642	-0.060002	-0.004827

6. Discussion of the Results

The finite element method is a tool of approximate numerical resolution of the problems of structures and, more generally, physical problems and mechanics and others. It allows the determination of the node displacements of a structure or a plate under any boundary conditions. The essential object of calculation for our case is to determine displacements, the reactions, as well as the stresses in any element of the bi dimensional plate. According to the results found for the various examples that we have considered, we can make the following observations:

6.1 For the Static Case

The results obtained using our developed program (in the static case):

1. The node displacements in axes X and Y direction are very close to the results founds using Sap2000 software with small for errors
2. For the values of the reactions we note that the obtained results are completely equivalent to those obtained using Sap2000.

6.2 For the Dynamic Case

The calculation program in a dynamic state gives the Eigen frequencies and the mode of vibration. For the modes, we say that the low frequencies of the first mode are 1505.1HZ for the example N°1; 5116.04 for the example N°4; and 2045.4HZ for example N°5. It is noted that the most important frequency of vibration is that of example 2 which corresponds to the clamped-clamped problem. Which means that this plate gets more of the vibrations.

7. Conclusion

The digital simulation of the mechanical or physical systems is based on several types of methods, in particular finite elements (variational principles), elements of border (linear equations of field put in the form of integral equations), finite differences (approximations of derived in differential equations or with the derivative partial) and finite volumes (integral form of the conservation equations). The simulation of the mechanical behavior of the deformable solids and the systems made up of such solids, tallies general of this memory, generally rests on the finite element method. This work will be subscribed in to develop a program as a Pascal allowing static and dynamic calculation thin sections in plane state of stress under various boundary conditions, even the determination in static analysis of displacements of each node, reactions, forced in some points of the element and the determination of the frequency and the clean modes in dynamic analysis. The comparison between the results obtained and those of software SAP2000 shows a very good agreement. The case of the plane stress will be applied to many examples like the structures with thin walls, the structures comprising of the veils of stiffening, the beams out of box and sometimes the hulls (the bridges, the tanks, ships and airplanes).

References

[1] ANDRE FORTIN " les éléments finis de la théorie a la pratique", Ecole polytechnique de Montréal, 2011.
[2] EL GHAZI A. et EL HAJJI S. (2004): " Un petit guide d'utilisation du logiciel MATLAB ", département mathématiques et informatique, université Mohammed V, Maroc.
[3] Pierre SPITERI, " présentation générale de la méthode des éléments finis ", Ecole nationale supérieur de Toulouse, article [AF505].
[4] M. Lalanne, P. Berthier, J. DerHagobian , "Mécanique des vibrations linéaires ", MASSON édition, (1995)
[5] Yang .T .Y "Finite element structural analysis", Prentice-Hall (1986).
[6] B. NECIB, « Théorie générale de la mécanique des milieux continus » Polycopié – Editions 2010, Université Mentouri Constantine.
[7] Ph. Trompette, " Mécanique des structures par la méthode des éléments finis (Statique et dynamique)", édition, (1992).
[8] M. Abdi. K. Mohamed. Numerical Simulation and Active Vibration Control of Piezoelectric Smart Structure. International Review of Mechanical Engineering. Vol 3.pp 175-181.2009
[9] O.C. ZIENKIEWIEZ: "The finite element method in engineering analysis", Paris, (1991).
[10] A. Lebeid. B.Necib. Analysis and numerical modelling of ceramic piezoelectric beam. International Review of Mechanical Engineering. Vol 3.pp 454-456.2011.
[11] Nicolas BRIE, Mise en oeuvre d'un calcul de modes propres d'une structure, Copyright 2014 EDF R&D - Document diffusé sous licence GNU FDL (http://www.gnu.org/copyleft/fdl.html), 2014

Author Profile

Department of Mechanical Engineering, Faculty of Engineering Sciences, University Constantine 1, Road of Ain-el-Bey. 25000, Constantine, Alegria

Fareh HAMRIT, Teacher Researcher, in the Mechanical Engineering Department, Faculty of Technology, University Msila, Road of Ichbilia 28000, Alegria

Prof. B. NECIB, PhD in Aeronautical Engineering Purdue University USA, Mechanical Engineering Department, Faculty of Engineering Sciences, University Constantine 1m Road of Ain-el-Bey.25000, Constantine, Algeria