Numerical Solution of the Partial Differential Equation Bilaplacian type by the Finite Element Method for the simulation of Accelerometer-type MEMS

José David, ALANÍS URQUIETA
Division of Technologies of Information, Technological University of Puebla
Puebla, Puebla, 72300, México

Blanca, BERMÚDEZ JUÁREZ
Faculty of Computer Science, Meritorious Autonomous University of Puebla
Puebla, Puebla, 72592, México

Paulo Daniel, VAZQUEZ MORA
Division of Technologies of Information, Technological University of Puebla
Puebla, Puebla, 72300, México

Armando, HERNÁNDEZ FLORES
Postgraduate Department, Master in Automation Engineering
Polytechnic University of Puebla
Juan C. Bonilla, Puebla, 72640, México

ABSTRACT [1]

In this paper, the numerical solution of the Partial Differential Equation Bilaplacian type by the finite element method is presented in order to simulate the accelerometer-type MEMS behavior. The above mentioned solution is used to emulate the behavior of the deformation of an Accelerometer-type MEMS. The technique is the physically based modelling as a methodology of simulation with visualization that was used to solve the current problem. The first step is to solve the partial differential equation, which represents the structure, by the finite element method. This numerical method was instrumented in Octave, taking into account the primitive functions that it contains, and taking advantage of the powerful language, and free software resource. For this problem, the software built, it has results suitable for these types of problems and has well rates of error. Once these types of results have been obtained, the next step will be the rendering and interpretation of the results in a graphical way. In spite of the complexity and size in memory used by the numerical method, this procedure results be a good alternative for this case and maybe in other similar cases. In future works will be looking for parallelize some numerical methods.

Keywords: Partial Differential Equation, Finite Element Method, MEMS, Numerical Solution, Simulation.

1. INTRODUCTION

A MEMS (Micro Electromechanical Systems: by its acronym in English) can be defined as a way to design and develop electrical and mechanical systems in a scale in order of the micrometers [1]. The effects of the application of the MEMS into more complex devices or adapt them as sensors, actuators, electronic noses, they are possible to perceive even to a big scale [1]. All above mentioned are used in order to instrument solutions that generally are used in electronic devices that they get measures or real-world parameters. The common use of the MEMS Accelerometer-type inside the industry of mobile devices, overall, is to control the display orientation into the cell phones, tablets, among other devices [4].

The modelling technique called physically based modelling is considered as a methodology of simulation with visualization that was used in order to solve the current problem. First a physical model is obtained from the real world. In this case, it is about a rectangular plate simply supported, subject in the four extremes and an initial force is applied over all the surface, in the Z axis, that means, in a perpendicular plane of the surface's plate in 3D perspective.

This force in general can be described as a sinusoidal function and as a result of the vibration it has a deformation.

There is a variety of properties of the material when the force is applied in different ways, however the most important property is the deformation of the material for this particular case [1] [4].

Then a mathematical model is obtained that results to be a partial differential equation, in this case Bilaplacian-type. This means a traditional Laplacian but squared, with certain function in the right side [1].

It is considered for this work a simple plate made of certain material like gallium, silicon, among others, chosen with the right parameters of Young's modulus, or traction modulus, defined as the degree of stiffness of elastic material, inertial moment or resistance to the flexion, and the Poisson ratio, known as the relationship of the contraction of the transversal tension with the axis where the impulse is exerted. So, when the mathematical model is completed, the surface considered for this work is the rectangular plate with certain measures in the x axis and y axis [4].

The next step is to solve the partial differential equation by the finite element method, first the mesh is obtained, this is chosen by taking into account another class of experiments without simulation in order to obtain major accuracy level, then by approximation functions the plate area is covered by these functions and finally the results are compared with those obtained by the exact solution [1] [4].

In the end it is to obtain the visualization of the solution in a graphical way in order to offer to the user a view of the results in 3D, and compute the errors rate of the compute between the approximation and the exact solution [4].

In this work the numerical solution of a partial differential equation bilaplacian-type by the finite element method is presented, implemented in Octave, taking in advantage the language's potency, the libraries included into the package of software and the fact of that is an open-source resource.

In Section 2 the mathematical model is presented, that is the partial differential equation. This equation describes the deformation of the accelerometer with the initial conditions for this device. In section 3 the finite element method, applied to this differential equation is described, as a base of the numerical simulation of the cited equation and the parameters that are necessary to numerical solution. In section 4 the results obtained in the numerical simulation are presented, given the instrumentation of the solution performed in the past section. In section 5 the conclusions and future works about the present topic are presented.

2. MATHEMATICAL MODEL

As can be seen in [1] as the theoretical support, it is a simulation of a MEMS accelerometer-type. MEMS is a non-deformable plate it can be square or rectangular, it subject to an extreme or in all its extremes. In this case, is taken an accelerometer to subject by all its extremes and that it receives an electrical impulse in order to make it vibrate, in such a way that this impulse provoke the deformation of the material in the axis z, this is the main subject of study for almost all the MEMS.

In order to make this simulation with visualization, it is used a mathematical model that governs the deformation of the MEMS accelerometer-type, which is given by the [1]:

$$\frac{\partial^4 \omega}{\partial x^4} + 2\frac{\partial^4 \omega}{\partial x^2 \partial y^2} + \frac{\partial^4 \omega}{\partial y^2} = q \qquad (1)$$

Where ω the deformation of the material and q is the intensity of the force applied to the plate. The equation (1) is called bilaplacian partial differential equation; due to that, it has a Laplacian square operator. This force it is described in terms of the follow equation [1][2]:

$$q = \frac{q_0}{D} \qquad (2)$$

Where q_0 is the force in the center of the plate, the parameter D, is the rigidness of the material in opposition to the flexion and it is determined by: [2]

$$D = \frac{EI}{(1-v^2)} \qquad (3)$$

From equation (3) it is obtained the meaning [1]:
E: is the Young's modulus or traction's modulus, is a measure of characterization of the materials that measure the degree of rigidity in an elastic material.
I: is the moment of inertia, this is, the resistance of a material to the flex, in accord with this, the major moment of inertia is given before the material it is folded completely.
v: is the Poisson's radio, that is the relationship of the contraction with transversal to the axis that exerted the tension or impulse.

For this problem, the following parameters are used, in order to obtain the solution of the differential equation, it is worth mention that are not a results of others simulations, but rather comes from an example designed in Chapra Book, and below it is presents in the Table 1 [2][4].

Parameter	Not mentioned material
Young's Modulus	$2 \times 10^{11}\ Pa$
Inertia moment	8.33×10^{-8}
Poisson's ratio	0.3

Table 1. Initial measures for the calculus of parameters D in the right side of the equation

For this case the initial value of the force $q_0 = 33.6\ N/m^2$, this value is used in equation (1).

Given the previous equations now it is shown the initial conditions and of border of the problem in question, firstly it is known that the accelerometer it is fastened by the four extremes, which describe the following border conditions to a plate simply supported [1].

$$\omega = 0, \quad \frac{\partial^2 \omega}{\partial x^2} = 0 \quad for\ x = 0\ and\ x = a \qquad (4a)$$

$$\omega = 0, \quad \frac{\partial^2 \omega}{\partial y^2} = 0 \quad for\ y = 0\ and\ y = b \qquad (4b)$$

In this case the dimensions of the sides are denoted by *a* and *b*, then the extremes are *x=a* and *y=b*, as it is said the last denote that the plate are simply supported and subject in its extremes: The border conditions come given as follow [1]:

$$\omega = 0, \quad M_x = 0 \ for \ x = 0 \ and \ x = a \quad \text{(5a)}$$

$$\omega = 0, \quad M_y = 0 \ for \ y = 0 \ and \ y = b \quad \text{(5b)}$$

Where M_x and M_y are the moments of flex that it is observed when the material is subjected to the force that already mentioned, in the sense of the z-axis.

The implementation of this type of problem and as it can see at [2][5][6] above mentioned equation can be solved by finite element method, and performed the numerical simulation as it can be seen in [2], and with following ideas taken from the method proposed in [2][6]:

Taking in account that the bilaplacian-type equation in (1) is proposed as a square Laplacian as it can see at [2]. First, a new variable is introduced, called *u* and defined as:

$$u = \frac{\partial^2 z}{\partial x^2} + \frac{\partial^2 z}{\partial y^2} \quad \text{(6)}$$

So, it can be defined a new variable that provoke it be necessary the solution of one Poisson-type equation and rewrite the equation number (6) as follow:

$$\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} = \frac{q}{D} \quad \text{(7)}$$

This manner, the problem is come down to solving two equations in a successive way. First the equation (7) must be resolved, and is obtained u subject to the first boundary condition denoted as *u=0* in the extremes. Then, these results are used in the solution of the equation (6), that obtain the deformation with the boundary condition *z= 0*. Now the method of finite element it is proposed in [2] will be explained in the next section of this work.

3. FINITE ELEMENT METHOD

The finite element method consists in the following phases or steps as follow [6] [2]:

- Discretization
- Weak Form
- Element Equations
- Boundary Conditions and Assembly
- Analysis and Post Processing

As following of this, each one of the stages are briefly described, since that this method is widely known in the area of Solution of Partial Differential Equation, and too is widely documented in many books, articles among many other bibliographic resources.

3.1 Discretization

In this stage, it is taken the region, either rectangular or square, and it is divided into small squares in accord with the number of divisions stablished by the user or by the needs of the system, by each side.

Then the squares conformed it is divided by one diagonal obtaining the triangular elements, as it can see in the Figure 1 as follow:
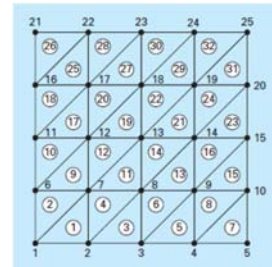

Figure 1 Discretized region into triangular elements, the nodes are numbering from 1 to 25 and the triangles are numbering from 1 to 32 (This figure was taken from Chapra Book) [2]

3.2 Weak Formulation

In the next paragraphs the way of obtain the weak (or weighted integral) formulation is presented of a partial differential equation, as a second step of finite element method applied to single variable problems in two dimensions. It is worth to mention that this is a summarized description of the methodology, for further information can be queried [6] [5].

By last, this step is very useful and important in order to solve the partial differential equations, and it will be used as a non-automatized part if not as a mathematical procedure for the solution of the partial differential equation presented in this work. This summary has been made taking into account the theoretical framework from [6] and [5].

First, is assumed that Ω_e, as a typical element, it does not care if this is triangular or whatever else, in the mesh. Is taken the finite element model as follows:

$$-\frac{\partial}{\partial x}\left(a_{11}\frac{\partial u}{\partial x} + a_{12}\frac{\partial u}{\partial y}\right) - \frac{\partial}{\partial y}\left(a_{21}\frac{\partial u}{\partial x} + a_{22}\frac{\partial u}{\partial y}\right) - a_{00}u - f = 0 \quad \text{(8)}$$

For given data a_{ij} $(i,j = 1,2)$, a_{00} and f, and specified boundary conditions. There are three steps in order to get the weak form, as it can see at [6].

The first step is multiplying the equation (8) with a weight function w, which is assumed to be differentiable once with respect to x and y and then integrate the equation over the element domain Ω_e:

$$0 = \int_{\Omega_e} w\left[-\frac{\partial}{\partial x}(F_1) - \frac{\partial}{\partial x}(F_2) + a_{00}u - f\right] dx \, dy, \quad \text{(9a)}$$

, where

$$F_1 = a_{11}\frac{\partial u}{\partial x} + a_{12}\frac{\partial u}{\partial y} \quad , F_1 = a_{21}\frac{\partial u}{\partial x} + a_{22}\frac{\partial u}{\partial y} \tag{9b}$$

The second step is distributed the differentiation among u and w equally. To achieve this, is integrated the first two terms in the equation (9a) by parts.
Given the identities

$$\frac{\partial}{\partial x}(wF_1) = \frac{\partial w}{\partial x}F_1 + w\frac{\partial F_1}{\partial x} \; or -w\frac{\partial F_1}{\partial x} = \frac{\partial w}{\partial x}F_1 + \frac{\partial}{\partial x}(wF_1) \tag{10a}$$

$$\frac{\partial}{\partial y}(wF_1) = \frac{\partial w}{\partial y}F_1 + w\frac{\partial F_1}{\partial y} \; or -w\frac{\partial F_1}{\partial y} = \frac{\partial w}{\partial y}F_1 + \frac{\partial}{\partial y}(wF_1) \tag{10b}$$

Using the component form of the gradient (or divergence) theorem:

$$\int_{\Omega_e}\frac{\partial}{\partial x}(wF_1)\,dx\,dy = \oint_{\Gamma_e} wF_1 n_x ds, \tag{11a}$$

$$\int_{\Omega_e}\frac{\partial}{\partial y}(wF_2)\,dx\,dy = \oint_{\Gamma_e} wF_2 n_x ds \tag{11b}$$

Where n_x and n_y are the components (i.e. the direction cosines) of the unit normal vector

$$\hat{\boldsymbol{n}} = n_x\hat{\boldsymbol{\imath}} + n_y\hat{\boldsymbol{\jmath}} = \cos\alpha\,\hat{\boldsymbol{\imath}} + \sin\alpha\,\hat{\boldsymbol{\jmath}} \tag{12}$$

On the boundary Γ_e, and ds is the length of an infinitesimal line element along the boundary. By the use of the equations (10a), (10b), (11a), (11b) in (8) is obtained the following:

$$0 = \int_{\Omega_e}\left[\frac{\partial w}{\partial x}\left(a_{11}\frac{\partial u}{\partial x} + a_{12}\frac{\partial u}{\partial y}\right) + \frac{\partial w}{\partial y}\left(a_{21}\frac{\partial u}{\partial x} + a_{12}\frac{\partial u}{\partial y}\right) + \right.$$

$$\left. a_{00}wu - wf\right]dx\,dy - \oint_{\Gamma_e} w\left[n_x\left(a_{11}\frac{\partial u}{\partial x} + a_{12}\frac{\partial u}{\partial y}\right)\right.$$

$$+ \hat{n}_y$$

$$\left. n_y\,a_{11}\frac{\partial u}{\partial x} + a_{12}\frac{\partial u}{\partial y}\right]ds \tag{13}$$

Is noted that the specification of u constitutes the essential boundary condition, and hence u is the primary variable. The specification of the coefficient or the weight function in the boundary expression:

$$q_n \equiv n_x\left(a_{11}\frac{\partial u}{\partial x} + a_{12}\frac{\partial u}{\partial y}\right) + n_y\left(a_{11}\frac{\partial u}{\partial x} + a_{12}\frac{\partial u}{\partial y}\right) \tag{14}$$

Constitutes the natural boundary condition; thus q_n is the secondary variable of the formulation. The function $q_n = q_n(s)$ denotes the projection of the vector $\boldsymbol{a} \cdot \nabla u$ along the unit normal $\hat{n}$. By definition, q_s is taken positive outward form the surface as we move counterclockwise along the boundary (in most problems the secondary variable q_n is of physical interest).
The third and last step of the formulation is to use de definition (x) in (y) and write the weak form of the (xx) equation as:

$$0 = \int_{\Omega_e}\left[\frac{\partial w}{\partial x}\left(a_{11}\frac{\partial u}{\partial x} + a_{12}\frac{\partial u}{\partial y}\right) + \frac{\partial w}{\partial y}\left(a_{21}\frac{\partial u}{\partial x} + a_{12}\frac{\partial u}{\partial y}\right) + \right.$$

$$\left. a_{00}wu - wf\right]dx\,dy - \oint_{\Gamma_e} wq_n\,ds \tag{15}$$

And finally, the equation (15) is known as a weak formulation for the equation (8). Now is needed to form the element function for this last equation, and taking into account this formulation for use it with the element function.

3.2 Element Function

In this type of problem, it is chosen the linear equation in order to approximate the triangular element as follow is shown in Figure 2:
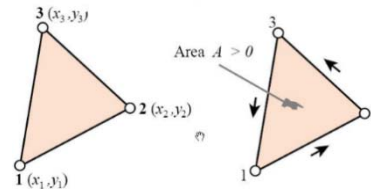

Figure 2 Triangular element shape and its area [3][7]

So, the equation in order to approximate this element is given by [2] [5] [6]:

$$u(x,y) = a_0 + a_{1,2}x + a_{1,2}y \tag{16}$$

Where the independent variable is x and y, and the coefficients given by the letters a. So due to the function must pass by the edges $(x_1,y_1), (x_2,y_2)$ and (x_3,y_3) of the triangle, it is formed by the next linear equations system [2][5][6]:

$$\begin{bmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \end{bmatrix}\begin{bmatrix} a_0 \\ a_1 \\ a_2 \end{bmatrix} = \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} \tag{17}$$

Therefore, it is obtained the follow results [2] [5][6]:

$$a_0 = \frac{1}{2A_e}[u_1(x_2y_3 - x_3y_2) + u_2(x_3y_1 - x_1y_3) + u_3(x_1y_2 - x_2y_1)] \tag{18a}$$

$$a_{1,1} = \frac{1}{2A_e}[u_1(y_2 - y_1) + u_2(y_3 - y_1) + u_3(y_1 - y_2)] \tag{18b}$$

$$a_{1,2} = \frac{1}{2A_e}[u_1(x_3 - x_2) + u_2(x_1 - x_3) + u_3(x_2 - x_1)] \tag{18c}$$

Where A_e is the area of the triangular element and it is calculated next [2] [5] :

$$A_e = \frac{1}{2}[(x_2y_3 - x_3y_2) + (x_3y_1 - x_1y_3) + (x_1y_2 - x_2y_1)]$$

In this way when the equations numbered as 18a, 18b and 18c are substituted into the equation number 16 (after of the common terms are grouped and vanished) it is expressed as follow [2][5]:

$$u = N_1 u_1 + N_2 u_2 + N_3 u_3 \qquad (19)$$

Where:

$$N_1 = \frac{1}{2A_e}\left[\left((x_2 y_3 - x_3 y_2) + (y_2 - y_3)x + (x_3 - x_2)y\right)\right.$$

$$N_2 = \frac{1}{2A_e}\left[(x_3 y_1 - x_1 y_3) + (y_3 - y_1)x + (x_1 - x_3)y\right.$$

$$N_3 = \frac{1}{2A_e}\left[(x_1 y_2 - x_2 y_1) + (y_1 - y_2)x + (x_2 - x_1)y\right]$$

It is notes that the equation number 19 is able to compute the intermediate values of the element and predict them using the values of the nodes [5] [6] [7].
Now taking in account this is possible to assembly the elements into a stiffness matrix in the next subsection.

3.3 Boundary Conditions and Assembly

This step is nearby to the mechanical problem raised, due to the matrix is assembled in accord with the geometry and topology of the problem. For this problem (the square or rectangular plate) specifically it is used the matrix method of stiffness [7].
This is a method accepted universally and it provides an algorithm fast and precise for the analysis of complex structures. The objective is to establish the relationship between the external charge and the displacement in the nodes of the structure. The current structure, that is continue, is substituted by the structure of discrete elements (finite), whose elastic properties and inertial are placed into the matrix. The blocks are built by the overlap of the derives, that they have been raised and the static and dynamic properties [6] [7].
Moreover, this method is based in the principles of the displacement and virtual force, the theorems of Castigliano and the theorem of minimum energy and complementary minimum energy and the theorem of displacement and unitary charge [7].
In this case the matrix, generally it is denoted as K, it is formed by elements of global element of size 3x3, which i is in the nodes. Each sub matrix $K_{i,i}$ in the main diagonal represents the stiffness of the node i or the sum of the all the matrixes of stiffness of elements that they start from i. The matrix K always is symmetric and if there are not elements between the nodes i and j, then the elements of the matrix $K_{i,j}$ is equal to zero. An example is shown in Figure 3 next [6] [7]:
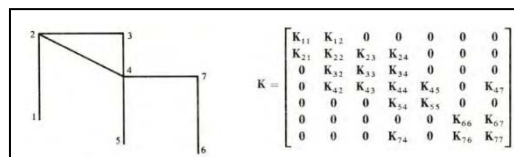

Figure 3. Complete model of the stiffness matrix of global stiffness rigid plane frame [7]

3.4 Analysis and Post Processing

The methods in order to solve this system are complex in many aspects; finally, they are a linear system of equations or simultaneous equations. The objective of solve this kind of systems for this case is to know the values of the independent variable in each one of the nodes [2][6][7]. Therefore, it is possible to study some iterative methods, given the nature of the matrix (huge size, but symmetric and tridiagonal), in order to know what the best numerical method for resolve this linear equation system [2][6][7].
Now it is possible to apply the Finite Element Method to the problem raised in this paper.

4. RESULTS

Taking into account the above mentioned in the past section, it was instrumented a solution by the use of Octave due to contains very much primitives in order to express and management matrixes, vectors, among very much other functions really powerful.
Moreover, is a free software resource and it has a potent programming language very similar to Matlab and with the additional advantage of be multiplatform.
The implementation of the solution was made considering the number of divisions of the sides of the plate. Given this, a square plate with 2 per side is used and with the right side considered with the parameters expressed in the section 2.
Moreover, the program was tested with the division equals in both sides x and y, with n=4, 8, 16 and 32 divisions per side.
The results obtained in this paper were compared with the results shown in Chapra's Book [2], in all tables it is find the error rate with respect these results.
First it is shown in the Table 2 the results of the implementation n=4 and the absolute and relative error (rate of error).

X	Y	FEM Z	Chapra	Absolute Error	Relative Error
0.5	0.5	0.0627	0.063	0.000300	0.48%
1	0.5	0.086	0.086	0.000010	0.01%
1.5	0.5	0.0627	0.063	0.000300	0.48%
0.5	1	0.086	0.086	0.000010	0.01%
1	1	0.1182	0.118	0.000240	0.20%
1.5	1	0.086	0.086	0.000010	0.01%
0.5	1.5	0.0627	0.063	0.000300	0.48%
1	1.5	0.086	0.086	0.000010	0.01%
1.5	1.5	0.0627	0.063	0.000300	0.48%

Table 2. The results omitting the boundary coordinates either that in this point are zero for n=4 divisions.

As follows, the Table 3 consider the average of the rates of error for each one of the number of divisions:

Number of Divisions	Average of Error's Rate
4	0.23940704
8	0.182579078
16	0.021875766
32	0.00220650

Table 3. The average of the rates of error by divisions

As it is can observe in the Table 3, the solution obtained by Finite Element method has a very good rate of error. Now in order to proof the graphical results it shows them next the comparison between graphics, in the left side the graphic of the solution obtained by Finite Element method and in the other side the graphic of the solution obtained by [2].

As it is can observe in the Table 3, the solution obtained by Finite Element method has a very good rate of error. Now in order to proof the graphical results it shows them next the comparison between graphics, in the left side the graphic of the solution obtained by Finite Element method and in the other side the graphic of the solution obtained by [2].

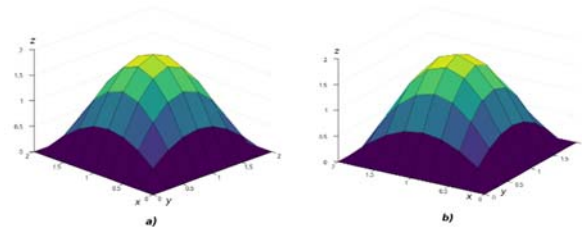**Figure 4:** Graphical results obtained with *n*= 4 divisions by a)finite element b) exact solution (Scaled to 16 elements)

Figure 5: Graphical results obtained with *n*= 8 divisions by a) finite element b) exact solution (Scaled to 64 elements)
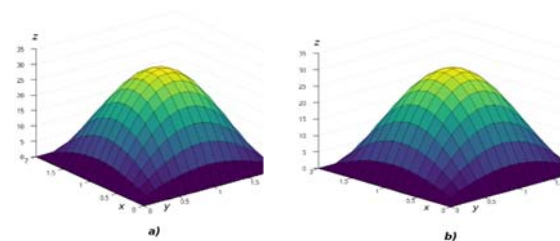

Figure 6: Graphical results obtained with *n*= 16 divisions by a) finite element b) exact solution (Scaled to 256 elements)
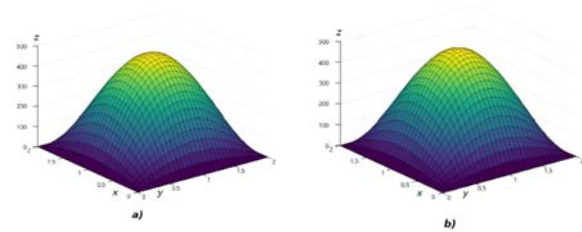

Figure 7: Graphical results obtained with *n*= 32 divisions by a) Finite element b) exact solution (Scaled to 512 elements)

Due to the great degree of precision of the calculation made by the finite element method, the graphics are very similar, as it can observe in the above figures.

5. CONCLUSIONS

The numerical solution of the Partial Differential Equation Bilaplacian type by the finite element method was presented with the objective to simulate the accelerometer-type MEMS behavior. The method of the finite element method used resulted suitable for this kind of problem, due to obtained very good rates of error. The partial differential equation Bilaplacian-type was resolved by the application of the numerical method and this method was described in general way. The solution to the problem was instrumented in Octave that it was resulted be a powerful tool and that it contains potent libraries, primitives in order to handle and management matrixes easily.

The next step for graphical results obtained is rendering with a control of the user and get an interpretation of the graphical results, for example, by the use of color code.

In future works it is possible to apply this method to another type of problems, and look for parallelize some parts of this method. In spite of the complexity and size memory used by the numerical method is possible looking for to experiment with another methods in order to solve the huge linear equations systems.

6. ACKNOWLEDGEMENTS

7. REFERENCES

[1] Alanís Urquieta, Sánchez Flores, Morales Riveroll, López Mendez, Bermúdez Juárez, **Experimental Comparison of the Implementation of MVC in Java and C#,** Proceedings of The 21st World Multi-Conference on Systemics, Cybernetics and Informatics (WMSCI 2017), IIIS, Florida, USA.

[2] Chapra, Steven C., Canale, Raymond P., del Valle Sotelo, Juan Carlos, **Métodos Numéricos para Ingenieros**, 5th Edition, McGraw Hill, Mexico, 2007

[3] Olver, J.L., **Elementos Finitos, Formulación de Elementos Finitos Lección 4: Elemento Triangular Lineal**, Universidad Politécnica de Valencia, España, 2005.

[4] Pelesko, John A., Berstein, David H., **Modeling MEMS and NEMS**, CRC Press, USA, 2002.

[5] Pepper, Darrell W., Henrich, Juan C., **The Finite Element Method, Basic Concepts and Applications with Matlab, Maple and COMSOL**, 3rd. Edition, CRC Press, Taylor and Francis Group

[6] Reddy, J.N., **Introduction to the Finite Element Method**, Third Edition (McGraw-Hill Education, 2006).

[7] Saravia Aramayo, Roger Gustavo, **Modelo Orientado a Objetos Independiente de la Plataforma para el Análisis de Estructuras Planas mediante el método de los Elementos Finitos**, Thesis of Master in Software Engineering, Bolivia, 2013