The Modal Superposition Method Using Maple®: A Structural Dynamics Application for the Classroom

Carlos E. Orozco, Ph.D.
Associate Professor
Department of Engineering Technology
University of North Carolina at Charlotte
Charlotte, NC 28223
ceorozco@uncc.edu

Abstract
The objective of the present paper is to illustrate the use of the commercially available mathematical package MAPLE® and other computer software for educational purposes. The problem chosen to illustrate the use of the software is the problem of forced vibrations of an elastic plate. The presentation of the topic is made in a simple manner to make it suitable for introductory graduate or undergraduate structural dynamics or vibration courses. The paper illustrates many concepts of structural dynamics in a step by step structured way: from the mathematical formulation, to the generation of a computer animation. The organization of the material and its computer implementation facilitate the introduction of the topic to students without having to resort to black-box type commercial finite element packages like ANSYS®, SAP2000®, RISA®, etc. A very simple finite element (FE) is used to model the plate. The simplicity of the FE model used makes it possible to implement the procedure using high-level computer languages like those available within MATLAB®, MAPLE®, or even EXCEL®. The classical modal superposition method is used to illustrate the concepts. A standard finite element procedure in conjunction with Jacobi's method is used to solve the eigenvalue problem. A sample numerical problem together with snapshots of the computer animation are presented.

Keywords
Maple®, vibration analysis, modal superposition, plates, finite elements.

1. Introduction

The problem of the forced vibrations of an elastic plate involves a variety of concepts of structural analysis and structural dynamics. For the analysis of simple, skeletal structures, some knowledge of the direct stiffness method may be sufficient. For the analysis of more complex, continuous structures, knowledge of the finite element method (FEM) is necessary. As a matter of fact, accurate solutions of the plate problem require finite element discretizations using complex plate or shell elements. These elements are readily available in commercial black-box type computer codes like ANSYS®, SAP2000®, RISA®, etc. One of the disadvantages of using black-box type packages in an instructional setting is that students miss many important details of the calculation procedure. As an alternative to the use of commercial FE packages, simple FE formulations of the problem like the one presented here provide a valuable alternative. The present FE formulation uses two degree-of-freedom finite elements to approximate the linear elastic behavior of the plate. This formulation can be easily implemented using high-level computer languages like those available within MATLAB®, MAPLE®, or even EXCEL®. This formulation will probably not be accurate enough for industrial applications. However, it captures all the important aspects of the problem from the academic point of view. In addition, because of its simplicity, it allows for the generation of a realistic animation of the vibrations of the plate using a readily available computer program such as MAPLE®.

Three methods are generally available to perform a numerical simulation of the forced vibrations of an elastic plate: 1. modal superposition, 2. frequency domain analysis, and 3. direct numerical integration of the differential equation of motion. Method 2. requires knowledge of Fourier transforms and Fourier series; it is mathematically more involved than the other two and probably not appropriate for an introductory course in vibrations. Method 3. is the method of choice when the modal superposition method cannot be used (i.e., when the structure exhibits nonlinear behavior, see e.g.,(Craig, 1981)). Method 1. on the other hand, is a method that can be used in many practical situations and that is rich in educational and practical content. The present discussion will therefore use method 1., or modal superposition. When modal superposition is used, the numerical solution of the problem of the forced vibrations of an elastic structure involves the following steps: 1. discretization of the problem using a suitable finite element model, 2. formulation of the corresponding eigenvalue problem, 3. solution of the eigenvalue problem, 4. determination of the normal equations of motion (uncoupling of the equations of motion), 5. calculation of the response to the given excitation for each of the normal coordinates, and 6. superposition of the normal response functions to obtain the response of the structure. The remaining of this paper describes the foregoing steps in some detail. A more complete account of the entire process can be found in (Orozco, 2005).

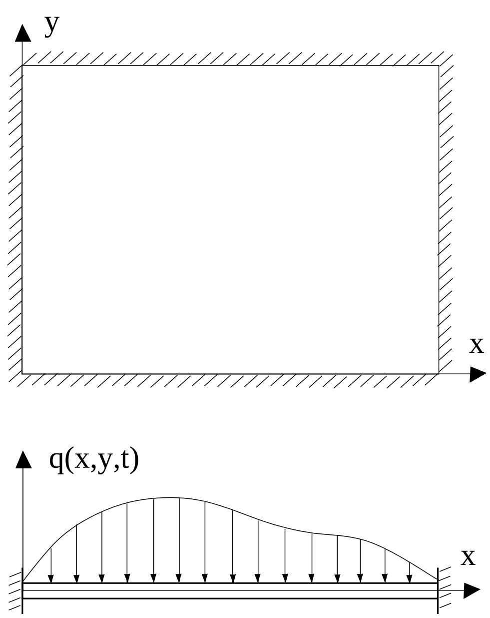

Figure 1. General, time-dependent load for a plate.

2. Mathematical Description of the Problem

Figure 1 shows a plan view of a rectangular plate together with a general, time dependent, transverse (i.e., perpendicular to the plane of the plate) load *q(x,y,t).* The simplest problem that can be defined for this plate consists of finding the vertical deflection or displacement field *u(x,y,t).* In the context of dynamics, this is referred to as the "response" of the system. In a rigorous mathematical context, this problem is a continuum problem. In other words, it has an infinite number of degrees of freedom (DOF). Its exact analytical solution requires the use of partial differential equations and Fourier series approximations.
Even the static problem (i.e., one in which the time coordinate *t* has been removed from the equations), requires the use of partial differential equations. A much more practical approach to the solution of this problem, consists of using a numerical approximation procedure like the finite element method (FEM).

An accurate model of the plate problem using the FEM requires the use of complex elements with many degrees of freedom. These elements are available in commercial FE packages. A much simpler finite element model was adopted here for the sake of educational clarity. Figure 2 illustrates schematically this FE model. It shows schematically the plate modeled as a grid of beam elements with two degrees of freedom.

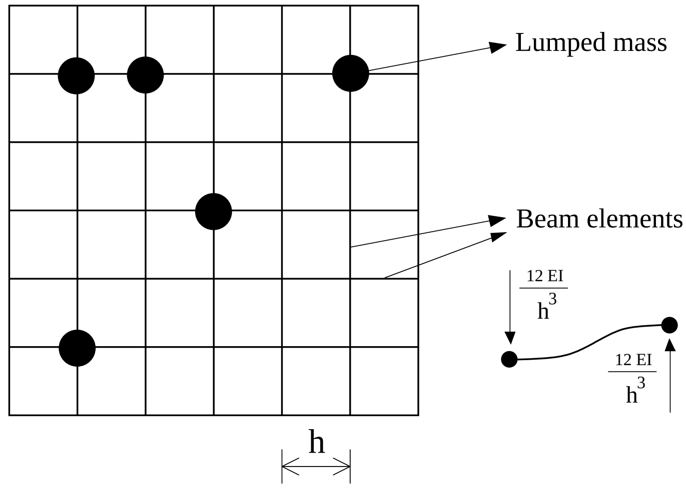

Figure 2. Two degree-of-freedom finite elements to approximately model the plate.

3. A Simplified Finite Element Model for a Plate

The finite element model illustrated in Figure 2 uses a 2-DOF finite element that leads to the following elemental stiffness matrix:

$$\mathbf{K}_e = \begin{bmatrix} \dfrac{12EI}{L^3} & -\dfrac{12EI}{L^3} \\ -\dfrac{12EI}{L^3} & \dfrac{12EI}{L^3} \end{bmatrix} \tag{1}$$

A global stiffness matrix **K** for the entire plate structure can now be assembled using the standard steps of the direct stiffness method (see e.g., (Chandrupatla and Belegundu, 1997)). For the case of a static load, the global equilibrium equations of the structure can be formally written as:

$$\mathbf{K}\,\mathbf{u} = \mathbf{P} \tag{2}$$

where **u** is the displacement vector, and **P**, the load vector. For the case of the plate, the displacement vector will contain the vertical displacements of the plate at the nodal points (see Figure 2). The simplest way to account for a distributed load like that shown in Figure 1 will be to add the contributions of four adjoining tributary areas and assign them to the node at the center of the areas as a nodal load. This procedure is referred to as the *lump* load method. The load vector will then contain these nodal loads. Once the plate is discretized in this way, its dynamical behavior can be mathematically described with the help of a Multi-Degree-Of-Freedom (MDOF) system like that shown in Figure 3b.

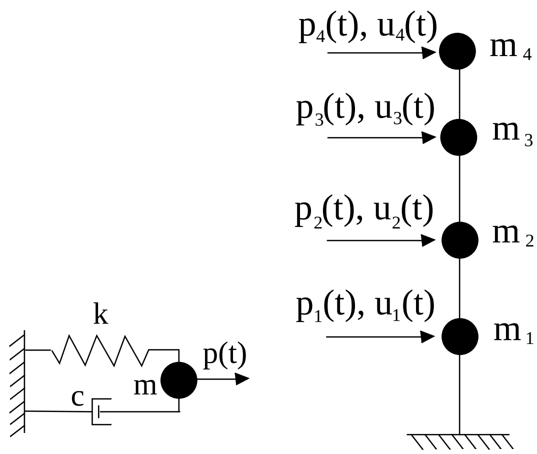

Figure 3. a) SDOF System. b) MDOF System.

4. Dynamics of Multi-Degree-Of-Freedom (MDOF) Systems

Figure 3 illustrates a single degree of freedom (SDOF) dynamical system together with a one dimensional multi-degree-of-freedom system. A simple application of D'Alambert's Principle to the SDOF system leads to the well know dynamic equilibrium equation:

$$m\ddot{u}(t) + c\dot{u}(t) + ku(t) = p(t) \qquad (3)$$

where *m* is the mass of the system, *k* the spring constant, *c* the damping constant, *u(t)* the displacement in the horizontal direction, and *p(t)* the applied load. For the MDOF system (Figure 3b), the dynamic equilibrium equation looks formally the same as Equation (3):

$$\mathbf{M}\,\ddot{\mathbf{u}}(t) + \mathbf{C}\,\dot{\mathbf{u}}(t) + \mathbf{K}\,\mathbf{u}(t) = \mathbf{p}(t) \qquad (4)$$

The difference is of course that in Equation (4), **M**, **C**, and **K** are matrices, and **u** and **p** are vectors. Notice also that **u** and **p** are functions of time. To solve Equation (4), it is convenient to first consider the problem of free vibrations of an undamped system, i.e., one in which **C** and **p** are zero. Equation (4) then becomes:

$$\mathbf{M}\,\ddot{\mathbf{u}}(t) + \mathbf{K}\,\mathbf{u}(t) = \mathbf{0} \qquad (5)$$

If a simple harmonic solution $\mathbf{u}(t) = \mathbf{u_0} \sin(\omega t + \theta)$ for (5) is assumed, the following eigenvalue problem is obtained:

$$\mathbf{E}\mathbf{u_0} = \omega^2\, \mathbf{u_0} \qquad (6)$$

where $\mathbf{E} \equiv \mathbf{M}^{-1}\,\mathbf{K}$ is the inverse of the so-called *dynamic matrix* (see e.g., (Clough and Penzien, 1975)). If *n* is the number of degrees of freedom of the system (i.e., the number of nodes in the plate model shown in Figure 2), there are *n* eigenvalues ω_i and *n* eigenvectors $\boldsymbol{\varphi}_i$ that constitute the solution of (6). The *n* eigenvalues ω_i are referred to as the natural frequencies of the system. The *n* eigenvectors $\boldsymbol{\varphi}_i$ are known as the modes of vibration (or modal shapes) of the system. The smallest frequency of the system is referred to as the *fundamental* frequency of the system. The smallest frequency corresponds to the largest period

of vibration. If the frequencies are ordered in ascending order, the fundamental frequency corresponds to ω_1 and the fundamental period to T_1. These frequency and period are related by:

$$T_1 = \frac{2\pi}{\omega_1} \tag{7}$$

An important characteristic of the modes of vibration of a dynamical system is that they are orthogonal to each other with respect of the mass **M** and stiffness **K** matrices (see, e.g., Bathe, 1996). Another important property of the modes of vibration of a dynamical system is that they constitute a basis for the *n*-dimensional space (see, e.g., Bathe, 1996). This property can be used to advantage to solve Equation (4). For this purpose, let **u(**t**)** be the solution of (4). Then **u(**t**)** can be written as a linear combination of the *n* modal shapes $\boldsymbol{\varphi}_i$ as follows:

$$\mathbf{u}(t) = \boldsymbol{\Phi}\, \mathbf{y}(t) \tag{8}$$

where $\boldsymbol{\Phi}$ is a matrix whose columns are the modes of vibration $\boldsymbol{\varphi}_i$. When Equation (8) is substituted into (4), and use is made of the orthogonality properties, a scalar equation for each mode of vibration *i* is obtained as follows:

$$m_i^* \ddot{y}_i(t) + c_i^* \dot{y}_i(t) + k_i^* y_i(t) = p_i^*(t) \tag{9}$$

where,

$$m_i^* \equiv \boldsymbol{\varphi}_i^T\, \mathbf{M}\, \boldsymbol{\varphi}_i \tag{10}$$

$$c_i^* \equiv \boldsymbol{\varphi}_i^T\, \mathbf{C}\, \boldsymbol{\varphi}_i \tag{11}$$

$$k_i^* \equiv \boldsymbol{\varphi}_i^T\, \mathbf{K}\, \boldsymbol{\varphi}_i \tag{12}$$

$$p_i^*(t) \equiv \boldsymbol{\varphi}_i^T\, \mathbf{p}(t) \tag{13}$$

The quantities in Equations (10) to (13) are known as the *generalized mass*, *generalized damping constant*, *generalized stiffness*, and *generalized load* of the structure respectively.

Equation (8) can also be written in a more convenient form as:

$$\ddot{y}_i(t) + 2\xi_i \omega_i \dot{y}_i(t) + \omega_i^2 y_i(t) = \frac{p_i^*(t)}{m_i^*} \tag{14}$$

where,

$$\xi_i \equiv \frac{c_i^*}{2 m_i^* \omega_i} \tag{15}$$

is the damping ratio, and

$$\omega_i \equiv \sqrt{\frac{k_i^*}{m_i^*}} \tag{16}$$

is the circular frequency corresponding to mode *i*.

The foregoing process shows that Equation (4) can be transformed into a system of *n* equations, one for each mode of vibration (Equation (9)). This means that the modes of vibration effectively *uncouple* the dynamic equations of motion. In other words, to solve a MDOF dynamics problem, it suffices to solve *n* separate (uncoupled) differential equations of the form (9) as opposed to having to solve a system of coupled differential equations (Equation (4)). This is the advantage of using the method of modal superposition to solve dynamic problems.

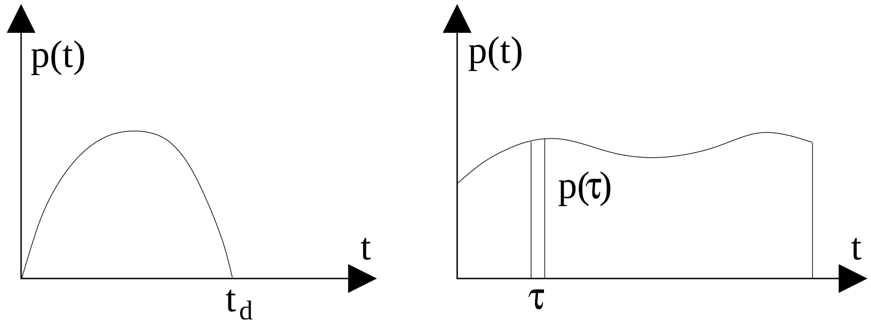

Figure 4. a) Impulse load. b) General dynamic load.

4. Dynamic Response to an Impulsive Load

One of the most common loads in structural systems is an impulsive load, i.e., a load that has a short duration with respect to the fundamental period of the structure (i.e., $t_d << T$). A schematic plot of such load is shown in Figure 4a. The *impulse* of this load is by definition:

$$I = \int_0^{t_d} p(t)dt \tag{17}$$

It can be shown that the response of a SDOF system to this impulse load is given by:

$$y(t) = \frac{I}{m\omega_d} e^{-\xi\omega t} \sin\omega_d t \tag{18}$$

where,

$$\omega_d \equiv \omega\sqrt{1-\xi^2} \tag{19}$$

is the *damped* frequency of the system (see, e.g., (Clough and Penzien, 1975)).

In the case of a MDOF system, there will be one equation like (18) for each uncoupled degree of freedom *i*, i.e.,

$$y_i(t) = \frac{I_i}{m_i^* \omega_{di}} e^{-\xi_i\omega_i t} \sin\omega_{di} t \tag{20}$$

Following the definition of the generalized force for mode *i* given by (13), and the definition of impulse given in (17), the impulse for mode of vibration *i*, is given by:

$$I_i = \int_0^{t_d} \boldsymbol{\varphi}_\mathrm{i}^\mathrm{T}\, \mathbf{p}(\mathrm{t})\ dt \tag{21}$$

5. Computational Aspects.

The FE program used in the present study is a research FE code written in FORTRAN. The input to the FE program consists of the geometry of the structure, the element properties, the element connectivity, and the boundary conditions (including loads). It also includes the data necessary to perform a dynamical analysis, i.e., the individual element masses. Damping is introduced into the system by means of individual damping ratios for each mode (see Equations (14) and (15)). The direct stiffness method of analysis is used to assemble the stiffness and mass matrices. The eigenvalue problem given by (6) is solved by means of Jacobi's method (Press et al, 1992).

5.1 Response Calculations

The output of the FE program consists of the modes of vibration of the plate and the corresponding modal frequencies. These data constitute the input to a "response" program. The "response" program uses the modal superposition method to generate a discrete version of the response function $u(x,y,t)$ mentioned in Section 2. above. The main component of the process is Equation (20) which gives the response of a SDOF system to an impulsive load. In the context of a MDOF system, each response function $y_i(t)$ is also the coefficient of φ_i in Equation (8). The process of generating the response of the structure is carried out in an incremental fashion using time steps. A suitable interval of time Δt is first chosen, the response calculated for time t, then for time $t+ \Delta t$, and so on. The first step is to calculate the impulse I_i corresponding to each mode of vibration. This is done using Equation (21). The calculations performed by the response code can be arranged in the form of an algorithm as follows: 1. set the problem parameters and initialize variables: set the duration of the impact (impulse) load to a small number, say $t_d = \dfrac{T_1}{1000}$; set the damping coefficient ξ_i (say ξ_i = 2 %); choose the time interval Δt and set the initial time to zero: $\Delta t = \dfrac{Duration\_of\_Simulation}{Number\_of\_Time\_Intervals}$; $t=0$. 2. Increment t: $t = t + \Delta t$. 3. For all x and y coordinates set $frame(x,y) = 0.0$. 4. Do For i=1 to nmodes; calculate m_i^* according to Equation (10) and ω_{di} according to Equation (19); calculate $I_i = P_0 * t_d * \varphi(x_p,y_p,i)$. Calculate y_i according to Equation (20). Superimpose mode contributions into the array frame: $frame(x,y)=frame(x,y)+y_i*\varphi(x,y,i)$. Write $frame(x,y)$. Enddo. 5. If $t \leq Duration\_of\_Simulation$ then goto 2, otherwise STOP. In this algorithm, nmodes refers to the number of modes considered for the simulation, which in general does not have to be equal to n. P_0 is the intensity of the applied load. The coordinates x_p and y_p refer to the point of application of the impact load. This position can be changed within the program "response". As shown in step 4. above, the "response" program outputs the animation frames to a file called "maplemodes.dat". The format of this file must be such that it can be understood by MAPLE®. This is accomplished by formatting this file according to the internal representational structure that MAPLE® uses for its plot3d function (this data structure will be displayed by MAPLE® when the command "?plot3d[structure];" is typed in the command line). The plot3d function is one of the functions that MAPLE® uses to produce animations. This function is used when the values of the surface to be animated (in this case the deflected shape of the plate) are specified by the user. Following this structure, the first line of the "maplemodes.dat" should read something like: animation:= PLOT3D(GRID(0.0..1.0, 0.0..1.0 [0.0,…,0.0], [0.0, -3.9,…,-3.9, 0.0], …The "maplemodes.dat" file is read by MAPLE® by means of the command 'read "maplemodes.dat";'. Once the file is read, the actual animation can be generated and displayed by simply typing 'animation;' in the command line.

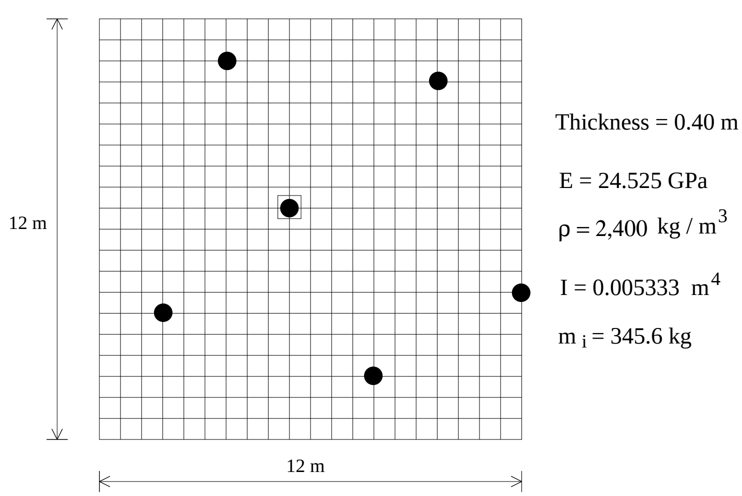

Figure 5. Plate dimensions and data for example problem.

6. Sample Results.

The data for the sample problem presented here is shown in Figure 5. The plate was discretized using 840 elements and 441 nodes. The lumped mass m_i at each node corresponds to the mass of a 0.6 m × 0.6 m (× 0.4 m) square (parallelepiped) that is centered at node *i* as indicated in Figure 5. For the results presented here, the impact load was applied at the center node of the plate. Note that as far as the smulation is concerned, the actual value of the load is immaterial since the deflection of the plate needs to be exaggerated (scaled) for purposes of graphical representation. The time increment used was $\Delta t = 10^{-4}$ seconds. Figure 6 illustrates a sequence of four frames obtained from the animation produced by MAPLE®. The number of modes considered for the modal superposition in the animation presented here was 100 (out of a maximum of 441 for the current discretization of the plate). Figure 7 shows modes of vibration 1, 4, 12, and 55. These modes were also obtained using the program "response". The speed of the computer simulation can be adjusted at will. However, it should be noted that the actual duration of the vibration phenomenon is very short. For instance, the interval of time that separates frames 5 and 12 in Figure 6 is actually $7\Delta t$ or 7×10^{-4} seconds.

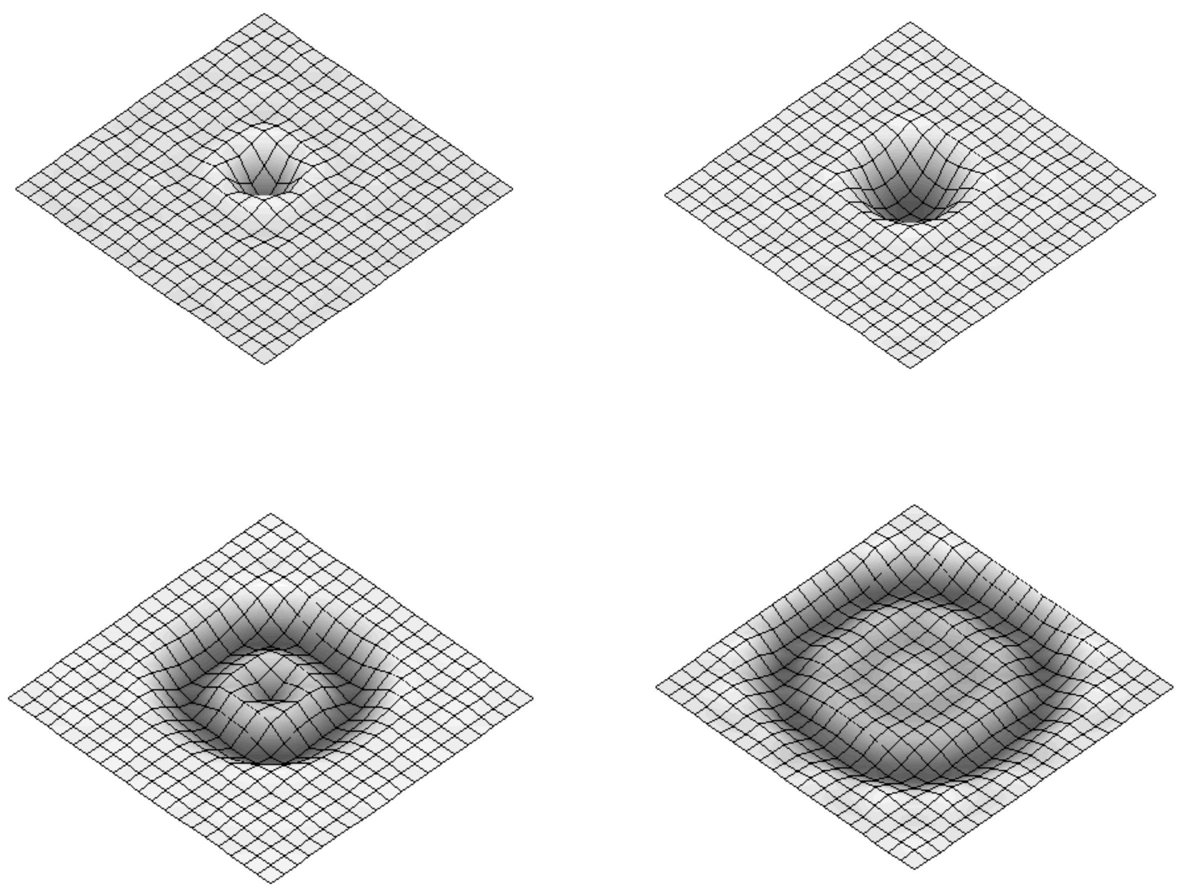

Figure 6. Clockwise From Top to Bottom: Frames 1, 5, 12, and 18 of computer animation.

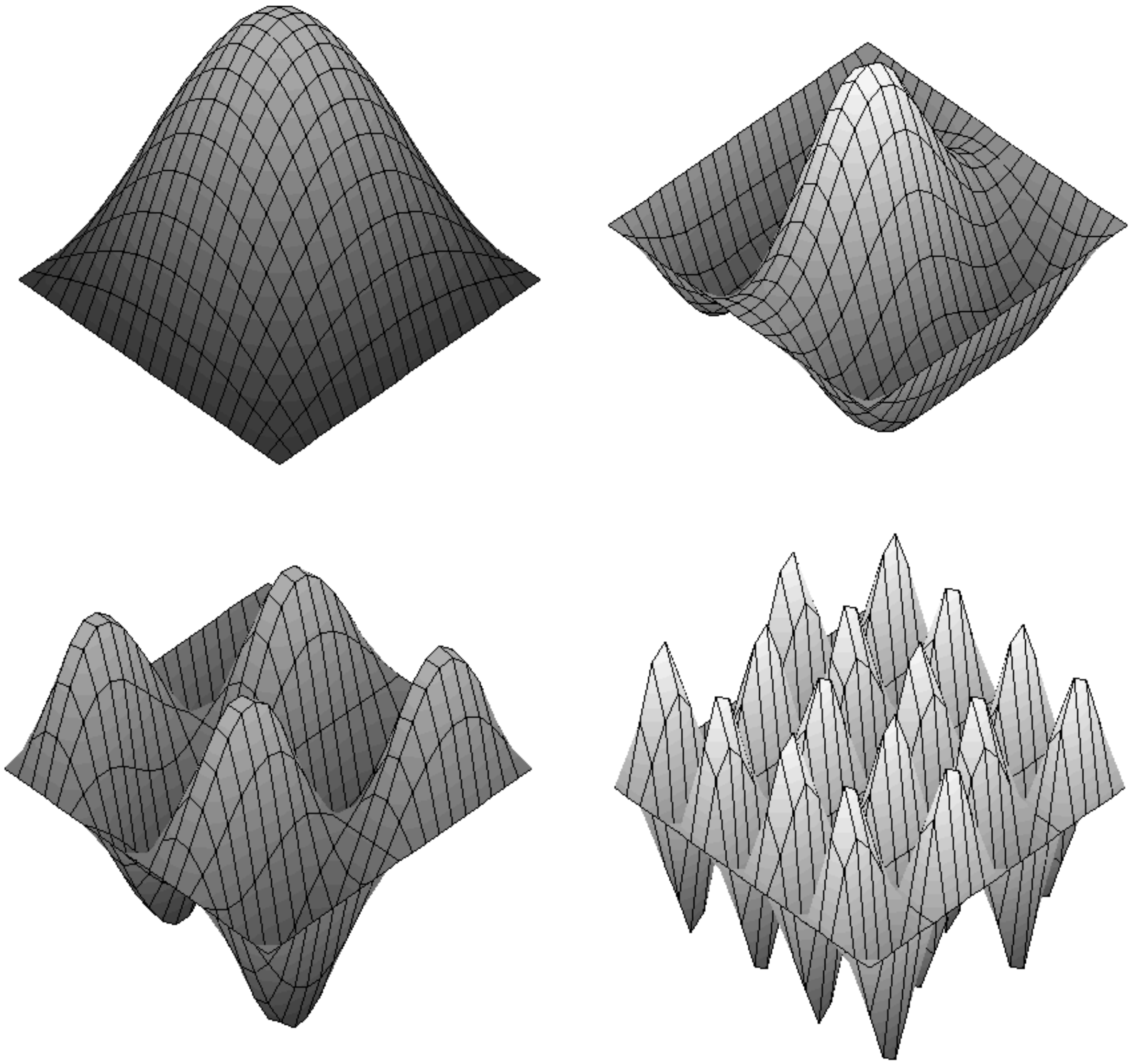

Figure 7. Clockwise From Top to Bottom: Modes of Vibration 1, 4, 12, and 55.

7. Concluding Remarks.

A simple formulation of the problem of forced vibrations of an elastic plate has been presented together with sample results and snapshots of a computer animation. The plate is discretized using a simple finite element. The eigenvalue problem resulting from the finite element analysis is solved using Jacobi's method. The response of the plate is determined using the modal superposition method. The simplicity of the finite element model used allows for a computer implementation of the problem that permits the generation of an animation of the vibrations of the plate using MAPLE®. The problem is presented in a detailed yet modular way so that it can be easily incorporated into lecture material for introductory graduate or undergraduate courses in structural dynamics or vibrations. The presentation offers an alternative to the use of black-box commercial computer codes to study the topic of forced vibrations of elastic structures in a classroom environment.

References

[1] Orozco, C. E. "Simulating the Forced Vibrations of an Elastic Plate Using a Simple Finite Element and Maple®". *Computers in Education Journal*, Vol. 15, No. 1, 2005

[2] Craig R. R. "*Structural Dynamics: An Introduction to Computer Methods*". Wiley & Sons, New York, 1981, Ch. 18.

[3] Chandrupatla, T. R. and Belegundu, A. D. "*Introduction to Finite Elements in Engineering*". Second Edition. Prentice Hall, Upper Saddle River, New Jersey, 1997, Ch. 8.

[4] Clough R. W. and Penzien J. "*Dynamics of Structures*". McGraw-Hill, New York, 1975.

[5] Bathe K-J. "*Finite Element Procedures*". Prentice Hall, Englewood Cliffs, 1996, Ch. 11.

[6] Press W. H., Teukolsky S. A., Vetterling W. T., and Flannery B. P. "*Numerical Recipes in FORTRAN*". Second Edition, Cambridge University Press, Cambridge, 1992. p. 460.

[7] Press W. H., Teukolsky S. A., Vetterling W. T., and Flannery B. P. "*Numerical Recipes Example Book (FORTRAN)*". Second Edition, Cambridge University Press, Cambridge, 1992. Ch. 11

Authorization and Disclaimer