

SIMULATING THE FORCED VIBRATIONS OF AN ELASTIC PLATE USING A SIMPLE FINITE ELEMENT AND MAPLE[®]

Carlos E. Orozco
Department of Engineering Technology
University of North Carolina at Charlotte
Charlotte, NC 28223

ABSTRACT

A complete formulation of the problem of forced vibrations of an elastic plate using the modal superposition method and a simple finite element is presented. An earlier version of this problem was presented as a project for a "Summer Institute on Supercomputing" at the Pittsburgh Supercomputing Center in Pittsburgh, Pennsylvania. The objective of the project was to demonstrate the use of massively parallel techniques in connection with practical engineering problems. The presentation here has been stripped from all references to highly parallel computation to make the problem usable in introductory graduate or undergraduate structural dynamics or vibrations courses. The problem illustrates many concepts of structural dynamics in a structured way and shows all the steps required to solve it: from the mathematical formulation, to the generation of a computer animation using the commercially available mathematical package MAPLE[®]. The particular organization of the material facilitates the introduction of the topic to graduate or undergraduate engineering students without having to resort to black-box type commercial finite element packages.

INTRODUCTION

Due to the inherent richness and complexity of the subject matter, it is difficult to find complete worked-out solutions of dynamics problems in the literature that are both simple enough for the classroom and sufficiently realistic to be of practical value. The purpose of this paper is to present a practical vibration analysis problem in a complete yet "classroom friendly" way.

The problem of the forced vibrations of an elastic plate involves many concepts of structural analysis and structural dynamics. For simple, skeletal structures, some knowledge of the direct stiffness method of analysis is sufficient. Other more complex, continuous structures require knowledge of the finite element method (FEM). As a matter of fact, accurate solutions of the plate problem require finite element (F.E.) discretizations using complex plate or shell elements. These elements are readily available in commercial black-box type computer codes like ANSYS[®], SAP2000[®], RISA[®], etc. The problem with using these black-box type packages in an instructional setting is that students usually miss important details of the calculations involved. As an alternative to the use of commercial F.E. packages, simple F.E. formulations of the problem like the one presented here provide a valuable alternative. The present F.E. 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[®]. It should be noted that this model will probably not be accurate enough for industrial applications. However, the resulting calculation procedure captures all the important aspects of the F.E. method when applied to dynamic problems. In addition, because of its simplicity, the calculation procedure allows for the generation of a realistic animation of the vibrations of the plate using the commercial computer package MAPLE[®]. The simplicity of the formulation also allows for a modular arrangement of the entire procedure that

facilitates the identification and execution of all the steps in a logical fashion.

In general, a numerical simulation of the forced vibrations of an elastic plate can be performed using one of the following three methods:

1. Modal superposition
2. Frequency domain analysis
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 setting. Method 3. is the method of choice when the modal superposition method cannot be used (like when the structure exhibits nonlinear behavior. See e.g.,[1]). Method 1. on the other hand, is a method that can be used in many practical situations and that is rich in terms of academic and practical content. The present discussion will therefore use method 1., or modal superposition. When this method 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.
6. Superposition of the normal response functions to obtain the response of the structure.

The remaining of this paper will discuss the foregoing steps in detail.

PROBLEM DESCRIPTION.

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 application of the theory of partial differential equations. A much more practical approach to the solution of this problem from the engineering point of view, 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 are the elements that commercial packages use. Since the objective of the current presentation is educational, a much simpler finite element model was adopted. Figure 2 illustrates schematically this finite element model. In this model, the plate is modeled as a grid of beam elements with just two degrees of freedom. This model allows for a realistic representation of the vibrations of the plate in the form of an animation. It also allows for the illustration of all the concepts related to the application of the modal superposition method to structural problems. It must be emphasized again however, that this model will not be accurate enough for industrial applications.

A SIMPLIFIED FINITE ELEMENT MODEL FOR A PLATE

The finite element model of the plate is illustrated in Figure 2. It uses a 2-DOF finite

element that leads to the following element stiffness matrix:

$$\mathbf{K}_e = \begin{bmatrix} \frac{12EI}{L^3} & -\frac{12EI}{L^3} \\ -\frac{12EI}{L^3} & \frac{12EI}{L^3} \end{bmatrix} \quad (1)$$

A global stiffness matrix \mathbf{K} for the entire plate structure can now be assembled using the standard steps of the direct stiffness method (See e.g., [2]). For the case of a static load, the global equilibrium equations of the structure can be formally written as:

$$\mathbf{K} \mathbf{u} = \mathbf{P} \quad (2)$$

where \mathbf{u} is the displacement vector, and \mathbf{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 is 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.

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'Alembert's Principle to the SDOF system leads to the well known dynamic equilibrium equation:

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

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):

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

The difference is of course that now \mathbf{M} , \mathbf{C} , and \mathbf{K} are matrices, and \mathbf{u} and \mathbf{p} are vectors. Notice also that \mathbf{u} and \mathbf{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 \mathbf{C} and \mathbf{p} are zero. Equation (4) then becomes:

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

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:

$$\mathbf{E} \mathbf{u}_0 = \omega^2 \mathbf{u}_0 \quad (6)$$

where $\mathbf{E} \equiv \mathbf{M}^{-1} \mathbf{K}$ is the inverse of the so-called *dynamic matrix* [3]. 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 $\boldsymbol{\phi}_i$ that constitute the solution of (6). The n eigenvalues ω_i are referred to as the natural frequencies of the system. The n eigenvectors $\boldsymbol{\phi}_i$ 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:

$$T_1 = \frac{2\pi}{\omega_1} \quad (7)$$

An important characteristic of the modes of vibration of a dynamical system is that they are orthogonal to each other with respect to the mass \mathbf{M} and stiffness \mathbf{K} matrices (for repeated eigenvalues, the modes are not orthogonal but can be constructed to be so by using a suitable orthogonalization procedure like Gram-Schmidt. See e.g., [4]). The mathematical statements of

the orthogonality properties of the modes of vibration are:

$$\boldsymbol{\varphi}_i^T \mathbf{M} \boldsymbol{\varphi}_j = 0, \quad i \neq j \quad (8)$$

$$\boldsymbol{\varphi}_i^T \mathbf{K} \boldsymbol{\varphi}_j = 0, \quad i \neq j \quad (9)$$

Another important property of the modes of vibration of a dynamical system is that they constitute a base for the n -dimensional space. This property can be used to advantage to solve Equation (4) as explained next. Let $\mathbf{u}(t)$ be the solution of (4). Then $\mathbf{u}(t)$ can be written as a linear combination of the n modal shapes $\boldsymbol{\varphi}_i$ as follows:

$$\mathbf{u}(t) = \boldsymbol{\Phi} \mathbf{y}(t) \quad (10)$$

where $\boldsymbol{\Phi}$ is a matrix whose columns are the modes of vibration $\boldsymbol{\varphi}_i$. By substituting (10) in (4), pre-multiplying this result by $\boldsymbol{\varphi}_i^T$ and using the orthogonality conditions (8) and (9), the following scalar equation for mode of vibration i is obtained:

$$m_i^* \ddot{y}_i(t) + c_i^* \dot{y}_i(t) + k_i^* y_i(t) = p_i^*(t) \quad (11)$$

where,

$$m_i^* \equiv \boldsymbol{\varphi}_i^T \mathbf{M} \boldsymbol{\varphi}_i \quad (12)$$

$$c_i^* \equiv \boldsymbol{\varphi}_i^T \mathbf{C} \boldsymbol{\varphi}_i \quad (13)$$

$$k_i^* \equiv \boldsymbol{\varphi}_i^T \mathbf{K} \boldsymbol{\varphi}_i \quad (14)$$

$$p_i^*(t) \equiv \boldsymbol{\varphi}_i^T \mathbf{p}(t) \quad (15)$$

The quantities in Equations (12) to (15) are known as the *generalized mass*, *generalized damping constant*, *generalized stiffness*, and *generalized load* of the structure respectively.

Equation (11) can also be written in a more convenient form as:

$$\ddot{y}_i(t) + 2\xi_i \omega_i \dot{y}_i(t) + \omega_i^2 y_i(t) = \frac{p_i^*(t)}{m_i^*} \quad (16)$$

where,

$$\xi_i \equiv \frac{c_i^*}{2m_i^* \omega_i} \quad (17)$$

is the damping ratio, and

$$\omega_i \equiv \sqrt{\frac{k_i^*}{m_i^*}} \quad (18)$$

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 (11)). This means that the modes of vibration effectively *uncouple* the dynamic equations of motion. In other words, to solve the dynamic problem, it suffices to solve n separate (uncoupled) differential equations of the form (11) 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.

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., $t_d \ll T$). A schematic plot of such load is shown in Figure 4a. The *impulse* of this load is by definition:

$$I = \int_0^{t_d} p(t) dt \quad (19)$$

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

$$y(t) = \frac{I}{m \omega_d} e^{-\xi \omega t} \sin \omega_d t \quad (20)$$

where,

$$\omega_d \equiv \omega \sqrt{1 - \xi^2} \quad (21)$$

is the *damped* frequency of the system [3].

In the case of a MDOF system, there will be one equation like (20) for each uncoupled degree of freedom i , i.e.,

$$y_i(t) = \frac{I_i}{m_i^* \omega_{di}} e^{-\xi_i \omega_i t} \sin \omega_{di} t \quad (22)$$

Following the definition of the generalized force for mode i given by (15), and the definition of impulse given in (19), the impulse for mode of vibration i , is given by:

$$I_i = \int_0^{t_d} \boldsymbol{\phi}_i^T \mathbf{p}(t) dt \quad (23)$$

The concept of response to an impulsive load can be used to advantage to find the response to general dynamic excitation of a SDOF system like the one shown in Figure 4b. This leads to the famous Duhamel's Integral written here for mode of vibration i ,

$$y_i(t) = \frac{1}{m_i^* \omega_{di}} \int_0^t p_i^*(\tau) e^{-\xi_i \omega_i (t-\tau)} \sin \omega_{di} (t-\tau) d\tau \quad (24)$$

COMPUTATIONAL ASPECTS.

The F.E. program used in the present study is a research F.E. code written in FORTRAN (it is possible to write a F.E. program using high-level programming languages like those available in MATLAB[®] or MAPLE[®] but the FORTRAN research code was already available and thus seemed a more logical choice for the purposes of this study). The F.E. program reads the standard input data, i.e., the geometry of the structure, the element properties, the element connectivity, and the boundary conditions (including loads). In this case it also reads 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 (16) and (17)). The program uses the direct stiffness method of analysis to assemble the stiffness and mass matrices. It then solves the eigenvalue problem given by (6) by means of Jacobi's method. Many other methods are available to solve eigenvalue problems and many are specifically tailored to eigenvalue problems arising from dynamical systems. Jacobi's method was used here for convenience, since a ready-made Jacobi routine is readily available in [5,6]. A standard Jacobi routine could be used here since the matrix \mathbf{E} given in (6) is symmetric. This is because the mass matrix for a lumped mass dynamical system is usually a multiple of the identity matrix. This makes \mathbf{M}^{-1} also a multiple of the identity matrix. Therefore the matrix \mathbf{E} is symmetric since \mathbf{K} is symmetric. For more general structural dynamics problems (i.e., those for which \mathbf{E} is not symmetric) other methods of solution like matrix iteration can be used [3,4]. It is possible however to use a modified version of Jacobi's method to solve general eigenvalue problems (those for which \mathbf{E} in (6) is not symmetric. See e.g., [4]).

RESPONSE CALCULATIONS

The output of the F.E. program consists of the modes of vibration of the plate and the corresponding modal frequencies. These data are read by a "response" program. This program can be easily written in MATLAB[®] or EXCEL[®]. The version used here was written in FORTRAN. 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 ingredient of the process is Equation (22) 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 actually the coefficient of $\boldsymbol{\phi}_i$ in Equation (10). 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 (23). Note that even though the impact load is applied to a particular node (or DOF), it affects many other degrees of freedom. In other words, even though the vector $\mathbf{p}(t)$ may have only one nonzero entry (that corresponding to the point (or node or DOF) of application of the load), many $p_i^*(t)$ will be nonzero (see Equation (15)). The reason is that whenever the value of ϕ_i at the point (node) of application of the load is nonzero, the integrand in Equation (23) will be nonzero.

All the calculations performed by the “response” program are better appreciated when arranged in the form of an algorithm, as follows:

ALGORITHM.

1. Set problem parameters and initialize variables: set the duration of the impact (impulse) load to a small number, say $t_d = \frac{T_1}{1000}$. Set the damping coefficient ξ_i (say $\xi_i = 2\%$). Choose the time interval Δt and set the initial time to zero:

$$\Delta t = \frac{\text{Duration_of_Simulation}}{\text{Number_of_Time_Intervals}}; t=0.$$
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 n_{modes} ,
 Calculate m_i^* according to Equation (12) and ω_{di} according to Equation (21).
 Calculate $I_i = P_0 * t_d * \phi(x_p, y_p, i)$
 Calculate y_i according to Equation (22).
 Superimpose mode contributions into the array frame:
 $frame(x,y) = frame(x,y) + y_i * \phi(x,y,i).$
 Enddo.
 Write $frame(x,y)$.

5. If $t \leq \text{Duration_of_Simulation}$ then goto 2, otherwise STOP.

In the foregoing algorithm, n_{modes} 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.

SAMPLE RESULTS.

The data for the sample problem presented here is shown in Figure 5. The plate has been 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

simulation 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, 7, and 25. 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 4 and 9 in Figure 6 is actually $5\Delta t$ or 5×10^{-4} seconds.

CONCLUDING REMARKS.

A complete 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 and 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. Craig R. R. “*Structural Dynamics: An Introduction to Computer Methods*”. Wiley & Sons, New York, 1981, Ch. 18.
2. 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.
3. Clough R. W. and Penzien J. “*Dynamics of Structures*”. McGraw-Hill, New York, 1975.
4. Bathe K-J. “*Finite Element Procedures*”. Prentice Hall, Englewood Cliffs, 1996, Ch. 11.
5. 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.
6. 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.

BIOGRAPHICAL INFORMATION

Dr. Carlos E. Orozco is an Associate Professor of Civil Engineering Technology at the University of North Carolina at Charlotte. Dr. Orozco has more than 20 years of experience in industry and academia and is the author of numerous technical articles in the fields of scientific computing, optimization, and structural mechanics. Dr. Orozco is a member of the American Institute of Aeronautics and Astronautics (AIAA), the American Society of Civil Engineers (ASCE), and the American Society for Engineering Education (ASEE). He was recently recognized as the 2004 Tau Alpha Pi Engineering Technology Professor of the Year at UNC Charlotte. Dr. Orozco has also been a Fulbright grantee.

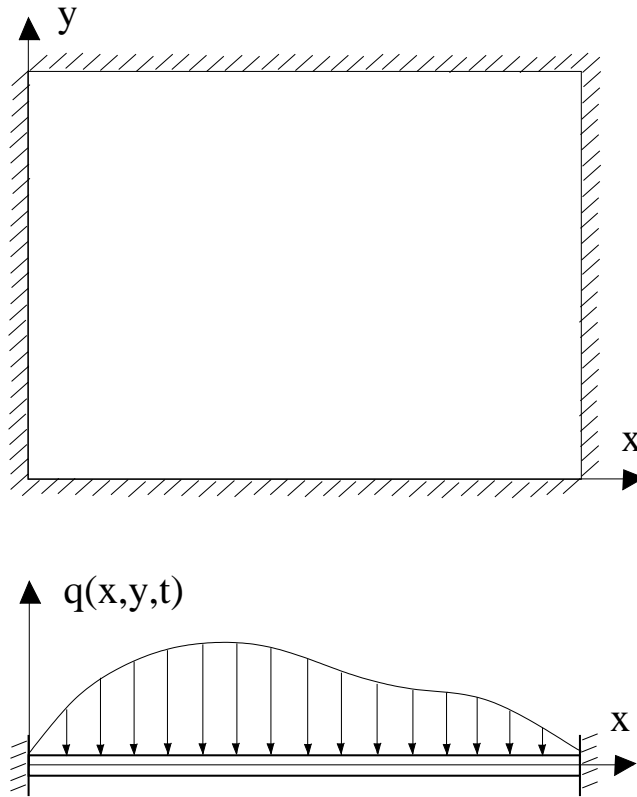


Figure 1. General, time-dependent load for a plate.

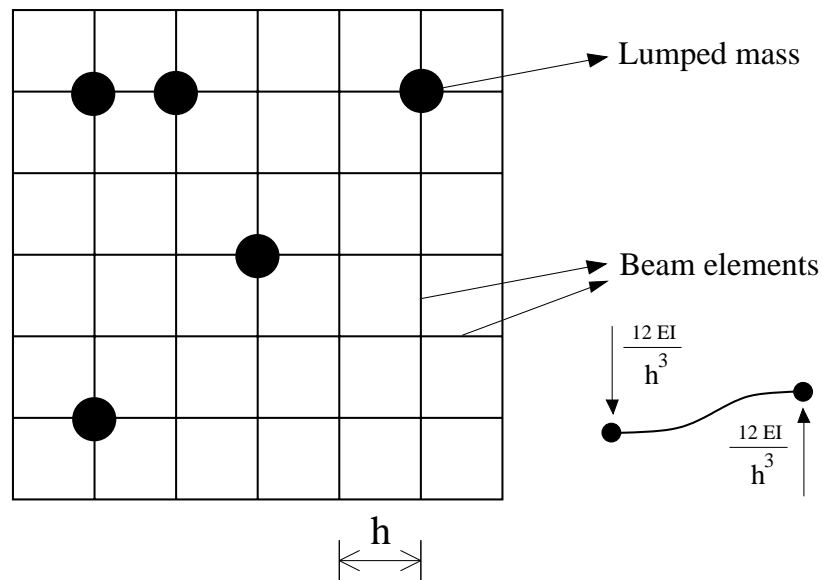


Figure 2. Two degree-of-freedom finite elements to approximately model the plate.

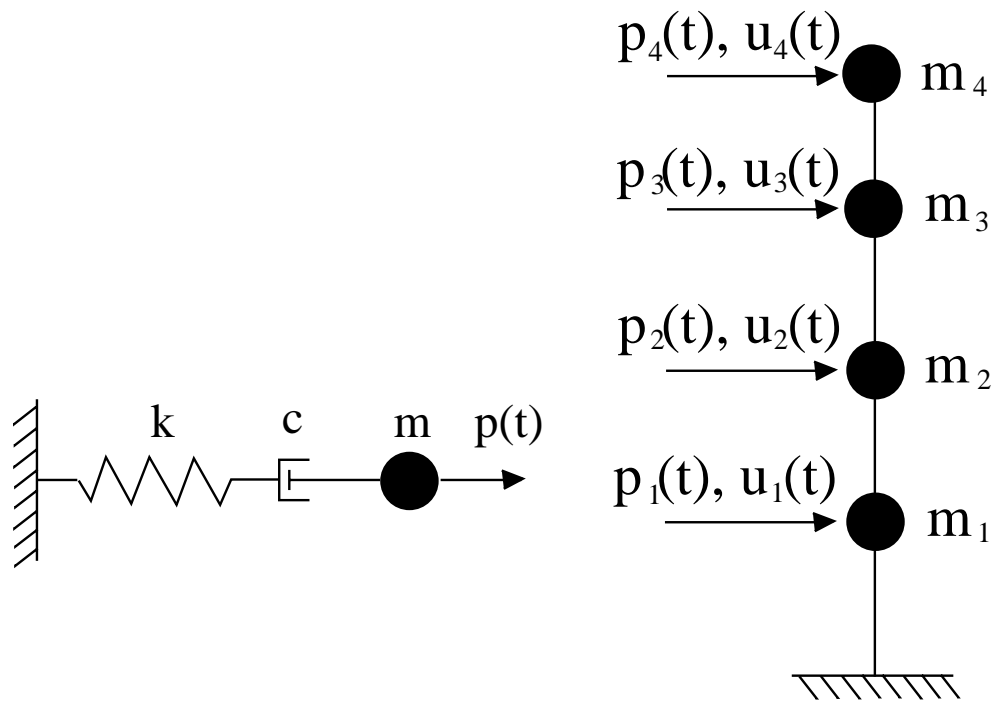


Figure 3. a) Single-degree-of-freedom (SDOF) System. b) Multi-degree-of-freedom (MDOF) System.

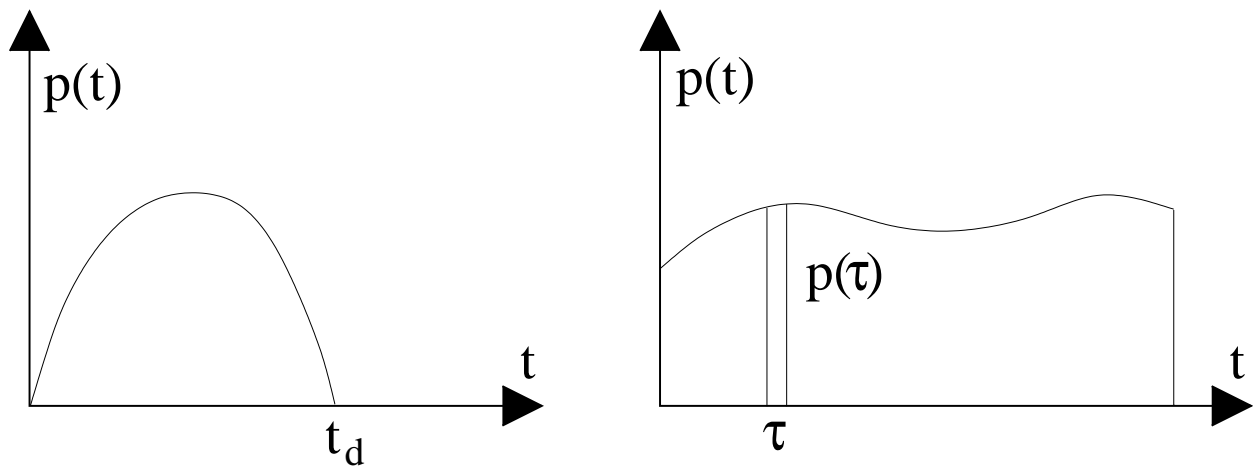


Figure 4. a) Impulse load.

b) General dynamic load

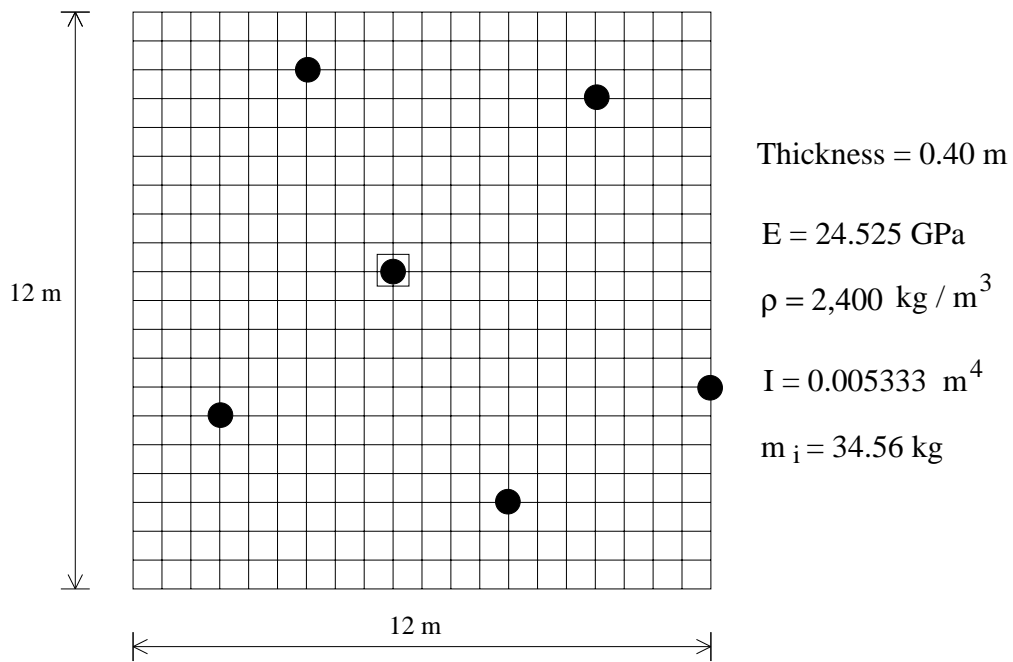


Figure 5. Plate dimensions and data for example problem.

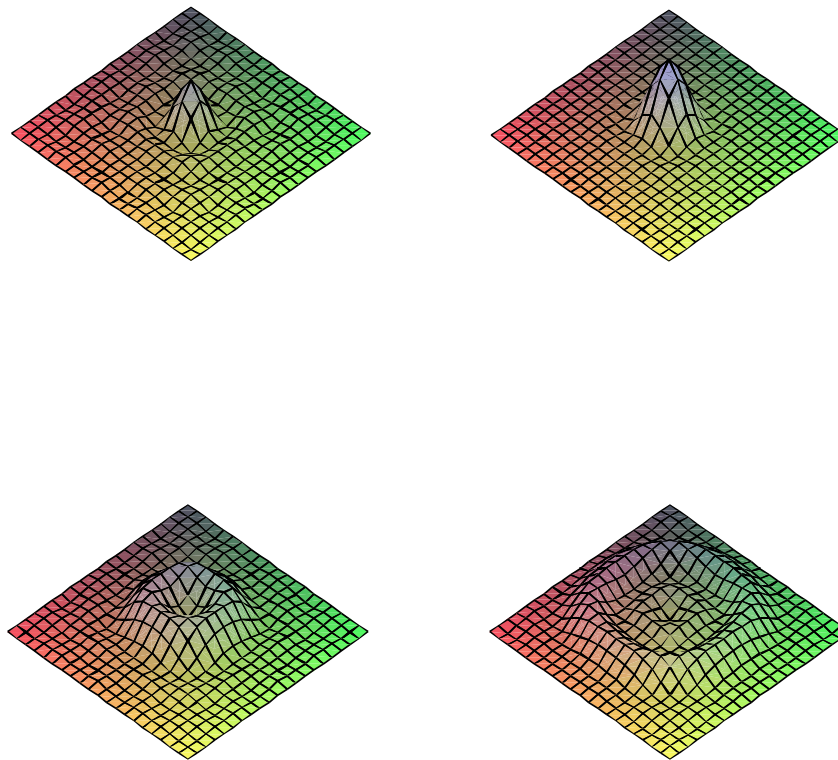


Figure 6. Clockwise From Top to Bottom: Frames 1, 4, 9, and 16 of computer animation.

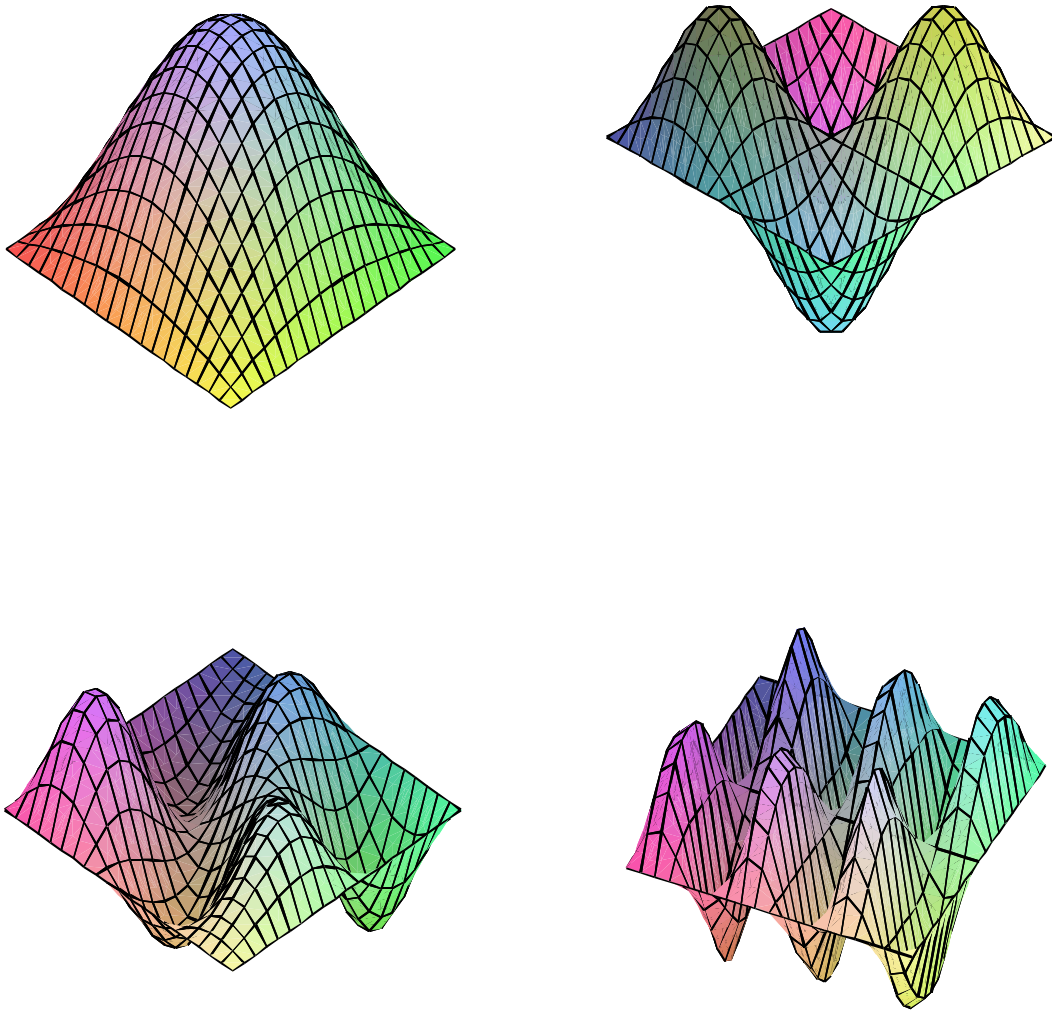


Figure 7. Clockwise From Top to Bottom: Modes of Vibration 1, 4, 7, and 25.