CHAPTER 4

FORMULATION OF FINITE ELEMENT METHOD
AND INTRODUCTION TO ANSYS

4.1 INTRODUCTION

The Navier and Levy solutions to the equations of composite plates are presented for rectangular geometries in Chapter 5. However, exact analytical or variational solutions to these problems cannot be developed when complex geometries, arbitrary boundary conditions or non-linearities are involved. Therefore, one must resort to approximate methods of analysis like FEM that are capable of solving problems.

The finite element method is a powerful computational technique for the solution of differential and integral equations that arise in various fields of engineering. The method is a generalization of the classical variation and weighted residual methods. Since most real world problems are defined on domains that are geometrically complex and may have different types of boundary conditions on different portions of the boundary of the domain, it is difficult to generate approximation functions required in the traditional variational methods. The basic idea of the finite element method is to view a given domain as an assemblage of simple geometric shapes called finite elements, for which it is possible to systematically generate the approximate functions needed in the solution of differential equations by any of the variational
elements makes the method a valuable practical tool for the solution of boundary, initial and
eigen value problems arising in various fields of engineering. The approximation functions are
often constructed using ideas from interpolation theory and hence they are also called
interpolation functions. Thus finite element method is a piecewise application of the
variational methods. For a given differential equation, it is possible to develop different finite
element approximation, depending on the choice of a particular variational method.

In this chapter the finite element models for laminated composite plates are developed.
The objective is to introduce the finite element formulations of laminated composite
structures.

4.2 FINITE ELEMENT FORMULATION OF THE PROBLEM

In the well-established finite element method, the total solution domain is discretized
into number of elements (ME) such that :

$$
\Pi(d) = \sum_{e=1}^{ME} \Pi^e(d)
$$

Where \( \pi \) and \( \pi^e \) are the potential energies of the total solution domain and the sub-
domain respectively. The potential energy for an element ‘e’ can be expressed in terms of the
internal strain energy, \( U^e \) and the external work done, \( W^e \), such that :

$$
\Pi^e(d) = U^e - W^e
$$

In which ‘d’ is the vector of nodal degrees-of-freedom of an element and is defined as :

$$
\{d\} = \{u_x, v_x, u_y, v_y, u_0^*, v_0^*, \theta_x^*, \theta_y^*\}
$$
Adopting the same shape function ‘\( N \)’ to define all the components of the generalized displacement vector ‘\( d \)’, the ‘\( d \)’ can be written as:

\[
d = \sum_{i=1}^{NE} N_i d_i
\]

.....4.4

In which, ‘\( NE \)’ is the number of nodes in the element. Now, referring to the expressions in Eq. (3.2a) the extensional strains \( \varepsilon_0, \varepsilon_0^* \), the bending curvatures \( k, k^* \) and the transverse shear strains \( \phi, \phi^* \) can be written in terms of the nodal displacement using the matrix notations as follows:

\[
\begin{align*}
\{\varepsilon_0\} &= L_E d \\
\{\varepsilon_0^*\} &= L_E^* d \\
\{k\} &= L_B d \\
\{k^*\} &= L_B^* d \\
\{\phi\} &= L_S d \\
\{\phi^*\} &= L_S^* d
\end{align*}
\]

.....4.5

In which the subscripts \( E, B \) and \( S \) refer to extension, bending and shear respectively and the matrices \( L_E, L_B \) and \( L_S \) attain the following form:

\[
L_E = \begin{bmatrix}
\frac{\partial}{\partial x} & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & \frac{\partial}{\partial y} & 0 & 0 & 0 & 0 & 0 & 0 \\
\frac{\partial}{\partial y} & 0 & \frac{\partial}{\partial x} & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & \frac{\partial}{\partial x} & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & \frac{\partial}{\partial y} & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & \frac{\partial}{\partial y} & \frac{\partial}{\partial x} & 0 & 0 \\
\end{bmatrix}
\]

.....4.6(a)
Knowing the generalized displacement vector, \( \mathbf{d} \) at all points within the element, the generalized strain vectors at any point are determined with the aid of Eq. (4.4) and (4.6) as follows:

\[
\begin{align*}
\{ \varepsilon_0 \} &= L_\varepsilon \mathbf{d} = L_\varepsilon \sum_{i=1}^{N_k} N_i \mathbf{d}_i = \sum_{i=1}^{N_k} B_{\varepsilon i} \mathbf{d}_i = B_\varepsilon \mathbf{a} \\
\{ k \} &= L_k \mathbf{d} = L_k \sum_{i=1}^{N_k} N_i \mathbf{d}_i = \sum_{i=1}^{N_k} B_i \mathbf{d}_i = B_k \mathbf{a} \\
\{ \phi \} &= I \ldots I \sum_{i=1}^{N_k} N_i \mathbf{d}_i = \sum_{i=1}^{N_k} B_\phi \mathbf{d}_i = B_\phi \mathbf{a}
\end{align*}
\]
in which

\[ B_i = L_i N_i, \quad B_k = \sum_{i=1}^{N_E} B_{ik}, \]
\[ B_\theta = L_\theta N_\theta, \quad B_\theta = \sum_{i=1}^{N_E} B_{i\theta}, \]
\[ B_\zeta = L_\zeta N_\zeta, \quad B_\zeta = \sum_{i=1}^{N_E} B_{i\zeta}, \]

and

\[ a = (d_1^T, d_2^T, \ldots, d_{N_k}^T) \]

Combining the expressions in Eq. (4.8), the B matrix for the i\textsuperscript{th} node can be written as:

\[ B_i = \begin{bmatrix} B_{ik} \\ B_{i\theta} \\ B_{i\zeta} \end{bmatrix} \]

The internal strain energy of an element is determined by integrating the products of in-plane, moment and shear stress resultants with the extensional, bending and shear strains, respectively, over the area of an element. This is expressed as:

\[ U^e = \frac{1}{2} \int_A \left[ \left( \epsilon_0, \epsilon_0^t \right) A \left[ \epsilon_0 \right] + \left( \epsilon_0, \epsilon_0^t \right) B \left[ k \right] + \left( \phi, \phi^t \right) D \left[ \phi \right] \right] dA \]

Replacing stress-resultants by the product of rigidity matrix and strains in the strain energy expression in Eq. (4.10), the energy can be expressed as:

\[ U^e = \frac{1}{2} \int_A \left[ \left( \epsilon_0, \epsilon_0^t \right) A \left[ \epsilon_0 \right] + \left( \epsilon_0, \epsilon_0^t \right) B \left[ k \right] + \left( \phi, \phi^t \right) D \left[ \phi \right] \right] dA \]
The internal strain energy expression in terms of the nodal displacements is derived by substituting relations in Eq. (4.7) into (4.11). The result is:

\[
U^e = \frac{1}{2} \int_A \left( a^i B^i E a B E a + a^t B^t B a + a^t B^t B a \right) dA
\]

or

\[
U^e = \frac{1}{2} a^t K^e a
\]

in which \( K^e \) is the element stiffness matrix and is expressed as

\[
K^e = \int_A \left( B^t E B + B^t B E B + B^t D B + B^t D B + B^t D B \right) dA
\]

The computation of the element stiffness matrix from Eq. (4.13) is economized by explicit multiplication of the \( B_a \), \( D \) and \( B_i \) matrices instead of carrying out the full matrix multiplication of the triple product. In addition, because of the symmetry of the stiffness matrix, only the blocks \( K_{ij} \) lying on one side of the main diagonal are formed. The integral is evaluated numerically, by the Gauss quadrature rule:

\[
K^e = \sum_{a=1}^g \sum_{b=1}^g W_a W_b |J| B^t D B
\]

In which \( W_a \) and \( W_b \) are weighting coefficients, \( g \) is the number of numerical quadrature points in each of the two directions \( (x, y) \) and \( |J| \) is the determinant of the standard Jacobian matrix. The subscripts \( i \) and \( j \) vary from one to the number of nodes per element. The matrices \( B_a \) and \( D \) are given by Eq. (4.9) and (3.11) respectively and \( B_i \) is obtained by replacing \( i \) by \( j \).
For the flexural analysis, the total external work done by the applied external loads for an element \( e \), is given by:

\[
W^e = a' F_i + a' \int_A (N_i q + N_i P_{mn}) dA
\]  

.....4.15

In which suffix \( i \), varies from one to the number of nodes per element. \( F \) is the vector of concentrated nodal loads corresponding to nodal degrees of freedom. \( q \) and \( P_{mn} \) are the uniform and sinusoidal distributed load intensities acting over an element \( e \) in the \( z \) direction.

The integral of Eq. (4.15) is evaluated numerically using the Gauss quadrature rule as follows:

\[
P_i = \sum_{i=1}^{n} \sum_{j=1}^{n} W_i W_j [J_i^0 001000000] \left[ q + P_{mn} \sin \frac{m \Pi x}{a} \sin \frac{n \Pi y}{b} \right]
\]  

.....4.16

In which \( a \) and \( b \) are the plate dimensions, \( x \) and \( y \) are the Gauss point coordinates and \( m \) and \( n \) are the usual harmonic numbers.

4.3 EQUATIONS OF MOTION AND ELEMENT MATRICES

With the finite element method for the discretization of space, Lagrange's equations of motion, when placed in matrix form in compact will become:

\[
M \ddot{d} + Kd = 0
\]  

.....4.17

Where \( K \) and \( M \) are the global stiffness and mass matrices respectively, obtained by the assembly of the corresponding elements matrices, and \( \ddot{d} \) is the second derivative of the displacements of the structure with respect to time.

The matrix Eq. (4.17) governing free vibration may also be expressed as:

\[
K \ddot{d} - \omega^2 M \ddot{d} = 0
\]  

.....4.18

Where \( \ddot{d} \) is a set of constant values at the nodes and is called the model vector, and \( \omega \) is the natural frequency of free vibration of the system. Eq. (4.18) can be solved, after
imposing the boundary conditions of the problem, by any standard eigen value program. For
the purpose of evaluation, Eq. (4.18) is converted into the standard eigen value format as :

\[(K - \lambda M)\ddot{d} = 0 \quad \lambda = \omega^2 \quad \text{....4.19}\]

4.4 ELEMENT MASS MATRIX

A special mass matrix diagonalization scheme which is more sophisticated is given by:

\[M' = \int_N^T mNdA, \quad N = [N_1, N_2, \ldots, N_m] \quad \text{....4.20}\]

\[
m = \begin{bmatrix}
m_{11} & 0 & 0 & m_{14} & 0 & m_{16} & 0 & m_{18} & 0 \\
0 & m_{22} & 0 & 0 & m_{25} & 0 & m_{27} & 0 & m_{29} \\
0 & 0 & m_{33} & 0 & 0 & 0 & 0 & 0 & 0 \\
m_{41} & 0 & 0 & m_{44} & 0 & m_{46} & 0 & m_{48} & 0 \\
0 & m_{52} & 0 & 0 & m_{55} & 0 & m_{57} & 0 & m_{59} \\
m_{61} & 0 & 0 & m_{64} & 0 & m_{66} & 0 & m_{68} & 0 \\
0 & m_{72} & 0 & 0 & m_{75} & 0 & m_{77} & 0 & m_{79} \\
m_{81} & 0 & 0 & m_{84} & 0 & m_{86} & 0 & m_{88} & 0 \\
0 & m_{92} & 0 & 0 & m_{95} & 0 & m_{97} & 0 & m_{99} \\
\end{bmatrix}
\]

\[m_{11} = m_{22} = m_{33} = 1_1, \quad m_{14} = m_{25} = 1_2, \quad m_{16} = m_{27} = m_{44} = m_{55} = 1_3, \]

\[m_{18} = m_{29} = m_{46} = m_{57} = 1_4, \quad m_{48} = m_{59} = m_{66} = m_{77} = 1_5, \quad m_{68} = m_{79} = 1_6, \]

\[m_{88} = m_{99} = 1_7, \]

The same procedure is adopted for the displacement model-2 and also for the thermal
analysis

4.5. INTRODUCTION TO ANSYS FEM PACKAGE

There are many powerful, general purpose finite element packages available
commercially. Some of the familiar finite element packages are ANSYS, ADINA, ASKA,
MARC AND SAP 7. The ANSYS packages is a proprietary of M/s. Swanson Analysis
The ANSYS programme is designed to be user oriented. We do not require special knowledge of the system operations or computer programming in order to use this package. The flexibility, capabilities and options made the ANSYS program to be applied to a wide variety of engineering applications. The programme contains many routines, all interrelated and all for the main purpose of achieving a solution to an engineering problem by the finite element method.

An engineering problem is usually solved in three phases.

1. pre-processing
2. solution
3. post processing

Table 4.1 Shows three phases along with some of the operations in each phase

<table>
<thead>
<tr>
<th>Pre processing phase</th>
<th>Solution phase</th>
<th>Post processing phase</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mesh Generation</td>
<td>Element matrix Formulation</td>
<td>Post solution operations</td>
</tr>
<tr>
<td>Geometry definition</td>
<td>overall Matrix</td>
<td>Post data printout</td>
</tr>
<tr>
<td>Material definitions</td>
<td>triangularization</td>
<td></td>
</tr>
<tr>
<td>Constraint definitions</td>
<td>Displacement stress etc. calculations</td>
<td>Post data scanning</td>
</tr>
<tr>
<td>Load definitions</td>
<td></td>
<td>Post data displays</td>
</tr>
<tr>
<td>Modal displays</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

The main routines and analysis types associated with these phases are shown in table 5.2. For example preprocessing may consists of using PREP 7 routine alone or for some transient analysis both PREP7 and PREP6 may be used. In general PREP7 is used for majority of all preprocessing. Once the preprocessing has been completed the user may progress through various analysis (on the same mode) in the same solution phase. Once the
solution phase has been completed the user may progress through various post processing operations.

Table 4.2 Main routines and analysis types

<table>
<thead>
<tr>
<th>PREP 7</th>
<th>Analysis types</th>
<th>Post 1,29,30</th>
</tr>
</thead>
<tbody>
<tr>
<td>(General Mesh generation</td>
<td>Linear, Nonlinear</td>
<td>(Tabular printout)</td>
</tr>
<tr>
<td>and Nodal definition)</td>
<td>Static</td>
<td></td>
</tr>
<tr>
<td>PREP 6</td>
<td>1 - Buckling</td>
<td>Post 26</td>
</tr>
<tr>
<td>(Additional transient</td>
<td>2 - Modal</td>
<td>(Tabular Printout Graph</td>
</tr>
<tr>
<td>boundary condition</td>
<td>3 - Full harmonic response</td>
<td>displays)</td>
</tr>
<tr>
<td>generation)</td>
<td>4 - Non linear transient dynamic</td>
<td></td>
</tr>
<tr>
<td></td>
<td>5 - Linear transient</td>
<td></td>
</tr>
<tr>
<td></td>
<td>6 - Reduced harmonic Response</td>
<td></td>
</tr>
<tr>
<td></td>
<td>7 - Sub-structuring</td>
<td></td>
</tr>
<tr>
<td></td>
<td>8 - Heat transfer</td>
<td></td>
</tr>
</tbody>
</table>

The ANSYS programme is designed to run fully batch or fully interactive or combination of both.

(A) Interactive Runs:

An interactive run may be proceed as follows:

1. Execute the prep version the interactive version or the full ANSYS programme.
2. Select interactive mode with /INT command.
3. Input each data as desired.
4. Immediate graphic displays are available during interactive session. After the first display is formed, the terminal screen is divided into a graphic screen and dialog screen. The graphic screen, standard text screen or both respectively.

5. Normal programme exit, a eof command input at any routine. Begin level will cause a normal exit from the ANSYS programme.

(B) Batch runs:

A batch run may proceed as follows:

1. Prepare a data file with text editor. The data may consists of a single phase or all phases of the analysis.
2. Submit the file for batch run.
3. Recover printout and graphic displays produced from the run.

4.5.1 WAVE FRONT EQUATION SOLVER:

The ANSYS programme uses an in memory wave front solutions procedure for the system of simultaneous linear equations developed from the assembled finite elements. The number of equations which are active after any element has been processed during the solution procedure is called wave front at that point. The wave front depends upon the amount of memory available for a given problem. In the wave front procedure, the ordering of elements is crucial to minimize the size of the wave front.

4.5.2 SHELL 99 100-LAYERS STRUCTURAL ELEMENT:

The element SHELL99 is used for layered applications of a structural shell model. The element is an extension of the 16-layer SHELL91 element in that up to 100 different material layers are permitted, more than 100 layers are required and a user input constitutive matrix option is available. The element has six degrees of freedom at each node i.e., translations in the x, y and z directions and rotations about the x, y and z-axes.
Figure 4-1  SHELL99  100-Layer Structural Shell

Element Coordinate System (shown for KEYOPT(4)=0)

LN = Layer Number
NL = Total Number of Layers