THE FINITE ELEMENT METHOD AND APPLICATIONS IN ENGINEERING USING ANSYS®

THE FINITE ELEMENT METHOD AND APPLICATIONS IN ENGINEERING USING ANSYS®

by

Erdogan Madenci
Ibrahim Guven
The University of Arizona

Springer

Erdogan Madenci
The University of Arizona

Ibrahim Guven
The University of Arizona

Library of Congress Control Number: 2005052017

ISBN-10: 0-387-28289-0 e-ISBN-10: 0-387-28290-4
ISBN-13: 978-0387-28289-3 e-ISBN-13: 978-0387-282909

Printed in the United States of America

9 8 7 6 5 4 3

springer.com

PREFACE

The finite element method (FEM) has become a staple for predicting and simulating the physical behavior of complex engineering systems. The commercial finite element analysis (FEA) programs have gained common acceptance among engineers in industry and researchers at universities and government laboratories. Therefore, academic engineering departments include graduate or undergraduate senior-level courses that cover not only the theory of FEM but also its applications using the commercially available FEA programs.

The goal of this book is to provide students with a theoretical and practical knowledge of the finite element method and the skills required to analyze engineering problems with ANSYS®, a commercially available FEA program. This book, designed for seniors and first-year graduate students, as well as practicing engineers, is introductory and self-contained in order to minimize the need for additional reference material.

In addition to the fundamental topics in finite element methods, it presents advanced topics concerning modeling and analysis with ANSYS®. These topics are introduced through extensive examples in a step-by-step fashion from various engineering disciplines. The book focuses on the use of ANSYS® through both the Graphics User Interface (GUI) and the ANSYS® Parametric Design Language (APDL). Furthermore, it includes a CD-ROM with the "*input*" files for the example problems so that the students can regenerate them on their own computers. Because of printing costs, the printed figures and screen shots are all in gray scale. However, color versions are provided on the accompanying CD-ROM.

Chapter 1 provides an introduction to the concept of FEM. In Chapter 2, the analysis capabilities and fundamentals of ANSYS®, as well as practical modeling considerations, are presented. The fundamentals of discretization and approximation functions are presented in Chapter 3. The modeling techniques and details of mesh generation in ANSYS® are presented in Chapter 4. Steps for obtaining solutions and reviews of results are presented in Chapter 5. In Chapter 6, the derivation of finite element equations based on the method of weighted residuals and principle of minimum potential energy

is explained and demonstrated through example problems. The use of commands and APDL and the development of macro files are presented in Chapter 7. In Chapter 8, example problems on linear structural analysis are worked out in detail in a step-by-step fashion. The example problems related to heat transfer and moisture diffusion are demonstrated in Chapter 9. Nonlinear structural problems are presented in Chapter 10. Advanced topics concerning submodeling, substructuring, interaction with external files, and modification of ANSYS®-GUI are presented in Chapter 11.

There are more than 40 example problems considered in this book; solutions to most of these problems using ANSYS® are demonstrated using GUI in a step-by-step fashion. The remaining problems are demonstrated using the APDL. However, the steps taken in either GUI- or APDL-based solutions may not be the optimum/shortest possible way. Considering the steps involved in obtaining solutions to engineering problems (e.g., model generation, meshing, solution options, etc.), there exist many different routes to achieve the same solution. Therefore, the authors strongly encourage the students/engineers to experiment with modifications to the analysis steps presented in this book.

We are greatly indebted to Connie Spencer for her invaluable efforts in typing, editing, and assisting with each detail associated with the completion of this book. Also, we appreciate the contributions made by Dr. Atila Barut, Mr. Erkan Oterkus, Ms. Abigail Agwai, Mr. Manabendra Das, and Mr. Bahattin Kilic in the solution of the example problems. The permission provided by ANSYS, Inc. to print the screen shots is also appreciated.

TABLE OF CONTENTS

LIST OF PROBLEMS SOLVED

Chapter 1

INTRODUCTION

1.1 Concept

The Finite Element Analysis (FEA) method, originally introduced by Turner et al. (1956), is a powerful computational technique for approximate solutions to a variety of "real-world" engineering problems having complex domains subjected to general boundary conditions. FEA has become an essential step in the design or modeling of a physical phenomenon in various engineering disciplines. A physical phenomenon usually occurs in a continuum of matter (solid, liquid, or gas) involving several field variables. The field variables vary from point to point, thus possessing an infinite number of solutions in the domain. Within the scope of this book, a continuum with a known boundary is called a domain.

The basis of FEA relies on the decomposition of the domain into a finite number of subdomains (elements) for which the systematic approximate solution is constructed by applying the variational or weighted residual methods. In effect, FEA reduces the problem to that of a finite number of unknowns by dividing the domain into elements and by expressing the unknown field variable in terms of the assumed approximating functions within each element. These functions (also called interpolation functions) are defined in terms of the values of the field variables at specific points, referred to as nodes. Nodes are usually located along the element boundaries, and they connect adjacent elements.

The ability to discretize the irregular domains with finite elements makes the method a valuable and practical analysis tool for the solution of boundary, initial, and eigenvalue problems arising in various engineering disciplines. Since its inception, many technical papers and books have appeared on the development and application of FEA. The books by Desai and Abel (1971), Oden (1972), Gallagher (1975), Huebner (1975), Bathe and Wilson (1976), Ziekiewicz (1977), Cook (1981), and Bathe (1996) have influenced the current state of FEA. Representative common engineering problems and their corresponding FEA discretizations are illustrated in Fig. 1.1.

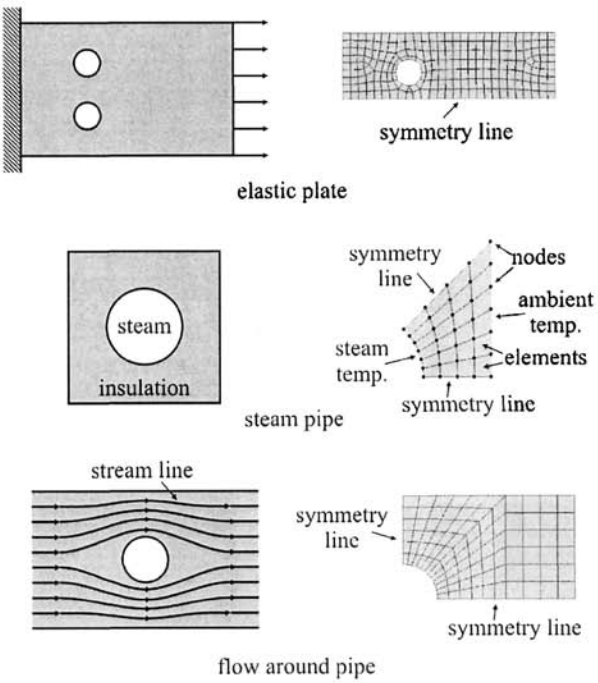

Fig. 1.1 FEA representation of practical engineering problems.

The finite element analysis method requires the following major steps:

- Discretization of the domain into a finite number of subdomains (elements).
- Selection of interpolation functions.
- Development of the element matrix for the subdomain (element).
- Assembly of the element matrices for each subdomain to obtain the global matrix for the entire domain.
- Imposition of the boundary conditions.
- Solution of equations.
- Additional computations (if desired).

There are three main approaches to constructing an approximate solution based on the concept of FEA:

Direct Approach: This approach is used for relatively simple problems, and it usually serves as a means to explain the concept of FEA and its important steps (discussed in Sec. 1.4).

***Weighted Residuals*:** This is a versatile method, allowing the application of FEA to problems whose functionals cannot be constructed. This approach directly utilizes the governing differential equations, such as those of heat transfer and fluid mechanics (discussed in Sec. 6.1).

Variational Approach: This approach relies on the calculus of variations, which involves extremizing a functional. This functional corresponds to the potential energy in structural mechanics (discussed in Sec. 6.2).

In matrix notation, the global system of equations can be cast into

$$\mathbf{K}\mathbf{u} = \mathbf{F} \tag{1.1}$$

where $\mathbf{K}$ is the system stiffness matrix, $\mathbf{u}$ is the vector of unknowns, and $\mathbf{F}$ is the force vector. Depending on the nature of the problem, $\mathbf{K}$ may be dependent on $\mathbf{u}$, i.e., $\mathbf{K} = \mathbf{K}(\mathbf{u})$ and $\mathbf{F}$ may be time dependent, i.e., $\mathbf{F} = \mathbf{F}(t)$.

1.2 Nodes

As shown in Fig. 1.2, the transformation of the practical engineering problem to a mathematical representation is achieved by discretizing the domain of interest into elements (subdomains). These elements are connected to each other by their "common" nodes. A node specifies the coordinate location in space where degrees of freedom and actions of the physical problem exist. The nodal unknown(s) in the matrix system of equations represents one (or more) of the primary field variables. Nodal variables assigned to an element are called the degrees of freedom of the element.

The common nodes shown in Fig. 1.2 provide continuity for the nodal variables (degrees of freedom). Degrees of freedom (DOF) of a node are dictated by the physical nature of the problem and the element type. Table 1.1 presents the DOF and corresponding "forces" used in FEA for different physical problems.

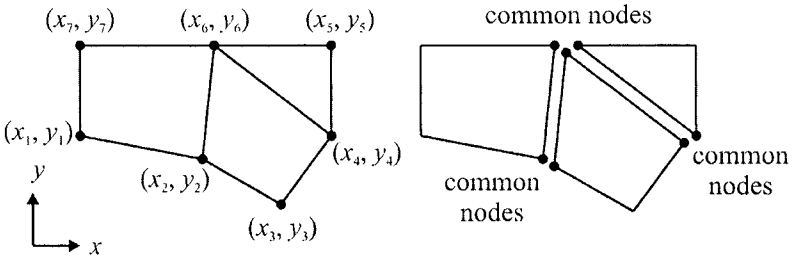

Fig. 1.2 Division of a domain into subdomains (elements).

Table 1.1 Degrees of freedom and force vectors in FEA for different engineering disciplines.

Discipline	DOF	Force Vector
Structural/solids	Displacement	Mechanical forces
Heat conduction	Temperature	Heat flux
Acoustic fluid	Displacement potential	Particle velocity
Potential flow	Pressure	Particle velocity
General flows	Velocity	Fluxes
Electrostatics	Electric potential	Charge density
Magnetostatics	Magnetic potential	Magnetic intensity

1.3 Elements

Depending on the geometry and the physical nature of the problem, the domain of interest can be discretized by employing line, area, or volume elements. Some of the common elements in FEA are shown in Fig. 1.3. Each element, identified by an element number, is defined by a specific sequence of global node numbers. The specific sequence (usually counter-clockwise) is based on the node numbering at the element level. The node numbering sequence for the elements shown in Fig. 1.4 are presented in Table 1. 2.

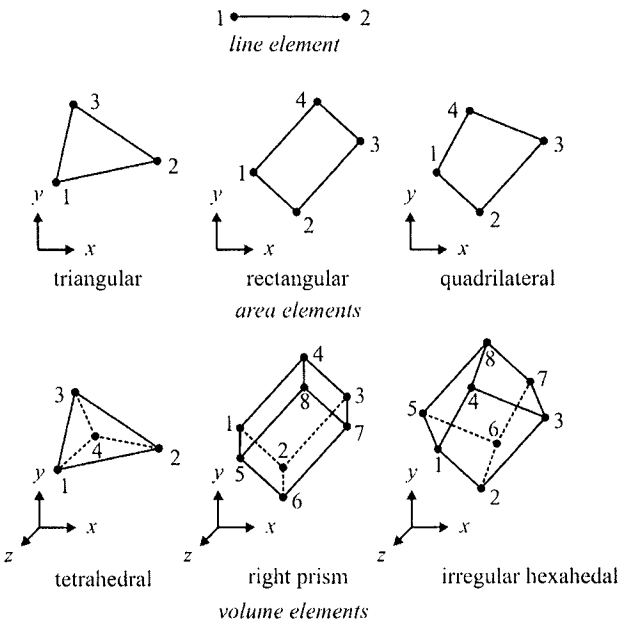

Fig. 1.3 Description of line, area, and volume elements with node numbers at the element level.

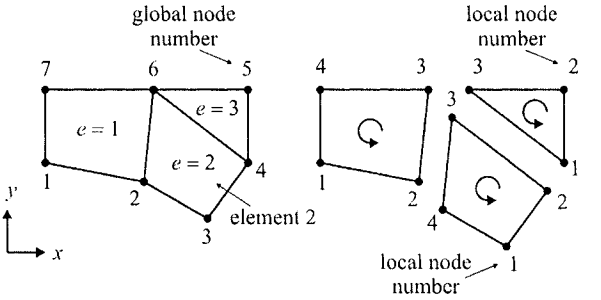

Fig. 1.4 Discretization of a domain: element and node numbering.

Table 1.2 Description of numbering at the element level.

Element Number	Node 1	Node 2	Node 3	Node 4
1	1	2	6	7
2	3	4	6	2
3	4	5	6	

1.4 Direct Approach

Although the direct approach is suitable for simple problems, it involves each fundamental step of a typical finite element analysis. Therefore, this approach is demonstrated by considering a linear spring system and heat flow in a one-dimensional (1-D) domain.

1.4.1 Linear Spring

As shown in Fig. 1.5, a linear spring with stiffness k has two nodes. Each node is subjected to axial loads of f_1 and f_2, resulting in displacements of u_1 and u_2 in their defined positive directions.

Subjected to these nodal forces, the resulting deformation of the spring becomes

$$u = u_1 - u_2 \tag{1.2}$$

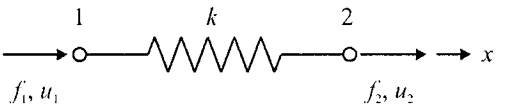

Fig. 1.5 Free-body diagram of a linear spring element.

which is related to the force acting on the spring by

$$f_1 = ku = k(u_1 - u_2) \tag{1.3}$$

The equilibrium of forces requires that

$$f_2 = -f_1 \tag{1.4}$$

which yields

$$f_2 = k(u_2 - u_1) \tag{1.5}$$

Combining Eq. (1.3) and (1.5) and rewriting the resulting equations in matrix form yield

$$\begin{bmatrix} k & -k \\ -k & k \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} f_1 \\ f_2 \end{Bmatrix} \quad \text{or} \quad \mathbf{k}^{(e)}\mathbf{u}^{(e)} = \mathbf{f}^{(e)} \tag{1.6}$$

in which $\mathbf{u}^{(e)}$ is the vector of nodal unknowns representing displacement and $\mathbf{k}^{(e)}$ and $\mathbf{f}^{(e)}$ are referred to as the element characteristic (stiffness) matrix and element right-hand-side (force) vector, respectively. The superscript (e) denotes the element numbered as 'e'.

The stiffness matrix can be expressed in indicial form as $k_{ij}^{(e)}$

$$\mathbf{k}^{(e)} \sim k_{ij}^{(e)} \tag{1.7}$$

where the subscripts i and j $(i, j = 1, 2)$ are the row and the column numbers. The coefficients, $k_{ij}^{(e)}$, may be interpreted as the force required at node i to produce a unit displacement at node j while all the other nodes are fixed.

1.4.2 Heat Flow

Uniform heat flow through the thickness of a domain whose in-plane dimensions are long in comparison to its thickness can be considered as a one-dimensional analysis. The cross section of such a domain is shown in Fig. 1.6. In accordance with Fourier's Law, the rate of heat flow per unit area in the x-direction can be written as

$$q = -kA\frac{d\theta}{dx} \tag{1.8}$$

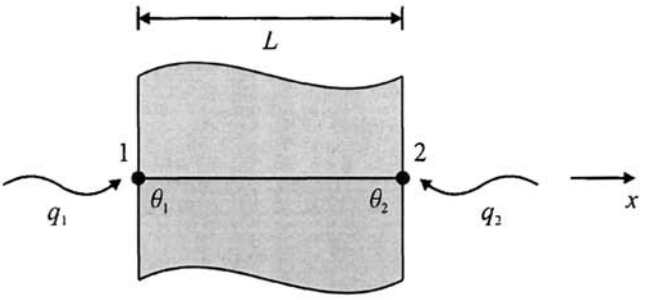

Fig. 1.6 One-dimensional heat flow.

where A is the area normal to the heat flow, θ is the temperature, and k is the coefficient of thermal conductivity. For constant k, Eq. (1.8) can be rewritten as

$$q = -kA\frac{\Delta\theta}{L} \tag{1.9}$$

in which $\Delta\theta = \theta_2 - \theta_1$ denotes the temperature drop across the thickness denoted by L of the domain.

As illustrated in Fig. 1.6, the nodal flux (heat flow entering a node) at Node 1 becomes

$$q_1 = \frac{kA}{L}(\theta_1 - \theta_2) \tag{1.10}$$

The balance of the heat flux requires that

$$q_2 = -q_1 \tag{1.11}$$

which yields

$$q_2 = -\frac{kA}{L}(\theta_1 - \theta_2) \tag{1.12}$$

Combining Eq. (1.10) and (1.12) and rewriting the resulting equations in matrix form yield

$$\frac{kA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} \theta_1 \\ \theta_2 \end{Bmatrix} = \begin{Bmatrix} q_1 \\ q_2 \end{Bmatrix} \quad \text{or} \quad \mathbf{k}^{(e)}\boldsymbol{\theta}^{(e)} = \mathbf{q}^{(e)} \tag{1.13}$$

in which $\boldsymbol{\theta}^{(e)}$ is the vector of nodal unknowns representing temperature and $\mathbf{k}^{(e)}$ and $\mathbf{q}^{(e)}$ are referred to as the element characteristic matrix and element right-hand-side vector, respectively.

1.4.3 Assembly of the Global System of Equations

Modeling an engineering problem with finite elements requires the assembly of element characteristic (stiffness) matrices and element right-hand-side (force) vectors, leading to the global system of equations

$$\mathbf{K}\mathbf{u} = \mathbf{F} \tag{1.14}$$

in which $\mathbf{K}$ is the assembly of element characteristic matrices, referred to as the global system matrix and $\mathbf{F}$ is the assembly of element right-hand-side vectors, referred to as the global right-hand-side (force) vector. The vector of nodal unknowns is represented by $\mathbf{u}$.

The global system matrix, $\mathbf{K}$, can be obtained from the "expanded" element coefficient matrices, $\mathbf{k}^{(e)}$, by summation in the form

$$\mathbf{K} = \sum_{e=1}^{E} \mathbf{k}^{(e)} \tag{1.15}$$

in which the parameter E denotes the total number of elements. The "expanded" element characteristic matrices are the same size as the global system matrix but have rows and columns of zeros corresponding to the nodes not associated with element (e). The size of the global system matrix is dictated by the highest number among the global node numbers.

Similarly, the global right-hand-side vector, $\mathbf{F}$, can be obtained from the "expanded" element coefficient vectors, $\mathbf{f}^{(e)}$, by summation in the form

$$\mathbf{F} = \sum_{e=1}^{E} \mathbf{f}^{(e)} \tag{1.16}$$

The "expanded" element right-hand-side vectors are the same size as the global right-hand-side vector but have rows of zeros corresponding to the nodes not associated with element (e). The size of the global right-hand-side vector is also dictated by the highest number among the global node numbers.

The explicit steps in the construction of the global system matrix and the global right-hand-side-vector are explained by considering the system of linear springs shown in Fig. 1.7. Associated with element (e), the element equations for a spring given by Eq. (1.6) are rewritten as

$$\begin{bmatrix} k_{11}^{(e)} & k_{12}^{(e)} \\ k_{21}^{(e)} & k_{22}^{(e)} \end{bmatrix} \begin{Bmatrix} u_1^{(e)} \\ u_2^{(e)} \end{Bmatrix} = \begin{Bmatrix} f_1^{(e)} \\ f_2^{(e)} \end{Bmatrix} \tag{1.17}$$

in which $k_{11}^{(e)} = k_{22}^{(e)} = k^{(e)}$ and $k_{12}^{(e)} = k_{21}^{(e)} = -k^{(e)}$. The subscripts used in Eq. (1.17) correspond to Node 1 and Node 2, the local node numbers of element (e). The global node numbers specifying the connectivity among the elements for this system of springs is shown in Fig. 1.7, and the connectivity information is tabulated in Table 1.3.

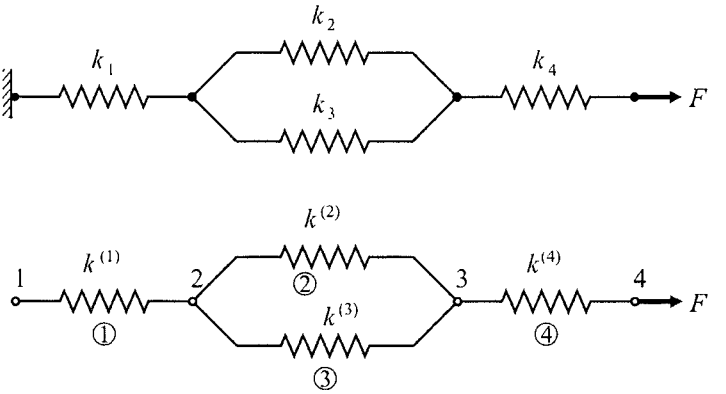

Fig. 1.7 System of linear springs (top) and corresponding FEA model (bottom).

Table 1.3 Table of connectivity.

Element Number	Local Node Numbering	Global Node Numbering
1	1	1
	2	2
2	1	2
	2	3
3	1	2
	2	3
4	1	3
	2	4

In accordance with Eq. (1.15), the size of the global system matrix is (4×4) and the specific contribution from each element is captured as

$$\text{Element 1:}\quad \begin{matrix}\boxed{1}&\boxed{2}\end{matrix}\begin{bmatrix}k_{11}^{(1)} & k_{12}^{(1)}\\ k_{21}^{(1)} & k_{22}^{(1)}\end{bmatrix}\begin{matrix}\boxed{1}\\ \boxed{2}\end{matrix}\Rightarrow \begin{matrix}\boxed{1}&\boxed{2}&\boxed{3}&\boxed{4}\end{matrix}\begin{bmatrix}k_{11}^{(1)} & k_{12}^{(1)} & 0 & 0\\ k_{21}^{(1)} & k_{22}^{(1)} & 0 & 0\\ 0 & 0 & 0 & 0\\ 0 & 0 & 0 & 0\end{bmatrix}\begin{matrix}\boxed{1}\\ \boxed{2}\\ \boxed{3}\\ \boxed{4}\end{matrix}\equiv \mathbf{k}^{(1)} \tag{1.18}$$

$$\text{Element 2:}\quad \begin{matrix}\boxed{2}&\boxed{3}\end{matrix}\begin{bmatrix}k_{11}^{(2)} & k_{12}^{(2)}\\ k_{21}^{(2)} & k_{22}^{(2)}\end{bmatrix}\begin{matrix}\boxed{2}\\ \boxed{3}\end{matrix}\Rightarrow \begin{matrix}\boxed{1}&\boxed{2}&\boxed{3}&\boxed{4}\end{matrix}\begin{bmatrix}0 & 0 & 0 & 0\\ 0 & k_{11}^{(2)} & k_{12}^{(2)} & 0\\ 0 & k_{21}^{(2)} & k_{22}^{(2)} & 0\\ 0 & 0 & 0 & 0\end{bmatrix}\begin{matrix}\boxed{1}\\ \boxed{2}\\ \boxed{3}\\ \boxed{4}\end{matrix}\equiv \mathbf{k}^{(2)} \tag{1.19}$$

$$\text{Element 3:}\quad \begin{matrix}\boxed{2}&\boxed{3}\end{matrix}\begin{bmatrix}k_{11}^{(3)} & k_{12}^{(3)}\\ k_{21}^{(3)} & k_{22}^{(3)}\end{bmatrix}\begin{matrix}\boxed{2}\\ \boxed{3}\end{matrix}\Rightarrow \begin{matrix}\boxed{1}&\boxed{2}&\boxed{3}&\boxed{4}\end{matrix}\begin{bmatrix}0 & 0 & 0 & 0\\ 0 & k_{11}^{(3)} & k_{12}^{(3)} & 0\\ 0 & k_{21}^{(3)} & k_{22}^{(3)} & 0\\ 0 & 0 & 0 & 0\end{bmatrix}\begin{matrix}\boxed{1}\\ \boxed{2}\\ \boxed{3}\\ \boxed{4}\end{matrix}\equiv \mathbf{k}^{(3)} \tag{1.20}$$

$$\text{Element 4:}\quad \begin{matrix}\boxed{3}&\boxed{4}\end{matrix}\begin{bmatrix}k_{11}^{(4)} & k_{12}^{(4)}\\ k_{21}^{(4)} & k_{22}^{(4)}\end{bmatrix}\begin{matrix}\boxed{3}\\ \boxed{4}\end{matrix}\Rightarrow \begin{matrix}\boxed{1}&\boxed{2}&\boxed{3}&\boxed{4}\end{matrix}\begin{bmatrix}0 & 0 & 0 & 0\\ 0 & 0 & 0 & 0\\ 0 & 0 & k_{11}^{(4)} & k_{12}^{(4)}\\ 0 & 0 & k_{21}^{(4)} & k_{22}^{(4)}\end{bmatrix}\begin{matrix}\boxed{1}\\ \boxed{2}\\ \boxed{3}\\ \boxed{4}\end{matrix}\equiv \mathbf{k}^{(4)} \tag{1.21}$$

Performing their assembly leads to

$$\mathbf{K}=\sum_{e=1}^{4}\mathbf{k}^{(e)}=\mathbf{k}^{(1)}+\mathbf{k}^{(2)}+\mathbf{k}^{(3)}+\mathbf{k}^{(4)} \tag{1.22}$$

or

$$\mathbf{K}=\begin{bmatrix} k_{11}^{(1)} & k_{12}^{(1)} & 0 & 0 \\ k_{21}^{(1)} & \left(k_{22}^{(1)}+k_{11}^{(2)}+k_{11}^{(3)}\right) & \left(k_{12}^{(2)}+k_{12}^{(3)}\right) & 0 \\ 0 & \left(k_{21}^{(2)}+k_{21}^{(3)}\right) & \left(k_{22}^{(2)}+k_{22}^{(3)}+k_{11}^{(4)}\right) & k_{12}^{(4)} \\ 0 & 0 & k_{21}^{(4)} & k_{22}^{(4)} \end{bmatrix} \tag{1.23}$$

In accordance with Eq. (1.16), the size of the global right-hand-side vector is (4×1) and the specific contribution from each element is captured as

$$\text{Element 1:} \quad \begin{Bmatrix} f_1^{(1)} \\ f_2^{(1)} \end{Bmatrix} \begin{matrix} \boxed{1} \\ \boxed{2} \end{matrix} \Rightarrow \begin{Bmatrix} f_1^{(1)} \\ f_2^{(1)} \\ 0 \\ 0 \end{Bmatrix} \begin{matrix} \boxed{1} \\ \boxed{2} \\ \boxed{3} \\ \boxed{4} \end{matrix} \equiv \mathbf{f}^{(1)} \tag{1.24}$$

$$\text{Element 2:} \quad \begin{Bmatrix} f_1^{(2)} \\ f_2^{(2)} \end{Bmatrix} \begin{matrix} \boxed{2} \\ \boxed{3} \end{matrix} \Rightarrow \begin{Bmatrix} 0 \\ f_1^{(2)} \\ f_2^{(2)} \\ 0 \end{Bmatrix} \begin{matrix} \boxed{1} \\ \boxed{2} \\ \boxed{3} \\ \boxed{4} \end{matrix} \equiv \mathbf{f}^{(2)} \tag{1.25}$$

$$\text{Element 3:} \quad \begin{Bmatrix} f_1^{(3)} \\ f_2^{(3)} \end{Bmatrix} \begin{matrix} \boxed{2} \\ \boxed{3} \end{matrix} \Rightarrow \begin{Bmatrix} 0 \\ f_1^{(3)} \\ f_2^{(3)} \\ 0 \end{Bmatrix} \begin{matrix} \boxed{1} \\ \boxed{2} \\ \boxed{3} \\ \boxed{4} \end{matrix} \equiv \mathbf{f}^{(3)} \tag{1.26}$$

$$\text{Element 4:} \quad \begin{Bmatrix} f_1^{(4)} \\ f_2^{(4)} \end{Bmatrix} \begin{matrix} \boxed{3} \\ \boxed{4} \end{matrix} \Rightarrow \begin{Bmatrix} 0 \\ 0 \\ f_1^{(4)} \\ f_2^{(4)} \end{Bmatrix} \begin{matrix} \boxed{1} \\ \boxed{2} \\ \boxed{3} \\ \boxed{4} \end{matrix} \equiv \mathbf{f}^{(4)} \tag{1.27}$$

Similarly, performing their assembly leads to

$$\mathbf{F}=\sum_{e=1}^{4}\mathbf{f}^{(e)}=\mathbf{f}^{(1)}+\mathbf{f}^{(2)}+\mathbf{f}^{(3)}+\mathbf{f}^{(4)} \tag{1.28}$$

or

$$\mathbf{F} = \begin{Bmatrix} f_1 \\ f_2 \\ f_3 \\ f_4 \end{Bmatrix} = \begin{Bmatrix} f_1^{(1)} \\ f_2^{(1)} + f_1^{(2)} + f_1^{(3)} \\ f_2^{(2)} + f_2^{(3)} + f_1^{(4)} \\ f_2^{(4)} \end{Bmatrix} \tag{1.29}$$

Consistent with the assembly of the global system matrix and the global right-hand-side vector, the vector of unknowns, $\mathbf{u}$, becomes

$$\mathbf{u} = \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{Bmatrix} = \begin{Bmatrix} u_1^{(1)} \\ u_2^{(1)} = u_1^{(2)} = u_1^{(3)} \\ u_2^{(2)} = u_2^{(3)} = u_1^{(4)} \\ u_2^{(4)} \end{Bmatrix} \tag{1.30}$$

1.4.4 Solution of the Global System of Equations

In order for the global system of equations to have a unique solution, the determinant of the global system matrix must be nonzero. However, an examination of the global system matrix reveals that one of its eigenvalues is zero, thus resulting in a zero determinant or singular matrix. Therefore, the solution is not unique. The eigenvector corresponding to the zero eigenvalue represents the translational mode, and the remaining nonzero eigenvalues represent all of the deformation modes.

For the specific values of $k_{11}^{(e)} = k_{22}^{(e)} = k^{(e)}$ and $k_{12}^{(e)} = k_{21}^{(e)} = -k^{(e)}$, the global system matrix becomes

$$\mathbf{K} = k^{(e)} \begin{bmatrix} 1 & -1 & 0 & 0 \\ -1 & 3 & -2 & 0 \\ 0 & -2 & 3 & -1 \\ 0 & 0 & -1 & 1 \end{bmatrix} \tag{1.31}$$

with its eigenvalues $\lambda_1 = 0$, $\lambda_2 = 2$, $\lambda_3 = 3 - \sqrt{5}$, and $\lambda_4 = 3 + \sqrt{5}$. The corresponding eigenvectors are

$$\mathbf{u}^{(1)} = \begin{Bmatrix} 1 \\ 1 \\ 1 \\ 1 \end{Bmatrix}, \quad \mathbf{u}^{(2)} = \begin{Bmatrix} 1 \\ -1 \\ -1 \\ 1 \end{Bmatrix}, \quad \mathbf{u}^{(3)} = \begin{Bmatrix} -1 \\ 2-\sqrt{5} \\ -2+\sqrt{5} \\ 1 \end{Bmatrix}, \quad \mathbf{u}^{(4)} = \begin{Bmatrix} -1 \\ 2+\sqrt{5} \\ -2-\sqrt{5} \\ 1 \end{Bmatrix} \tag{1.32}$$

Each of these eigenvectors represents a possible solution mode. The contribution of each solution mode is illustrated in Fig. 1.8.

In order for the global system of equations to have a unique solution, the global system matrix is rendered nonsingular by eliminating the zero eigenvalue. This is achieved by introducing a boundary condition so as to suppress the translational mode of the solution corresponding to the zero eigenvalue.

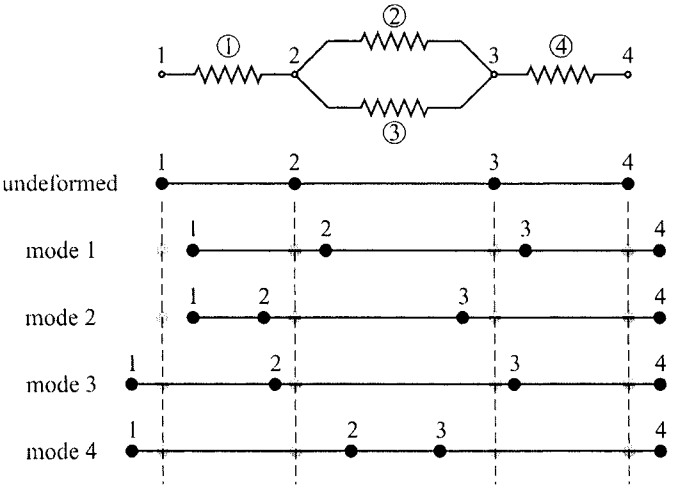

Fig. 1.8 Possible solution modes for the system of linear springs.

1.4.5 Boundary Conditions

As shown in Fig. 1.7, Node 1 is restrained from displacement. This constraint is satisfied by imposing the boundary condition of $u_1 = 0$. Either the nodal displacements, u_i, or the nodal forces, f_i, can be specified at a given node. It is physically impossible to specify both of them as known or as unknown. Therefore, the nodal force f_1 remains as one of the unknowns. The nodal displacements, u_2, u_3, and u_4 are treated as unknowns, and the corresponding nodal forces have values of $f_2 = 0$, $f_3 = 0$, and $f_4 = F$.

These specified values are invoked into the global system of equations as

$$k^{(e)}\begin{bmatrix} 1 & -1 & 0 & 0 \\ -1 & 3 & -2 & 0 \\ 0 & -2 & 3 & -1 \\ 0 & 0 & -1 & 1 \end{bmatrix}\begin{Bmatrix} u_1 = 0 \\ u_2 \\ u_3 \\ u_4 \end{Bmatrix} = \begin{Bmatrix} f_1 \\ f_2 = 0 \\ f_3 = 0 \\ f_4 = F \end{Bmatrix} \tag{1.33}$$

leading to the following equations:

$$k^{(e)}\begin{bmatrix} 3 & -2 & 0 \\ -2 & 3 & -1 \\ 0 & -1 & 1 \end{bmatrix}\begin{Bmatrix} u_2 \\ u_3 \\ u_4 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ F \end{Bmatrix} \tag{1.34}$$

and

$$-k^{(e)}u_2 = f_1 \tag{1.35}$$

The coefficient matrix in Eq. (1.34) is no longer singular, and the solutions to these equations are obtained as

$$u_2 = \frac{F}{k^{(e)}}, \quad u_3 = \frac{3}{2}\frac{F}{k^{(e)}}, \quad u_4 = \frac{5}{2}\frac{F}{k^{(e)}} \tag{1.36}$$

and the unknown nodal force f_1 is determined as $f_1 = -F$. The final physically acceptable solution mode is shown in Fig. 1.9.

There exist systematic approaches to assemble the global coefficient matrix while invoking the specified nodal values (Bathe and Wilson 1976; Bathe 1996). The specified nodal variables are eliminated in advance from the global system of equations prior to the solution.

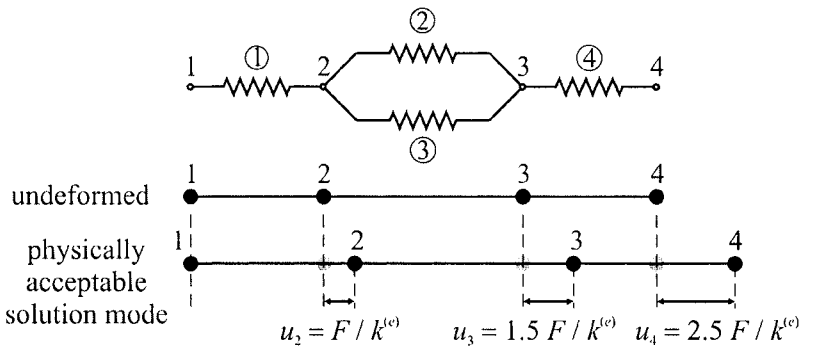

Fig. 1.9 Physically acceptable solution mode for the system of linear springs.

Chapter 2

FUNDAMENTALS OF ANSYS

2.1 Useful Definitions

Before delving into the details of the procedures related to the ANSYS program, we define the following terms:

Jobname: A specific name to be used for the files created during an ANSYS session. This name can be assigned either before or after starting the ANSYS program.

***Working Directory*:** A specific folder (directory) for ANSYS to store all of the files created during a session. It is possible to specify the *Working Directory* before or after starting ANSYS.

***Interactive Mode*:** This is the most common mode of interaction between the user and the ANSYS program. It involves activation of a platform called *Graphical User Interface* (*GUI*), which is composed of menus, dialog boxes, push-buttons, and different windows. *Interactive Mode* is the recommended mode for beginner ANSYS users as it provides an excellent platform for learning. It is also highly effective for postprocessing.

***Batch Mode*:** This is a method to use the ANSYS program without activating the *GUI*. It involves an *Input File* written in *ANSYS Parametric Design Language* (*APDL*), which allows the use of parameters and common programming features such as *DO* loops and *IF* statements. These capabilities make the *Batch Mode* a very powerful analysis tool. Another distinct advantage of the *Batch Mode* is realized when there is an error/mistake in the model generation. This type of problem can be fixed by modifying a small portion of the *Input File* and reading it again, saving the user a great deal of time.

***Combined Mode*:** This is a combination of the *Interactive* and *Batch Modes* in which the user activates the *GUI* and reads the *Input File*. Typically, this method allows the user to generate the model and obtain the solution using the *Input File* while reviewing the results using the

Postprocessor within the *GUI*. This method combines the salient advantages of the *Interactive* and *Batch Modes*.

2.2 Before an ANSYS Session

The construction of solutions to engineering problems using FEA requires either the development of a computer program based on the FEA formulation or the use of a commercially available general-purpose FEA program such as ANSYS. The ANSYS program is a powerful, multi-purpose analysis tool that can be used in a wide variety of engineering disciplines. Before using ANSYS to generate an FEA model of a physical system, the following questions should be answered based on engineering judgment and observations:

- What are the objectives of this analysis?
- Should the entire physical system be modeled, or just a portion?
- How much detail should be included in the model?
- How refined should the finite element mesh be?

In answering such questions, the computational expense should be balanced against the accuracy of the results. Therefore, the ANSYS finite element program can be employed in a correct and efficient way after considering the following:

- Type of problem.
- Time dependence.
- Nonlinearity.
- Modeling idealizations/simplifications.

Each of these topics is discussed in this section.

2.2.1 Analysis Discipline

The ANSYS program is capable of simulating problems in a wide range of engineering disciplines. However, this book focuses on the following disciplines:

Structural Analysis: Deformation, stress, and strain fields, as well as reaction forces in a solid body.

Thermal Analysis: Steady-state or time-dependent temperature field and heat flux in a solid body.

2.2.1.1 Structural Analysis

This analysis type addresses several different structural problems, for example: